Hợp tác Đức
DEUTSCHE ZUSAMMENARBEIT

Funding programme
develoPPP
Where business meets development.

Implemented by
giz Deutsche Gesellschaft für Internationale Zusammenarbeit (GIZ) GmbH

SIEMENS

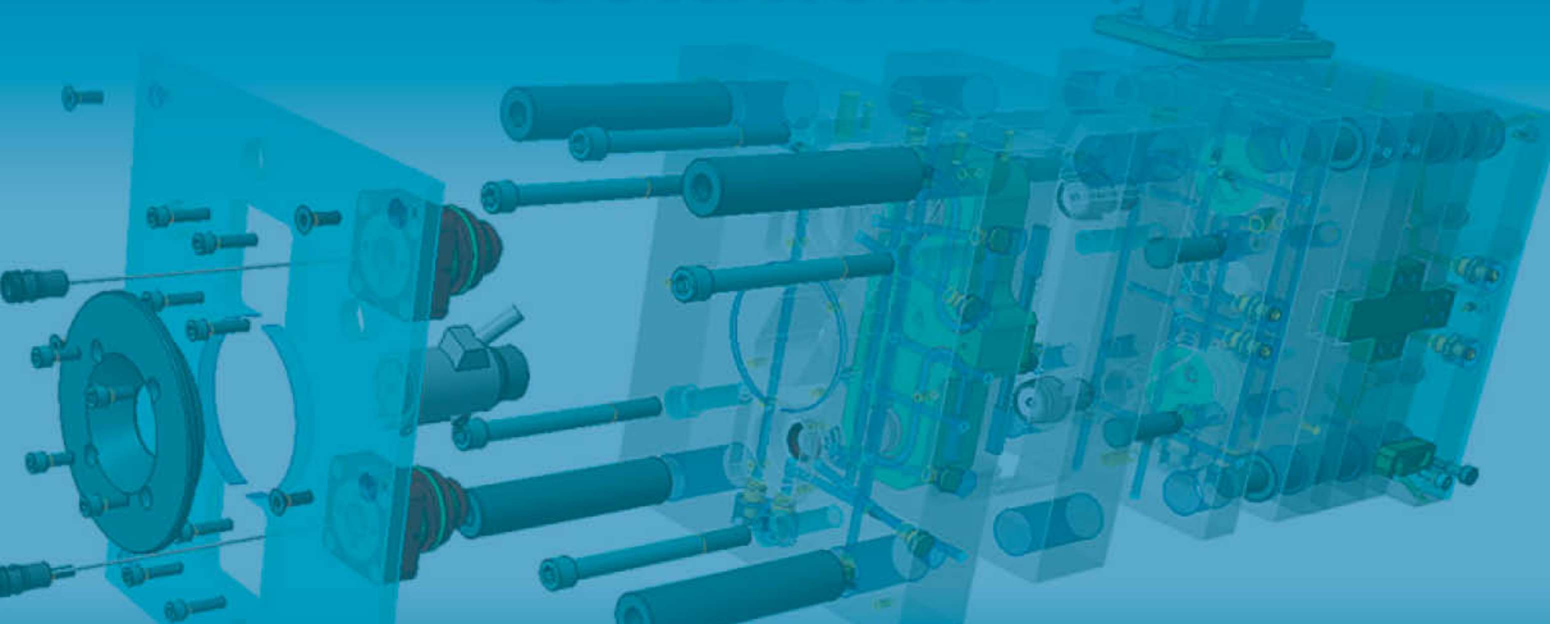
LILAMA2 COLLEGE

Development Partnership with the Private Sector

Vocational Training for Smart Manufacturing in Machine Tools

Training Module

Advanced Mold and Die Design with Integrated CAD/CAM/CAE Solutions



develoPPP.de programme of the German Federal Ministry for Economic Cooperation and Development

Hợp tác Đức
DEUTSCHE ZUSAMMENARBEIT

Funding programme
develoPPP
Where business meets development.

Implemented by
giz Deutsche Gesellschaft für Internationale Zusammenarbeit (GIZ) GmbH

SIEMENS

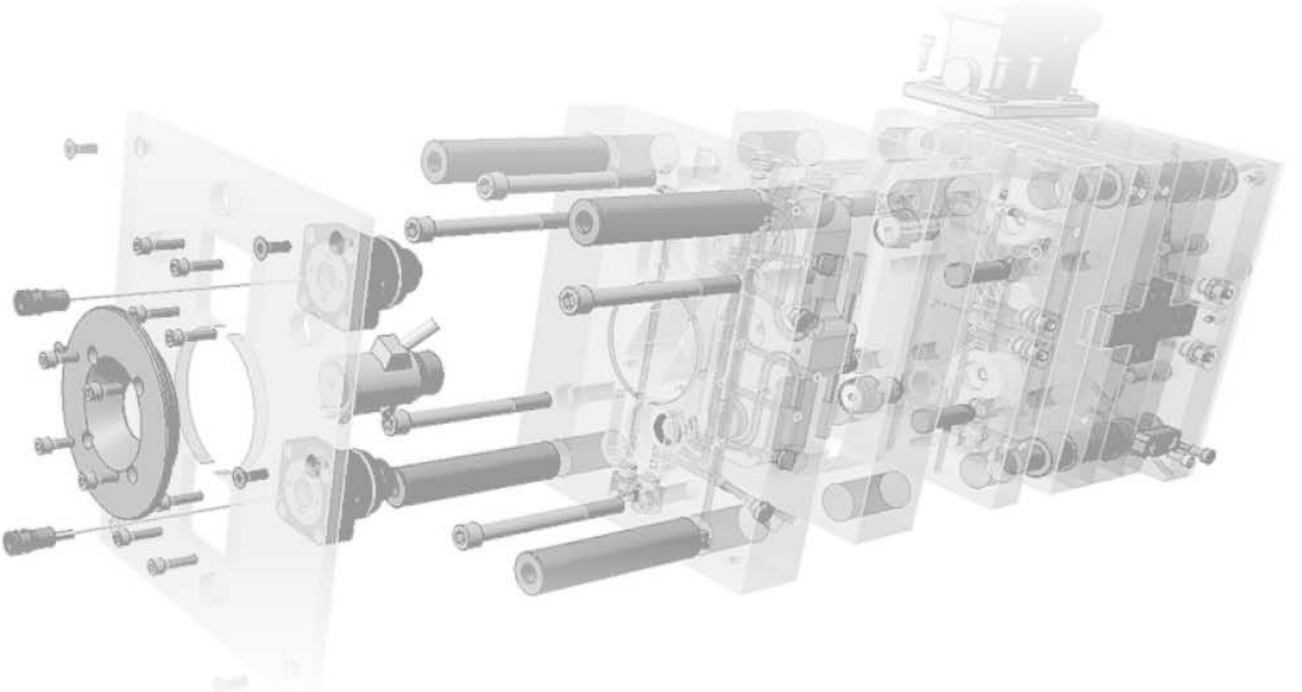
LILAMA2 COLLEGE

Development Partnership with the Private Sector

Vocational Training for Smart Manufacturing in Machine Tools

Siemens Vietnam - GIZ - LILAMA 2

Training Module
Advanced Mold and Die Design with Integrated CAD/CAM/CAE Solutions



Development Partnership with the Private Sector
Vocational Training for Smart Manufacturing in Machine Tools
Siemens Vietnam - GIZ - LILAMA 2

Training Module
Advanced Mold and Die Design with Integrated CAD/CAM/CAE Solutions
1st Edition

Training Materials – Exercises – Project Works – Solutions

Editorial Board

Nguyen Xuan Huy, Nguyen Trong Luc
Phan Thi Anh Tu, Le Tuyen Giao, Nguyen Hong Tien

Contributions

LILAMA 2 International Technology College
Mechanical Engineering Faculty
Training Department
Quality Assurance Department

GIZ Programme Reform of TVET in Viet Nam
Ralf Hill
Pham Thi Thanh Truc
Nguyen Minh Cong
Tran Thi Bich Tuyen

The book ***Training Module Advanced Mold and Die Design with Integrated CAD/CAM/CAE Solutions*** *is officially published by*
the cooperation partners of the development partnership
"Vocational Training for Smart Manufacturing in Machine Tools".

It has been dissemniated to eleven Institutes of the ***Vietnamese-German Programme Reform of Technical and Vocational Education and Training (TVET) in Viet Nam****, and other relevant stakeholders in TVET and in manufacturing industry.*

Address: *#1, Lane 17, Ta Quang Buu Street, Bach Khoa Ward,*
Hai Ba Trung District, Hanoi, Viet Nam
Tel: *+84 (0) 24 39 74 64 71*
Fax: *+84 (0) 24 39 74 65 70*
Website: *www.tvet-vietnam.org*
www.giz.de/vietnam

FOREWORD

Dear Industry 4.0 stakeholders

New competence requirements due to digitalisation and Industry 4.0 (I 4.0) are arising in the Vietnamese manufacturing industries, which need to be addressed by Technical and Vocational Education and Training (TVET) institutes in close cooperation with the business sector. TVET plays a crucial role in the development of a skilled and competent workforce for those advanced industries.

With support of the Vietnamese Directorate of Vocational Education and Training (DVET) under the Ministry of Labour, Invalids and Social Affairs (MoLISA), the partners Siemens Viet Nam, LILAMA 2 International Technology College (LILAMA 2) and the Deutsche Gesellschaft für Internationale Zusammenarbeit (GIZ), established the development partnership **"Vocational Training for Smart Manufacturing in Machine Tools".** Development partnerships with the private sector promote activities where entrepreneurial opportunities and development policy potential meet and are supported by the ***www.develoPPP.de*** programme of the German Federal Ministry for Economic Cooperation and Development (BMZ).

The main achievements of the partners comprised:

- "Digital Technology Education Centre" for smart manufacturing at LILAMA 2 established;
- Inclusive training modules and teaching and learning materials on higher digitalization and I 4.0 competencies based on the industry needs developed;
- Technical and didactical competencies of TVET teachers and in-company trainers for smart manufacturing for metal processing occupations improved;
- Teachers of LILAMA 2 and other partner TVET institutes of the Vietnamese-German *Programme Reform of TVET in Viet Nam* have been trained as master trainers and multipliers for further dissemination in the TVET system;
- Gender equality promoted and awareness about the employment potential of people of disadvantaged groups and their career options in industrial occupations and in TVET raised.

National and international smart manufacturing experts and curriculum designers developed the training module **"Advanced Mold and Die Design with Integrated CAD/CAM/CAE Solutions in NX".** The newly developed training module provides additional qualification in sophisticated smart manufacturing application as in high demand of national and international companies in Viet Nam. It is complying with Circular 03/2017 of the MoLISA and oriented on German standard.

The module can be offered as an elective training module for college level graduates of the three-years training programme *Metal Cutting Technician* but also as further training to industry technicians and engineers in the field of mechanical engineering and machine tools. TVET teachers are recommended to offer the training module as hybrid learning or blended learning, in which trainees learn via electronic and online media, as well as traditional face-to-face teaching and training.

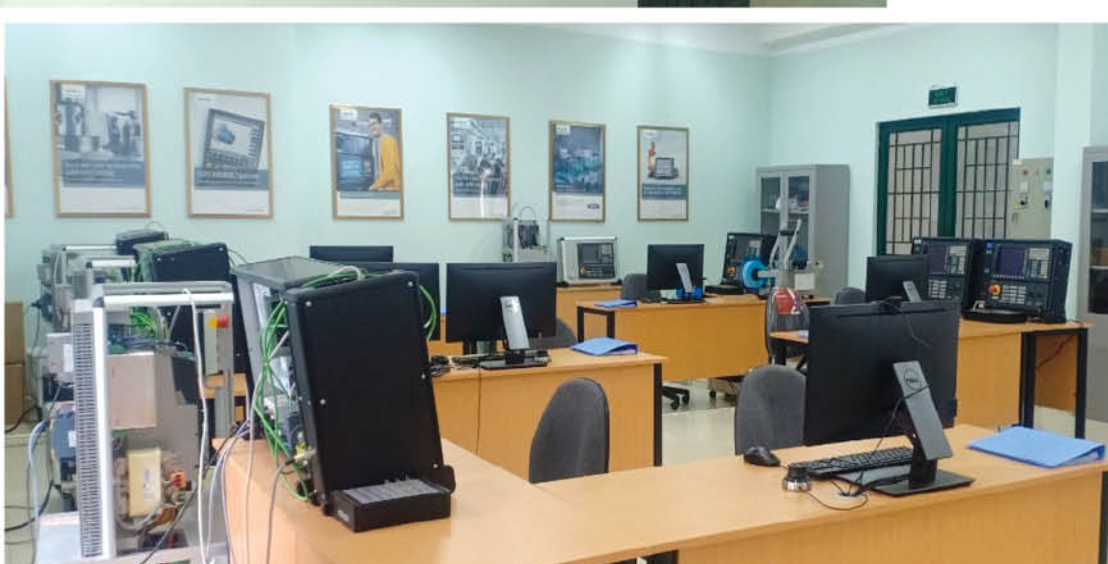
Initial and further training programmes, developed in the frame of the Vietnamese-German *Programme Reform of TVET in Viet Nam*, are available for free download at ***www.tvet-vietnam.org.***

On behalf of the partners, we would like to express our utmost gratitude to all parties involved in the development of the training module "Advanced Mold and Die Design with Integrated CAD/CAM/CAE Solutions in NX". We wish all I 4.0 and TVET stakeholders a successful application of the module to further enhance relevant workforce competencies.

Beate Dippmar
GIZ Programme Manager
Programme Reform of TVET in Viet Nam

Tran The Hien
Head of Digital Industries
SIEMENS Viet Nam

Nguyen Khanh Cuong
Rector, LILAMA 2 International
Technology College, Viet Nam



ABBREVIATIONS

2D	Two-Dimensional
3D	Three-Dimensional
ABS	Absolute Coordinate System
ANSI	American National Standards Institute
API	Application Programming Interfaces
ASC	Association for Survey Computing
ASME	The American Society of Mechanical Engineers
BMZ	Federal Ministry for Economic Cooperation and Development
CAD	Computer-aided Design
CAM	Computer-Aided Manufacturing
CNC	Computer Numerical Control
CSYS	Absolute Coordinate System
DIN	Deutsches Institut für Normung e.V. (German Institute for Standardization)
DPP	Development Partnerships with the Private Sector
DVET	Directorate of Vocational Education and Training
DWG	From drawing (.dwg format file)
DXF	Drawing Interchange Format
ESKD	ESKD Data Standards
FCF	Feature Control Frame
FPCD	Flexible Printed Circuit Design
GB	Guojia Biaozhun (International Standard)
GD&T	Geometric dimensioning and tolerancing
GIZ	Deutsche Gesellschaft für Internationale Zusammenarbeit (GIZ) GmbH
I.4.0	Industrial 4.0
IGES	Initial Graphics Exchange Specification
ISO	International Standardization Organization
JIS	Japan Industrial Standard
LILAMA 2	LILAMA 2 International Technology College
MoLISA	Ministry of Labour, Invalids and Social Affairs
MRO	Overhaul processes
NX	Officially NX does not stand for anything but the name came from the merging of technology from SDRC I-DEAS with UGS Unigraphics to deliver the "Next Generation" CAD system
PMI	Product Manufacturing Information
PPO	Proximal Policy Optimization
PS	Sets
SCA	Software Composition Analysis
TVET	Technical Vocational Education and Training
WCS	Workpiece Coordinate System

TABLE OF CONTENTS

TABLE OF CONTENTS

A. TRAINING MODULE

(According to Circular No. 03/2017/TT-BLĐTBXH March 1, 2017 of the Ministry of Labour, Invalids and Social Affairs)

Module name:

Advanced Mold and Die Design with Integrated CAD/CAM/CAE Solutions

Module code: MD02

Module duration: 320 hours (Contents: 76h; Practice, experiment, discussion, assignments: 228h; Examinations / Assessments: 16h)

I. Module classification and characteristics:

- **Classification:**
  Training module for AdvaTCed Mold and Die Design with Integrated CAD/CAM/CAE Solutions qualification – International College Level. Enrollment requirement: National College Diploma in Metal Cutting.
- **Characteristics:**
  - The module is set to follow the German training and examination standards. It provides training topics and competencies on Mold and Die Design, as well as advanced training in Metal cutting. The module is structured into 5 training units which upscale and continue one another. This can be offered as a supplementary training module for college graduates of initial training programmes as well as to the industry in hybrid short-term form of training for technicians, engineers and in-company trainers
  - You can design the complete product shape needed with NX
  - You can simply import data files from other software without abundant modification, similar to products designed directly on NX
  - You can check the machinability of mold products
  - Advanced separating tools specifically designed by NX are available
  - Core and cavity generation are fully automatic
  - NX can check against new product versions created later automatically
  - List of standard parts from mold manufacturers is provided
  - Customized tools for push pins, oblique cores, slide cores are available
  - You can assign cooling system, pipes and linking valves
  - NX can create and extract drawings on demand automatically.
  - Learners are aware of and comply to regulations on IT Security, labour safety, as well as health and environmental protection.

II. Module objectives:

By the end of this module, the trainees will be able to:

- **Knowledge:**
  - Understand detailed models, the process of creating 3D drawings
  - Understand drawing and modifying commands with NX software to create 3D product models
  - Understand the sheet metal model, the process of creating products from sheet metal
  - Understand the material standards applied in sheet metal
  - Understand drawing and modifying commands on NX software to create sheet metal product models
  - Understand the overview of mold design and manufacturing technology
  - Understand the structure and classification of plastic injection molds
  - Understand the process of designing and manufacturing plastic injection molds
  - Understand the design standards of plastic injection molds
  - Present the concept of sheet presswork technology and classify the types of this technology
  - Analyze the functions of the components that make up a common press die
  - Explain the use of support commands of the PDW tool/module
  - Read and understand simulation and analysis results
  - Analyze the causes and find ways to fix design errors
  - Understand the process of simulating plastic flow with NX software.
  - Apply benefits of integrated programme measurement to reduce the time for quality functions to be establishes and ensured.
- **Skills:**
  - Draw products and 3D models in simple and complex forms with NX software
  - Export 2D technical drawings for 3D models with NX software
  - Assemble subassembly models
  - Draw simple and complex sheet metal products and models with NX software
  - Create technical drawing from sheet metal with NX software
  - Calculate the technical parameters of plastic injection molds
  - Perform automatic mold design steps with NX software
  - Implement the simulation process to analyze and optimize the mold design process with NX software
  - Implement the management of mold data after design (BOM)
  - Export drawings to manufacture mold details
  - Use the commands of the PDW tool/module to design a complete set of progressive press die
  - Solve application exercises with the help of the PDW tool/module
  - Export the report of simulation results of plastic injection mold with NX software.
  - Plan and implement the distribution of complicated industrial tasks
- **Autonomy and responsibility:** the trainees are able to work independently and in teams according to the work assignment, to

- Determine work steps and processes according to functional, organizational and manufacturing criteria
- Design workflow diagrams for custom tasks according to the order
- Research and evaluate information for order planning in digital networks
- Carry out complex manufacturing processes in compliance with regulations on IT security and occupational safety as well as environmental protection
- Arrange, monitor and control sub orders
- Develop special task solutions in terms of customer requests

III. Module contents:

1. General contents and time allocation:

No	Training module units	Time allocation (hours)			
		Total (hours)	Contents	Practice, experiment, discussion, assignment	Examination/ Evaluation
1	**1. Fundamentals of 2D & 3D Design and Assembly** 1.1. Overview of NX 1.2. 2D, 3D design with NX 1.3. Exporting 2D drawings with NX 1.4. Assembling sub-assemblies 1.5. Practice exercises	80	24	54	2
2	**2. Advanced Sheet Metal Design with NX: Tools and Techniques** 2.1. General sheet metal design with NX 2.2. Material standards and features of sheet metal 2.3. Basic tools for sheet metal design and relevant geometrical elements 2.4. Advanced tools for sheet metal design and modification tools 2.5. Creating technical drawings of sheet metal with NX 2.6. Practice exercises	40	12	26	2

	3. Design Plastic Injection Molds with NX software: From Concept to Detailing 3.1. Overview of NX mold 3.2. Creation of mold design project 3.3. Definition of drawing direction of mold, workpiece, etc. 3.4. Definition of core and cavity 3.5. Definition of partition face 3.6. Flexible Mold Separator 3.7. Insert of base mold parts 3.8. Design of plastic channel system, cooling channel, etc. 3.9. Excluding mold parts 3.10. Creation of 2D drawings for mold details 3.11. Practice exercises	80	20	56	4
4	**4. Design the press Dies with NX Progressive Die Wizard: Streamlining Workflow** 4.1. Overview of press die 4.2. Overview of NX PDW (Progressive Die Wizard) 4.3. Design of Punch and Die 4.4. Design of the base mold structure 4.5 Practice exercises	80	8	70	2
5	**5. Integrated NX Easy Fill: Plastic injection mold design through simulation** 5.1. Overview of NX Easy Fill 5.2. Product analysis process with NX Easy Fill Example exercises.	40	12	22	6
	Total:	**320**	**76**	**228**	**16**

2. Deatailed contents:

Training Unit 1: Fundamentals of 2D & 3D Design and Assembly

Duration: 40 hours

1. **Objective:** By the end of this training unit, the trainees will be able to:

- Understand detailed models, the process of creating 3D drawings
- Understand drawing and editing commands with NX software to create 3D product models
- Draw products, 3D models from simple to complex forms with NX software
- Export 2D technical drawings for 3D models with NX software
- Assemble the subassemblies

2. **Contents:**

2.1. NX software overview

2.1.1. Software overview

2.1.2. Overview of the interface and functions, toolbar software NX

2.1.3. Process for creating 3D parts

2.2. 2D design with NX

2.2.1. Basic commands in 2D part design

2.2.2. Correction commands in 2D part design

2.2.3. Measuring tools in 2D design

2.3. 3D design with NX

2.3.1. Convert 3D drawings from 2D

2.3.2. Basic commands in 3D part design

2.3.3. Advanced commands in 3D part design

2.4. Export 2Ddrawings from 3D drawings with NX

2.4.1. International Drawing Standards

2.4.2. Editing information on the drawing frame

2.4.3. Selection of the projection method for objects

2.4.4. Measurement of parts for required size outputs

2.4.5. Advanced tools in 2D drawing export

2.4.6. Checking drawings and printing drawings

2.5. Practice exercises

Training Unit 2: Advanced Sheet Metal Design with NX: Tools and Techniques

Duration: 40 hours

1. **Objective:** By the end of this training unit, the trainees will be able to:

- Understand the sheet metal model, the process of creating products from sheet metal
- Understand the material standards applied in sheet metal
- Understand drawing and modifying commands with NX software to create sheet metal product models
- Draw simple and complex sheet metal product models with NX software
- Create technical drawing from sheet metal with NX software

2. **Contents:**

2.1. Getting started with Sheet Metal with NX software

2.1.1. Sheet metal

2.1.2. Recommended workflow to create a simple part

2.1.3. Recommended workflow to create enclosures with odd shapes or angles

2.2. Sheet metal material standards and features

2.2.1. Sheet metal material standards

2.2.2. Sheet metal features

2.3. Designing tools for sheet metal parts and their features

2.3.1. Tools for transforming block metal into sheet metal

2.3.2. Basic tools for designing sheet metal

2.3.3. Tools for fabrication of sheet metal

2.4. Advanced commands for sheet metal and modification commands

2.4.1. Advanced commands for sheet metal

2.4.2. Validating sheet metal parts

2.4.3. Patterning in sheet metal

2.4.4. Mirroring in sheet metal

2.4.5. Updating object features

2.4.6. Weld preparations on sheet metal parts

2.5. Creating technical drawing for sheet metal with NX software

2.6. Practice exercises.

Training Unit 3: Design of Plastic Injection Molds with NX: From Concept to Detailing Duration: 80 hours

1. **Objectives:** By the end of this training unit, the trainees will be able to:
   - Understand the overview of mold design and manufacturing technology
   - Understand the structure and classification of plastic injection molds
   - Understand the process of designing and manufacturing plastic injection molds
   - Understand the design standards of plastic injection molds
   - Calculate the technical parameters of plastic injection molds
   - Perform automatic mold design steps with NX software
   - Implement the simulation process to analyze and optimize the mold design process with NX software
   - Implement the management of mold data after design (BOM)
   - Export drawings to manufacture mold details.
2. **Contents:**

2.1. Overview of NX mold

2.1.1. Two-plate mold

2.1.2. Three-plate mold

2.2. Creating a mold design project

2.2.1. Product Analysis

2.2.2.Entering shrinkage factor

2.2.3.Setting the order information

2.3. Definition of withdrawal direction.

2.3.1.Definition of coordinate system

2.3.2.Definition of workpiece

2.3.3.Arranging the quantity of products

2.4. Definition of core and cavity

2.4.1.Selecting the withdrawal direction

2.4.2.Defining Core and Cavity areas

2.4.3.Patching holes

2.5. Partition face definition

2.5.1.Partition line setting

2.5.2.Partition face setting

2.5.3.Creating core and cavity

2.6. Flexible Mold Separator

2.6.1.Automatic parting line setting

2.6.2.Automatic parting face setting

2.6.3.Creating core and cavity automatically.

2.7. Inserting base mold parts

2.7.1.Inserting basic mold components

2.7.2.Inserting bushing and nozzle

2.8. System design

2.8.1.Design of plastic channel system

2.8.2.Design of propulsion system

2.8.3.Design of cooling system

2.8.4.Inserting some extra details

2.9. Subtract the parts in the mold

2.9.1.Checking geometrical intersection details

2.9.2.Subtracting blocks to create complete parts

2.10. Material management (BOM)

2.11. Creating 2D drawings for mold details

2.12. Practice exercises.

Training Unit 4: Design of Press Dies with NX Progressive Die Wizard: Streamlining Workflow

Duration: 80 hours

1. **Objective:** By the end of this training unit, the trainees will be able to
   - Present the concept of sheet presswork technology and classify different types of this technology
   - Analyze the functions of the components that make up a common press die

- Explain the use of support commands of the PDW tool/module
- Use the commands of the PDW tool/module to design a complete set of progressive press die
- Solve application exercises with the help of the PDW tool/module.

2. Contents:

2.1. Overview of press mold

2.1.1.Pressing and Stamping technology

2.1.2.Structure of progressive press die

2.2. Overview of NX PDW

2.2.1.Introduction

2.2.2.Related Commands

2.3. Design of punch and die structure

2.3.1.Design of punch structure

2.3.2.Design of die structure

2.4. Design of base mold structure

2.4.1.Designing the major components

2.4.2.Designing the minor components

2.5. Practice exercises

2.5.1.Practice Exercise No. 1

2.5.2.Practice Exercise No. 2

Training Unit 5: Integrated NX Easy Fill: Plastic Injection Mold Design through Simulation

Duration: 40 hours

1. Objective: By the end of this training unit, the trainees will be able to

- Read and understand simulation and analysis results
- Analyze the causes and find ways to fix design errors
- Understand the process of simulating plastic flow with NX software
- Export the report of simulation results of plastic injection mold with NX software.

2. Contents:

2.1. Overview of NX Easy Fill

2.1.1.The influence of materials on the process of forming plastic products

2.1.2.The influence of pressing parameters on the formation of plastic products

2.1.3.The influence from the texture and shape of plastic products on the forming process

2.1.4.NX Easy Fill as a solution to overcome the above factors to make plastic products easier and more accurate

2.2. Product analysis process with NX Easy Fill

2.2.1.Informing simulation prediction (Set Working, Set Cavity, Set Masterial)

2.2.2.Informing the injection port (Set Runner, Set Melt Entrance)

2.2.3.Informing the withdrawal direction (Set Parting Direction)

2.2.4.Exporting the analysis command (Start Analysis)

2.2.5.Exporting the analysis result report (Show Result)

2.3 Example exercises.

IV. Requirements for module implementation

1. Specialised classroom, training workshop

a. Specialised classroom: Computer room

2. Equipment and machinery

- Computer
- Projector
- Printer

3. Teaching and learning materials, tools, consumption materials

- Detailed drawings, assemblies and overall drawings
- Assembly instructions, maintenance and reparation plans
- Plan for machining, arrangement and work
- Table of safety data
- Table of characteristic values, measurement checklist, assessment record
- Text books, reference manuals
- Pocket calculator, drawing materials.
- NX software
- Handbooks
- Handouts
- Exercise system.

4. Other requirements and conditions: None

V. Assessment contents and methodology

1. Assessment contents:

- **Knowledge:**
  - Understand drawing and modifying commands with NX software to create 3D product models
  - Understand drawing commands and modifying commands with NX software to create sheet metal product models
  - Understand the overview of mold and die design and manufacturing technology
  - Understand the structure and classification of plastic injection molds
  - Understand the process of designing and manufacturing plastic injection molds
  - Understand the design standards of plastic injection molds
  - Present the concept of pressing technology and classify the types of this technology
  - Analyze the functions of the components that make up a common press die

- Explain the use of support commands of the PDW tool/module
- Read and understand simulation and analysis results
- Analyze the causes and find ways to fix design errors
- Understand the process of simulating plastic flow with NX software.

- **Skills:**
  - Draw products, 3D models in simple and complex forms with NX software
  - Export 2D technical drawings for 3D models with NX software
  - Assemble the subassemblies
  - Draw simple and complex sheet metal product models with NX software
  - Create technical drawing from sheet metal with NX software
  - Calculate the technical parameters of plastic injection molds.
  - Perform automatic mold design steps with NX software
  - Implement the simulation process to analyze and optimize the mold design process with NX software.
  - Implement the management of mold data after design (BOM).
  - Export drawings to manufacture mold details.
  - Analyze the functions of the components that make up a common press die
  - Explain the use of support commands of the PDW tool/module
  - Use the commands of the PDW tool/module to design a complete set of progressive press die
  - Solve application exercises with the help of the PDW tool/module.
  - Export the report of simulation results of plastic injection mold with NX software.

- **Autonomy and responsibility:** the trainees are able to work independently and in teams according to the work assignment, to
  - Determine work steps and processes according to functional, organizational and manufacturing criteria
  - Design workflow diagrams for custom tasks according to the order
  - Research and evaluate information for order planning in digital networks
  - Carry out complex manufacturing processes in compliance with regulations on IT security and occupational safety as well as environmental protection
  - Arrange, monitor and control sub-orders
  - Develop customized task solutions required by customers

2. Assessment methods:

The assessment is based on the project work carried out and products manufactured by the trainee/learner and is carried out in accordance with the provisions on the minimum knowledge, skills, autonomy and responsibility required for module.

- **Knowledge:**

The trainee's/learner's knowledge, skills and attitudes are determined based on oral and written tests such as quizzes, technical discussions, and multiple-choice questions, as well as through exercises integrating both theoretical and practical contents or practical exercises during implementing the teaching units of the module. The evaluations are calculated according to the valid point rules.

- **Skills:**

The trainee's/learner's practice results are evaluated based on practiceexercises, project work and actual tasks at workshops via evaluation sheet / scales in accordance with the following criteria:

- Labour safety
- Organization of the workplace
- Technical standards
- Planning and implementation
- Target time
- Self – assessment.

- **Autonomy and responsibility:**

Based on attitudes and characters of the trainees/learners that are determined and evaluated through observation over the training period: ethics in working, learning and cooperating, regulation and morale, diligence, conscientiousness, disciplines, teamwork ability, punctuality, independence, sense of responsibility, prudence, initiative, activeness, enthusiastic participation in lessons and supporting/motivating others during the learning process.

VI. Guidelines for module implementation:

1. Scope of application

Training module Advanced Mold and Die Design with Integrated CAD/CAM/CAE Solutions – International College Level.

2. Guidelines on module teaching and learning methods

- For teachers, lecturers and in-company trainers: Teachers (in TVET institutes/in-company trainers) are responsible for compliance with the following guidelines for the professional implementation of the theoretical and practical training contents of the enire module as well as each single training unit:
  - The trainees/learners must be instructed in details in compliance with regulations on labour safety, health and environmental protections as well as fire prevention & protection and security. Such compliance must be continuously monitored by responsible teacher/ in-company trainer. The trainee/learner must be informed with clarity and aware of the appropriate measures and consequences for not complying with regulations.
  - The learning process and progress of trainees/learners shall be regularly monitored and evaluated, in particular the consistent compliance with labour safety regulations and environmental protection conditions.

- Ensure the highest possible quality of teaching and training by providing references to the corresponding teaching unitsin planning and implementing lessons.
- Within the framework of the practice training units, the necessary working steps shall be carefully explained to trainees/learners and demonstrated correctly.
- Individual knowledge and skill level shall be checked and assessed individually on each actual teaching unit based on periodical work reports drawn up by the trainee.
- The quality of teaching is enhanced and guaranteed by the increase in using different teaching and learning methods such as the 4-step method, project method, guiding text, self-study and group work as well as by the efficient use of teaching and learning materials and other aids.
- The work results of the trainees/learners must be evaluated and discussed transparently between trainees/learners, responsible vocational teacher and in-company trainer.

- For trainees/learners: The trainees/learners are instructed to
  - Follow instructions of vocational teachers or in-company trainers disciplinarily.
  - Participate regularly and actively in class and each training lesson of the training module.
  - Comply with the regulations regarding labour safety, health, fire prevention & protection, and environmental protection.
  - Contribute actively to environmental protection.
  - Comply with teaching and workshop regulations.
  - Participate in class attentively, take notes and ask questions in case of uncertainty.
  - Raise questions to vocational teachers or in-company trainers or other trainees/learners to ask for support with difficult tasks and to address problems
  - Set up and keep the workplace clean and tidy
  - Prepare, handle and maintain equipment properly
  - Make daily and weekly reports on theoretical and practical lessons of the training modules that learners have attended.

3. Focuses

The training focuses of the training module are units: 1, 2, 3, 4, and 5.

4. References

- Metal Engineering, Young Publishing House, Vietnam
- Mechanical and Metal Trades Handbook (Tabellen Buch Metall), 3rd English Edition, EUROPA – LEHRMITTEL, Germany
- Training materials related to SINUMERIK Operate 840Dsl, Siemens Corp., 2019
- NX Documents from Siemens.

5. Further notes and explanations (if any)

B. TRAINING CONTENTS

UNIT 1: FUNDAMENTALS OF 2D & 3D DESIGN AND ASSEMBLY

Objective: By the end of this training unit, the trainees will be able to:

- Understand detailed models, the process of creating 3D drawings
- Understand drawing and editing commands with NX software to create 3D product models
- Draw products, 3D models from simple to complex ones with NX software
- Export 2D technical drawings for 3D models with NX software
- Assemble the subassemblies.

Contents:

1.1. NX software overview

1.1.1 Software overview

Figure 1.1 NX overview

1.1.2 Overview of the interface and functions, toolbar software NX

NX interface

The NX Windows interface provides access to frequently used commands with a minimum number of mouse clicks while maintaining a maximum graphics window area.

1.1.1.1 NX main frame window

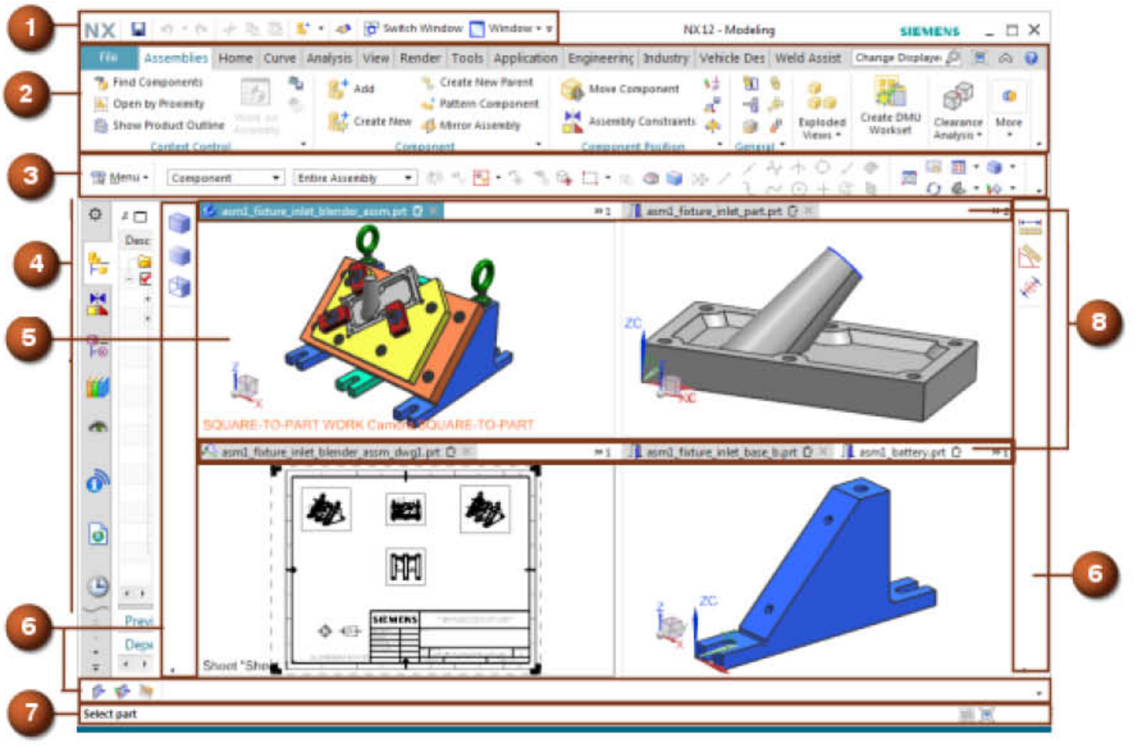
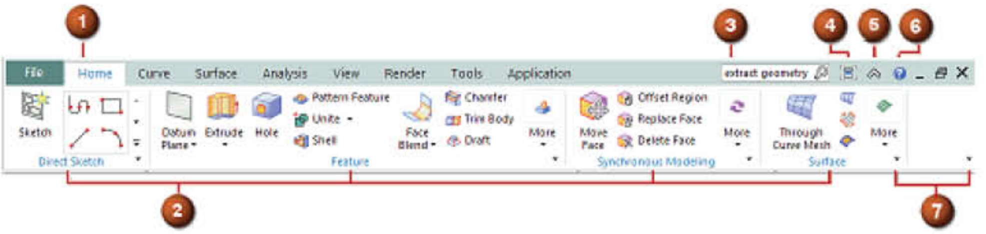


Figure 1.2 NX Windows interface

#	Component	Description
1	Quick Access toolbar	Contains commonly used commands such as **Save** and **Undo**.
2	Ribbon bar	Organizes commands in each application into tabs and groups.
3	Top Border bar	Contains the **Menu**, **Selection Group**, **View Group**, and **Utility Group** commands.
4	Resource bar	Contains navigators and palettes, including the **Part Navigator** and the **Roles** tab.
5	Graphics window	Allows modeling, visualizing, and analyzing models.
6	Left, Right, and Bottom Border bars	Displays the additional commands.
7	Cue/Status line	Prompts you for the next action and displays messages.
8	Tab area	Displays the name of the part files open in the tabbed window.

1.1.1.2 NX Ribbon bar



#	Component	Description
1	Tab	Organizes commands into groups of related functions in each application.
2	Group	Organizes commands by function on each tab. Related commands appear in lists and galleries.
3	Command Finder	Finds commands.
4	Full Screen	Maximizes screen space.
5	Minimize Ribbon	Collapses the groups on the Ribbon tab.
6	Help	Displays on-context Help (F1).
7	Ribbon Options	Allows turning on or turning off commands in each group.

1.1.1.3 NX floating frame

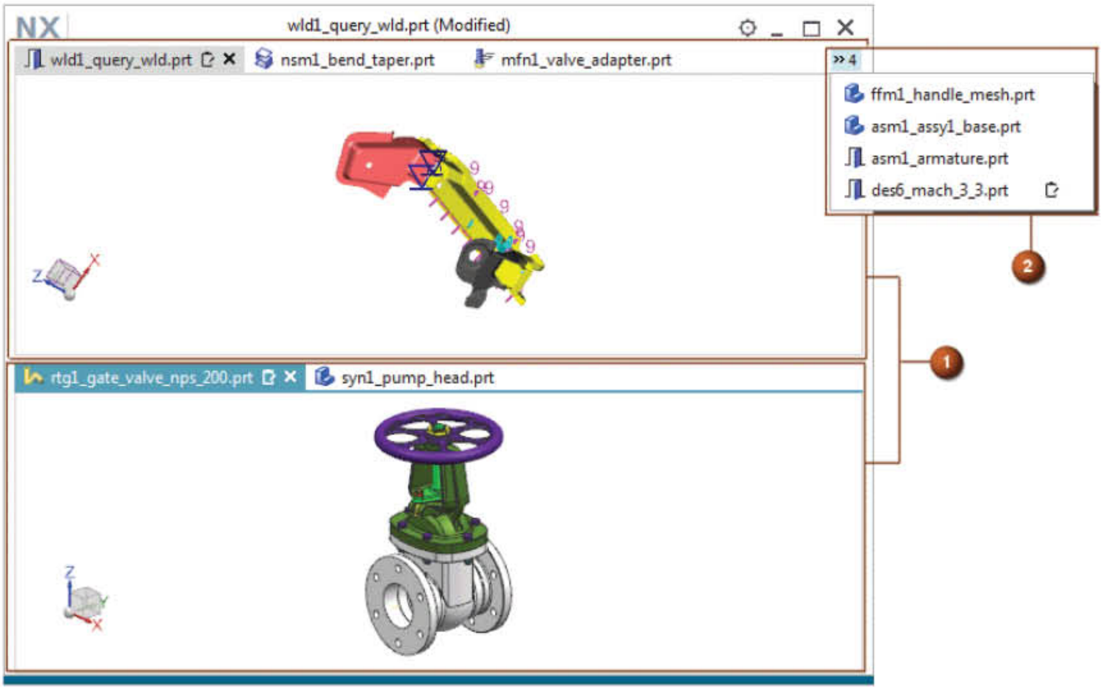


Figure 1.3 Floating frame

#	Component	Description
1	Tab group	Tabbed windows can be arranged into one or more *tabbed groups* within the main frame or floating frame.
2	**More Windows** list	Appears in the main frame and floating frame when a tabbed group contains too many tabs in the tab area. This provides access to the remaining windows in the tabbed group.

1.1.1.4 Shortcut toolbar

A *shortcut toolbar* is a toolbar that contains the commands that you are most likely to use for the selected objects.

If one or more objects are selected of the same type, the shortcut toolbar displays the commands that are specific to the selected object type. If you select multiple objects of different types, the shortcut toolbar displays commands that you can use on all the selected object types.

When no dialog box is open and one or more objects are selected in the graphics window, NX displays only the shortcut toolbar.

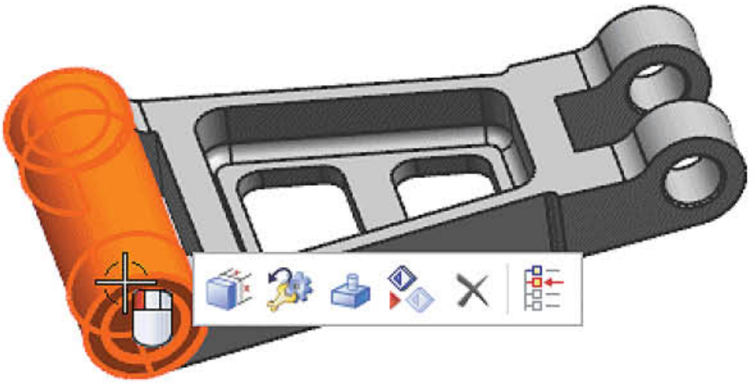


Figure 1.4 *Shortcut toolbar*

When no dialog box is open and one or more selected objects are right-clicked in the graphics window, the **Part Navigator**, or the **Assembly Navigator**, NX displays the shortcut toolbar and the shortcut menu.

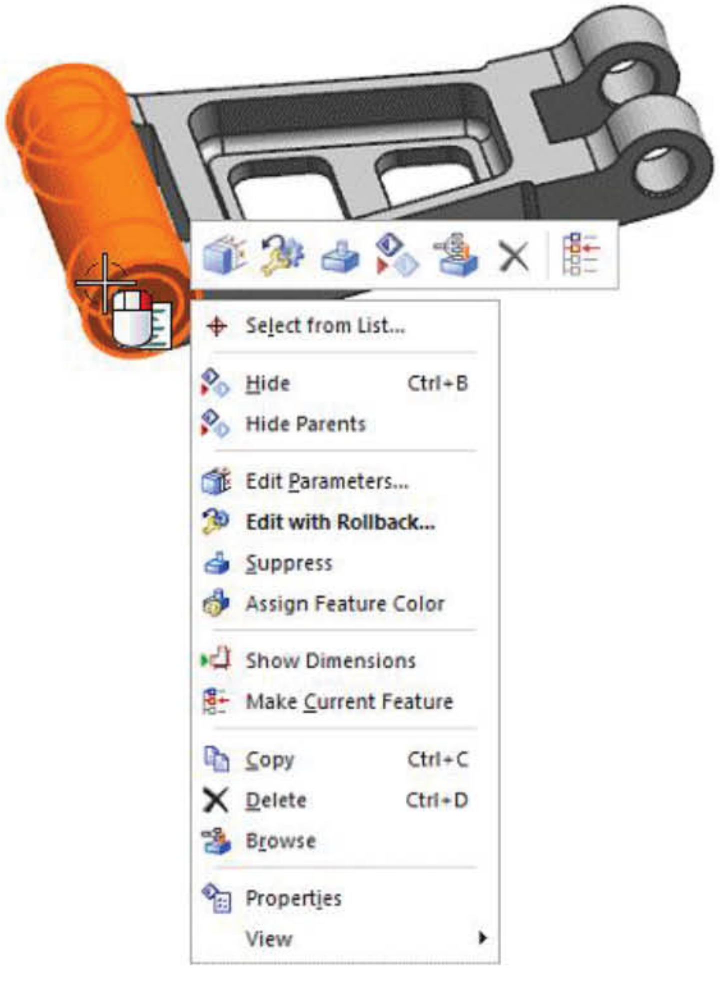
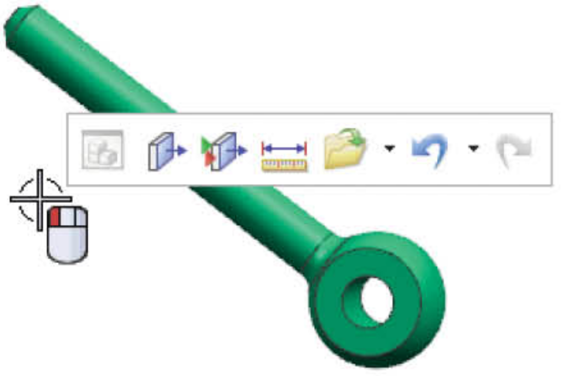


Figure 1.5 Shortcut menu

1.1.1.5 View shortcut toolbar

The **View** shortcut toolbar is displayed when:

- Clicking in the background of the graphics window.



- Holding Ctrl and clicking an object.

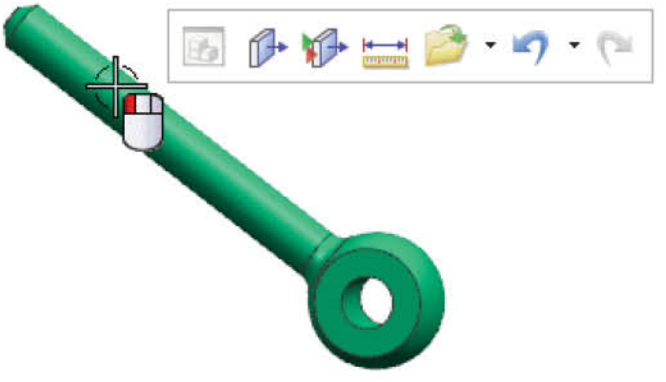


Figure 1.6 Tool bar

Note

If the **Drag and Drop between Windows**  command which enables dragging function is chosen, NX does not display the shortcut toolbar

1.1.1.6 Radial toolbars and radial shortcuts

Radial toolbar

Radial toolbars are available when Shift+Ctrl is held and a mouse button is clicked. The radial toolbar has been shown depends on the mouse button is clicked.

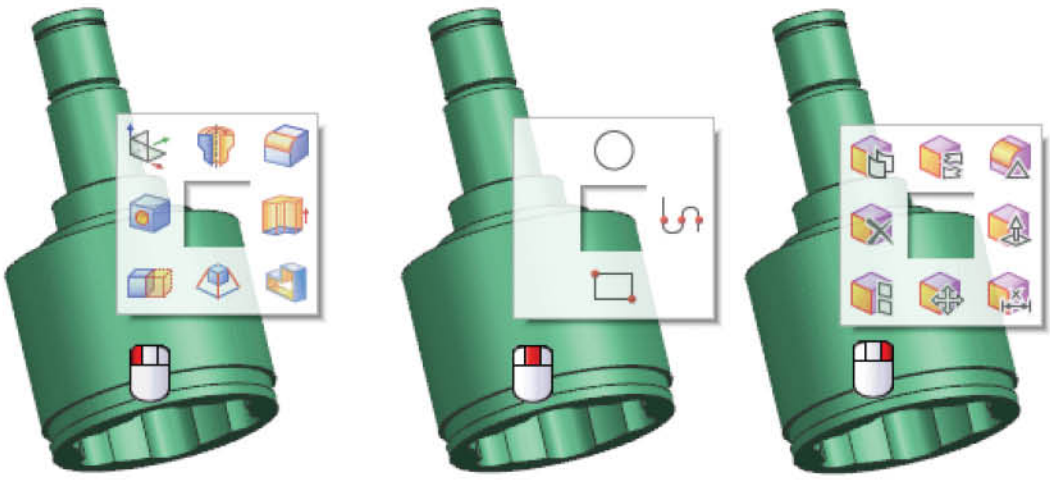
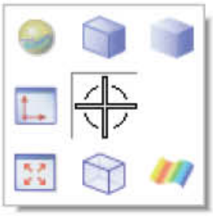


Figure 1.7 Radial toolbar

1.1.2.7 Radial shortcut

Radial shortcuts are available when you click and hold the right mouse button is held and clicked.

- A radial shortcut with view commands appears when you the right mouse button is held and clicked in the background of the graphics window.



- A radial shortcut with object-specific commands appears when the right mouse button is held and clicked on a selected object.

Note

If the **Drag and Drop between Windows**  command which enables dragging function is chosen, NX does not display the radial shortcut.

1.1.2.8 Keyboard accelerators

A *keyboard accelerator* is also known as a keyboard shortcut, a shortcut key, or a hot key. A new keyboard accelerator can be added or an existing one can be modified.

The following table lists a small number of keyboard accelerators in NX.

To do this task	Perform this action
Orient geometry in a trimetric view	Press Home
Orient geometry in an isometric view	Press End
Fit geometry to the graphics window	Press Ctrl+F
Switch between standard and full screen display	Press Alt+Enter
View Help on context	Press F1
View the Information window	Press Ctrl+Shift+S

Note When a text box is active, single letter accelerators do not work. The letter is entered in the text box instead.

Menu	**Information→Custom Menu Bar→Shortcut Keys** Note If the **Custom Menu Bar** submenu cannot be seen, change**Roles** to **Advanced**.

1.1.2.9 Mouse buttons and mouse key combinations

Many tasks can be done with NX by using the mouse or using a combination of a mouse button and a keyboard key.

To do this task		Perform this action
Select commands from menus or options in dialog boxes		Click the command or option.
Select objects in the graphics window		Click the object.
Select contiguous items in a list box		Hold Shift and click the items.
Select or de-select non-contiguous items in a list box		Hold Ctrl and click the items.
Start the default action for an object		Double click the object.

Cycle through all the required steps in a command prior to clicking the **OK** or **Apply** button		Click the middle mouse button.
Cancel a dialog box		Hold Alt and click the middle mouse button.
Display an object-specific shortcut menu		Right-click the object.
Display the **View Popup** menu		Right-click in the background of the graphics window or hold Ctrl and right-click anywhere on the graphics window.

1.1.1.10 3D input devices

A 3D input device is also known as a 3D mouse, a 3D motion control device, or a Spaceball. To control motion, use the puck or ball on a 3D device.

Figure 1.8 3D input devices

Using a 3D input device to:

- Rotate the model at any time without selecting the **Rotate** command or pressing any mouse buttons.
- Rotate the model with one hand while using the mouse with the other hand to select objects.
- Rotate, pan, and zoom at the same time in one fluid motion.
- Rotate, pan, and zoom in one direction.

If multiple windows are opened in the current session of the software, the 3D input device performs the action in the window in which the cursor is placed. If the cursor is not in any graphics window, the 3D input device performs the action in the work view of the active window.

1.1.1.11 Resource bar

The *Resource bar* contains tabs for navigators, a browser, and palettes. Each tab displays a page of information.

The location of the Resource bar, as well as the tabs displayed on the bar, depends on the specific configuration.

The Resource bar can also be displayed as a Ribbon bar tab.

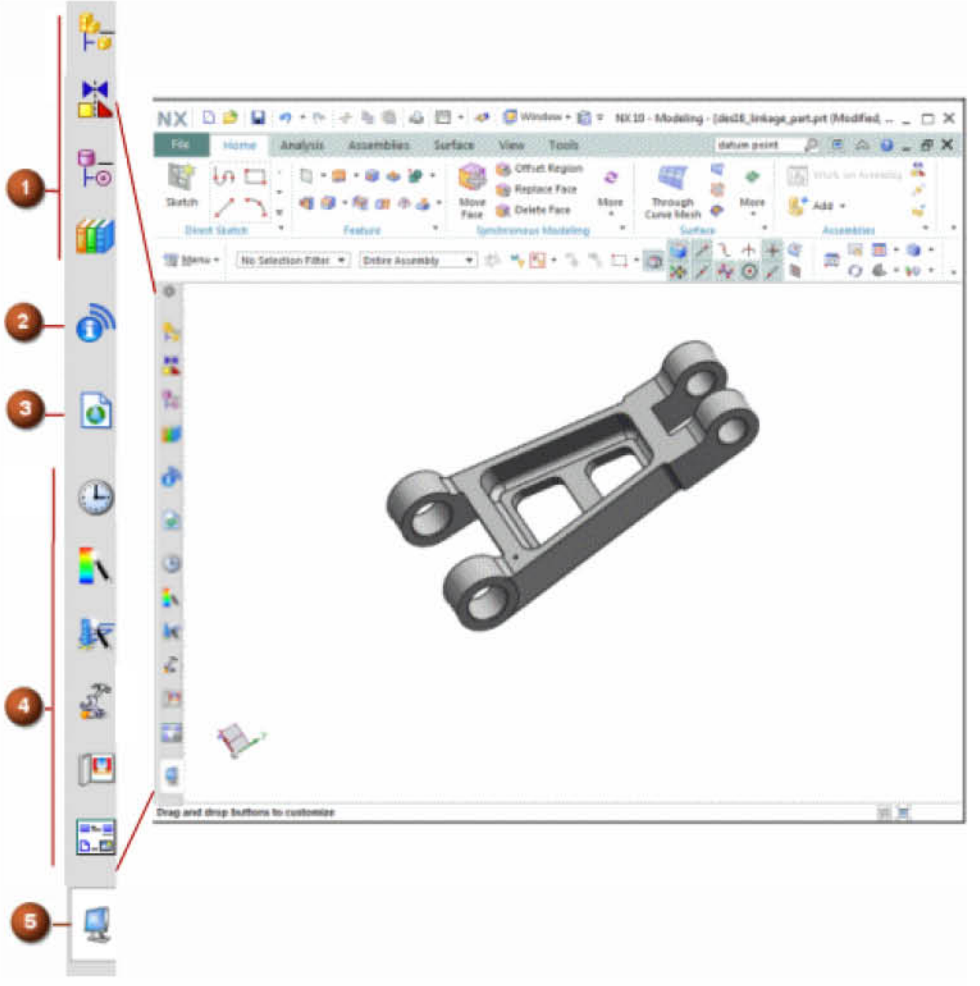


Figure 1.9 Resource bar

#	Component	Description
1	Navigators	Display information such as the features in a part and the components in an assembly. Navigators can be used to manage and edit data, view and change creation order, and select objects such as features, tools, or operations.
2	HD3D Tools	Provides access to HD3D tools that allow displaying and interacting with information directly on the 3D model.
3	Integrated Web browser	Provides access to the Internet from within the software.
4	Palettes	Provide access to standard and frequently used model data. Palettes can be used to access user-defined templates, roles, system visualization scenes, and system materials.
5	Custom tab	Provides access to third-party applications and dialog boxes into NX. For more information, see *Displaying host windows for third-party applications*

1.1.1.12 Open files in native NX

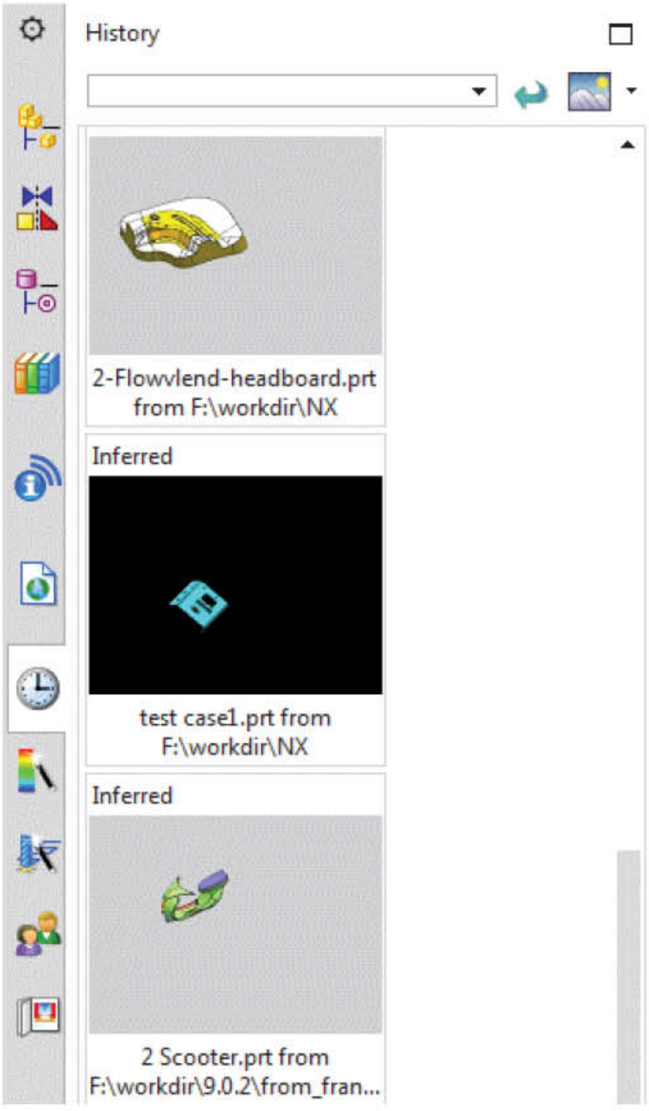
Several methods can be used to open a file in NX.

File→Open command

Use this command to open NX files and files from other compatible CAD/CAM products as .prt files.

This command is also useful when there is a desire to change loading options, for example, to open files with a lightweight representation.

History palette

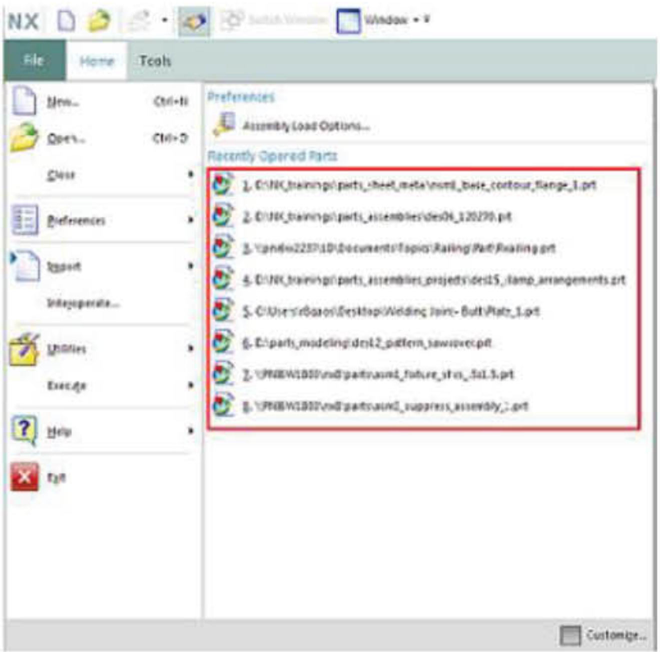


Use the **History** palette on the Resource bar to open a previously opened NX part file. NX organizes the files into folders based on when they are last opened. Thumbnails make it easy to identify parts.

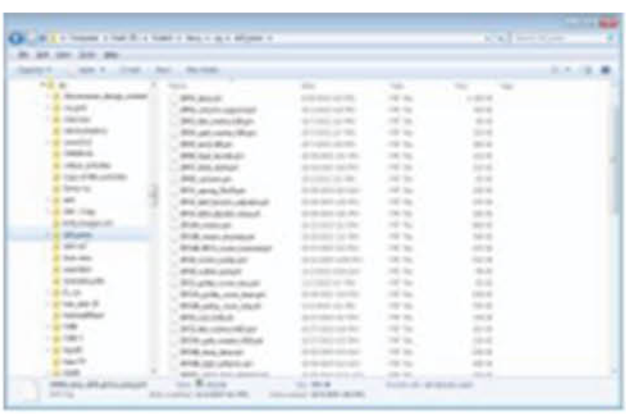
- To open the file as a displayed part, double-click a thumbnail.
- To add a part as a component to the currently displayed part, drag the part into the graphics window.

Recently Opened Parts list

Use this list on the **File** tab to open the most recently opened parts. NX lists the name and folder path for each part.



System folders



- To open the file as a displayed part, double-click the file name.
- To add a part as a component to the currently displayed part, drag the part into the graphics window.

1.1.3 Process for creating 3D parts

NX is a parametric and feature-based system that allows you to create 3D parts, assemblies, and 2D ***drawings***. The design process in NX is shown below.

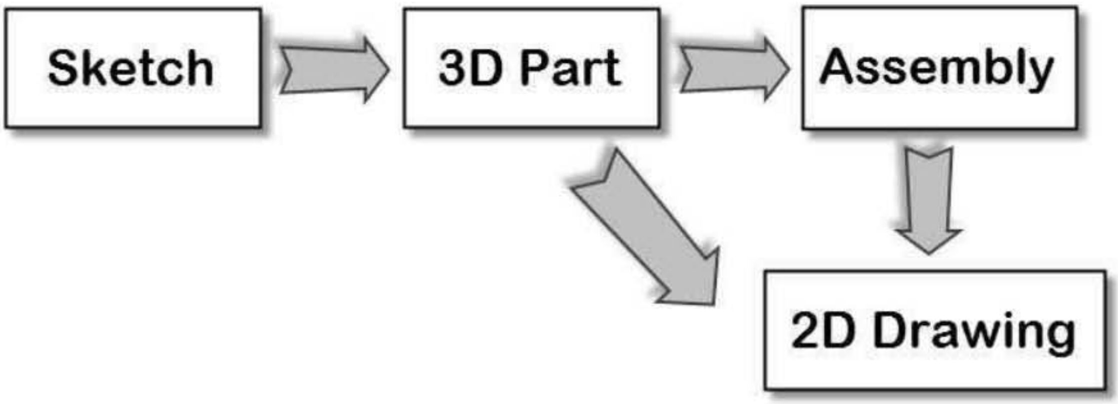
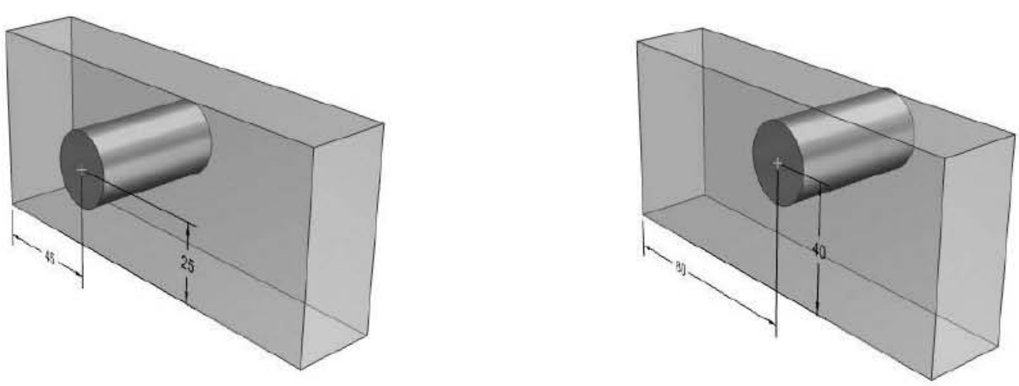
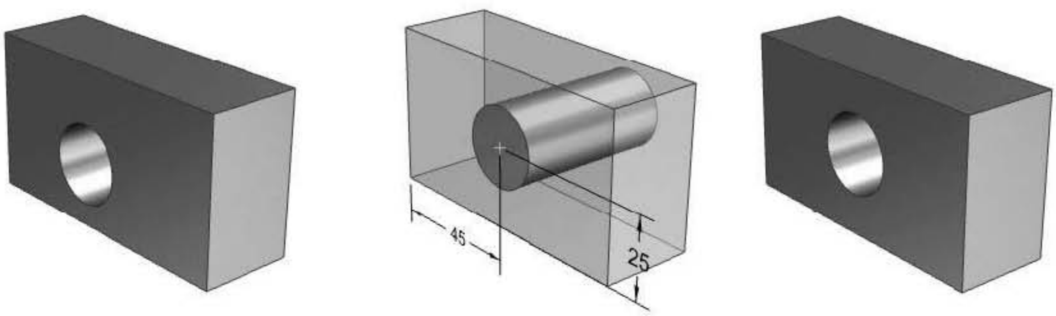


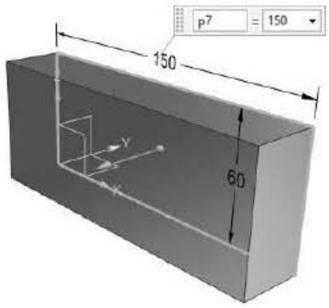
Figure 1.10 Process for creating 3D parts

In NX, parameters, dimensions, or relations control everything. For example, if there is a desire to change the position of the hole shown in figure, it is necessary to change the dimension or constraint that controls its position.

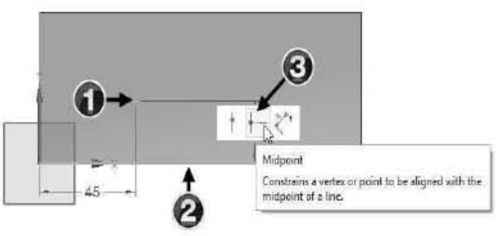


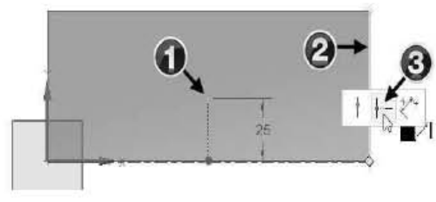
The parameters and constraints allow having control over the design intent. The design intent describes the way the 3D model will behave when dimensions and constraints are applied to it. For example, to position the hole at the center of the block, one way is to add dimensions between the hole and the adjacent edges. However, when the size of the block is changed, the hole will not be at the center.

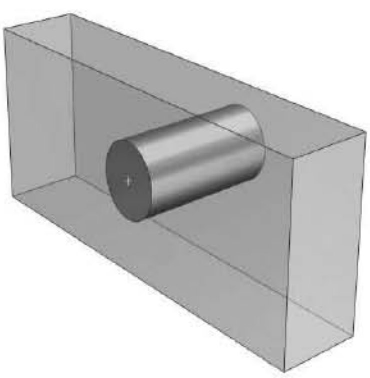




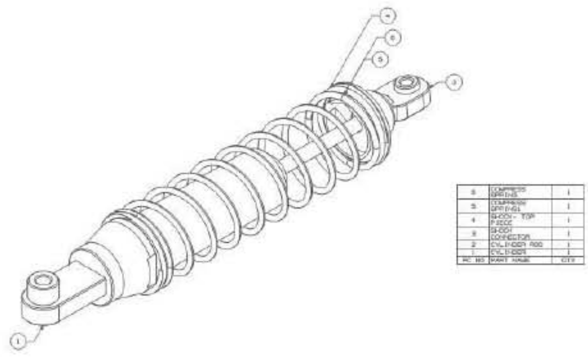
It is able to make the hole to be at the center, even if the size of the block changes. To do this, it is necessary to delete the dimensions and apply the **Midpoint** constraint relationships between the hole point and thehorizontal and vertical edges. Now, even if you change the size of the block, the hole will always remain at the center.







The other big advantage of NX is the associativity between parts, assemblies, and drawings. When the changes are made to the design of a part, they will take place in any assembly that it is a part of. In addition, the 2D drawing will update automatically.



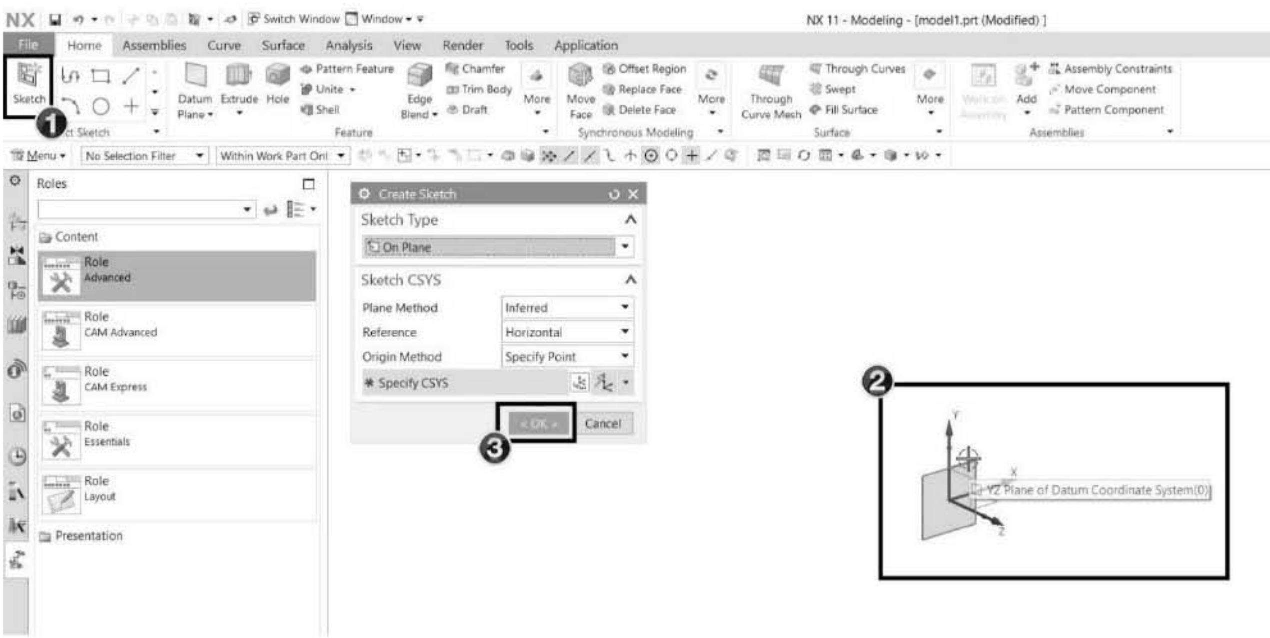
1.2. Design 2D with NX

This part covers the methods and commands to create sketches used in the part-modeling environment. The commands and methods are discussed in context to the part-modeling environment. In NX, you can create sketches in two environments: Part and Sketch task environment. Instructions will be given on how to create sketches in both environments.

In NX, create a rough sketch, and then apply dimensions and constraints that define its shape and size. The dimensions define the length, size, and angle of a sketch element, whereas constraints define the relations between sketch elements.

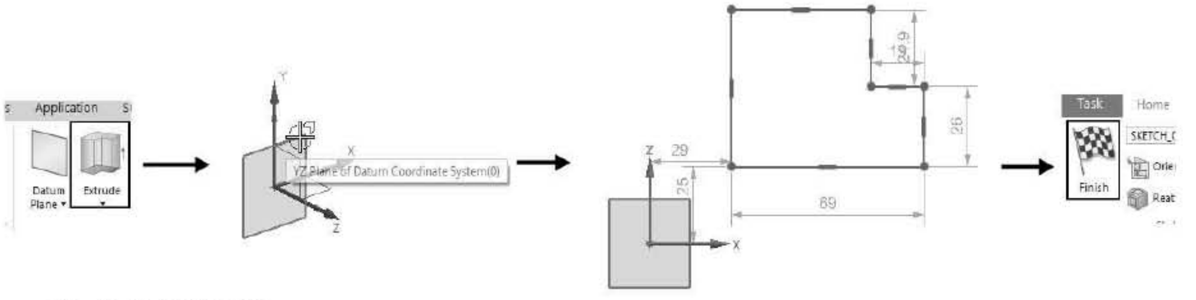
1.2.1 Basic commands in 2D part design

Creating sketches in NX Part environment is very easy. The **Sketch** command must be activated, and then define a plane on which you want to create the sketch. To do this, click **Home > Direct Sketch > Sketch** on the ribbon. Next, click on any of the Datum Planes located at the center of the graphics window. On the **Create Sketch** dialog, click **OK** to start the sketch. It is now able to start drawing sketches on the selected plane. After creating the sketch, click **Home > Direct Sketch > Finish Sketch** on the ribbon to finish the sketch.



1.2.1.1 Sketching in the Sketch Task environment

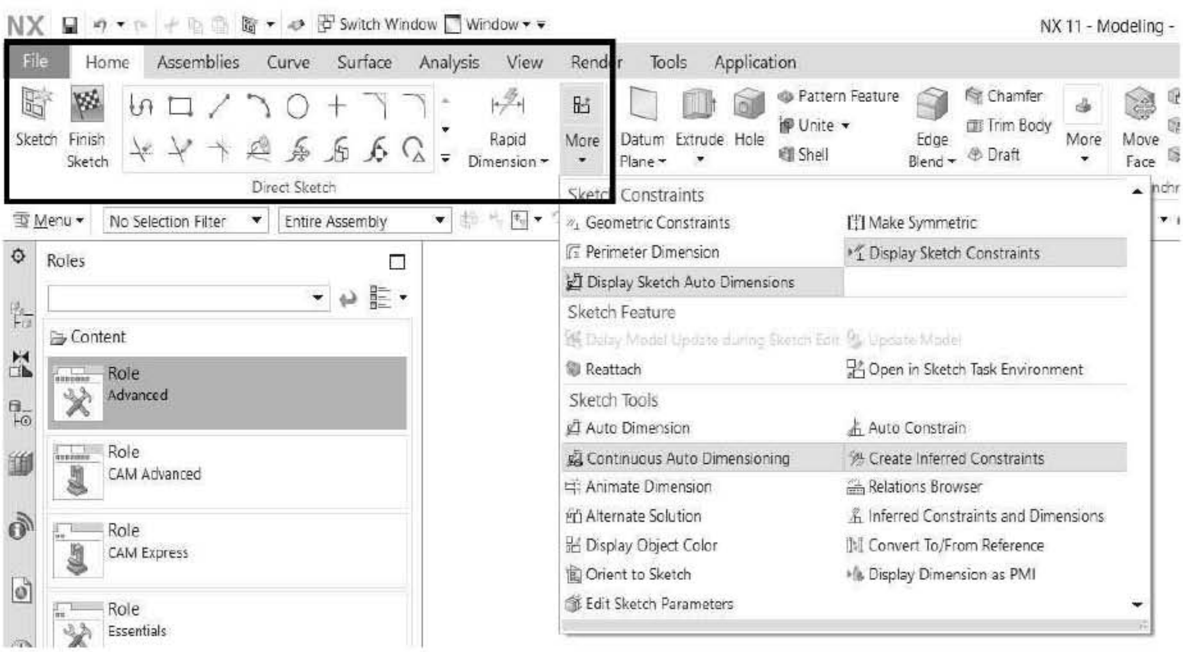
The Sketch Task environment is used to create sketches only to complete a specific task. For example, to create a sketch to construct an *Extrude* feature, activate the **Extrude** command (On the ribbon, click **Home > Feature > Extrude**), and then click on a Datum plane. The Sketch Task environment will be active. It can be noticedthat the **Profile** command is also activated, by default. It is able to start sketching lines or select any other sketching command. After completing the sketch, click **Home > Sketch > Finish** on the ribbon to come out of the Sketch



Taskenvironment.

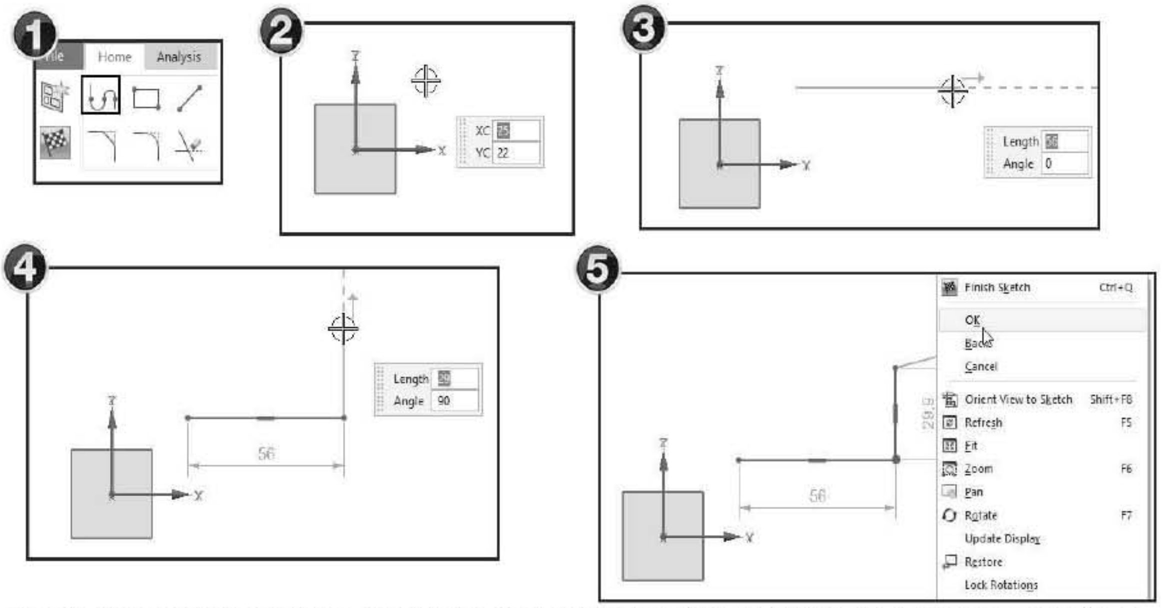
1.2.1.2 Drawing Commands

NX providesa set of commands to create sketches. These commands are located on the **Direct Sketch**panel of the **Home** ribbon.



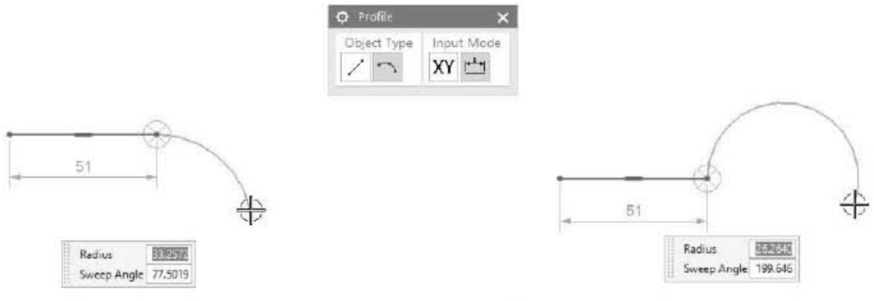
1.2.1.3 The Profile command

This is the most commonly used command while creating a sketch. To activate this command, click **Home > Direct Sketch > Profile** on the ribbon. As you move the pointer in the graphics window, it can be noticed that a box is attached to it. It displays the X and Y coordinates of the pointer. To create a line, click in the graphics window,move the pointer and click again. After clicking for the second time, it can be seen that an end point is added and another line segment is started. This is a convenient way to create a chain of lines. Continue to click to add more line segments. To end the chain, right-click in the graphics window and click OK. Notice that the Profile command is still active. Create another chain of line segments or press Esc to deactivate this command.



Tip: *To create a horizontal line, specify the start point of the line and move the pointer horizontally; a dotted horizontal line appears. Click the on the dotted line to create a horizontal line. In addition, the Horizontal constraint is applied to the line. Constraints will be learned later in this chapter. Likewise, you can create a vertical line by moving the pointer vertically and clicking.*

The **Profile** command can also be used to draw arcs continuous with lines. Click the **Arc** icon on the **Profile** dialogto draw this type of arc. The figure below shows the procedure to draw arcs connected to lines. To switch from tangent arc to normal or vice-versa, move the pointer to the end point of the previous line.



Move the cursor along the direction of the line to create a tangent arc

Move the cursor in the direction normal to the line to create a normal arc

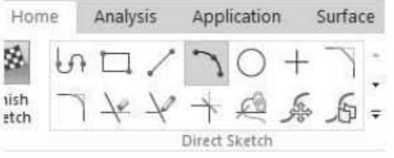
To delete a line, select it and press the **Delete** key. To select more than one line, click on multiple line segments; thelines will be highlighted. Multiple lines can also be selected by dragging a box from left to right. Press and hold theleft mouse button and drag a box from left to right; the lines inside the box boundary will be selected.

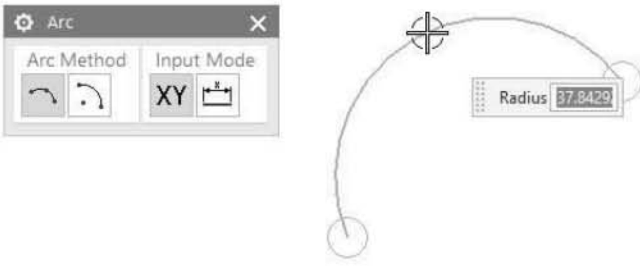
1.2.1.4 The Arc command

This command creates an arc using two methods: **Arc by 3 Points** and **Arc by Center and Endpoints**.

The Arc by 3 Points method

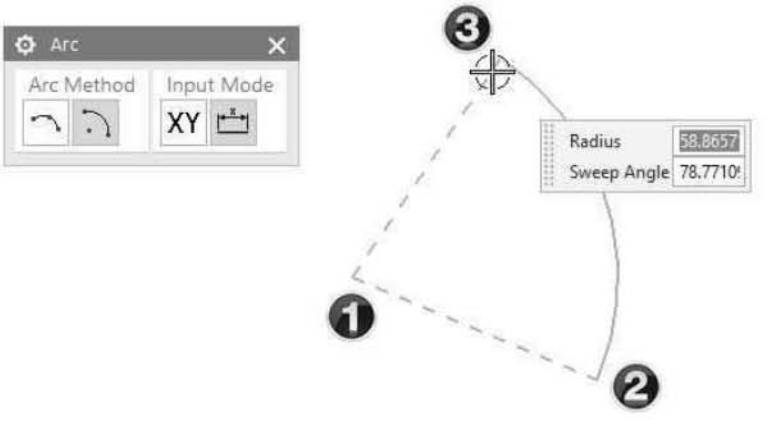
This method creates an arc by defining its start, end, and radius. Activate the **Arc** command (click **Home > DirectSketch > Arc** on the ribbon). On the **Arc** dialog, click the **Arc by 3 Points** button, and then click to define the startpoint of the arc. Click again to define the end point. After defining the start and end of the arc, the size and position of the arc need to be specified. Move the pointer and click to define the radius and position of the arc.





The Arc by Center and Endpoints method

This method creates an arc by defining its center, start and end. Activate the **Arc** command and click **Arc by Center and Endpoints**. Click to define the center point. Next, move the pointer and it can be noticed that a dotted line appears between the center and the pointer. This line is the radius of the arc. Now, click to define the start point of the arc and move the pointer; it can be noticed that an arc is drawn from the start point. Once the arc appears as desired, click to define its end point.

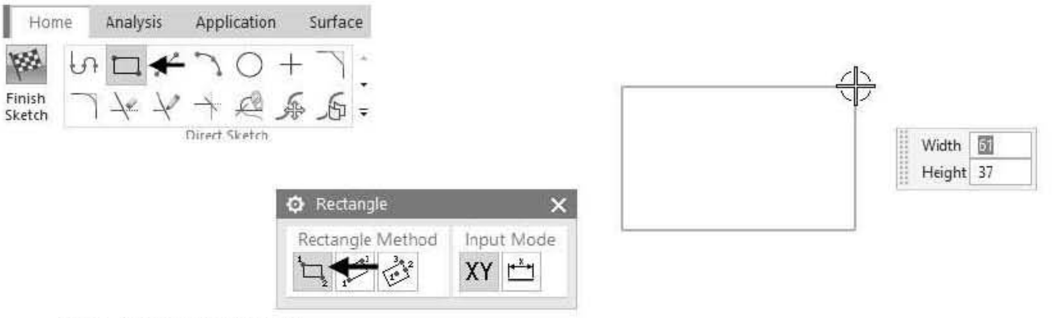


1.2.1.5 The Rectangle command

This command creates a rectangle using three different methods: **By 2 Points**, **By 3 Points**, and **From Center**.

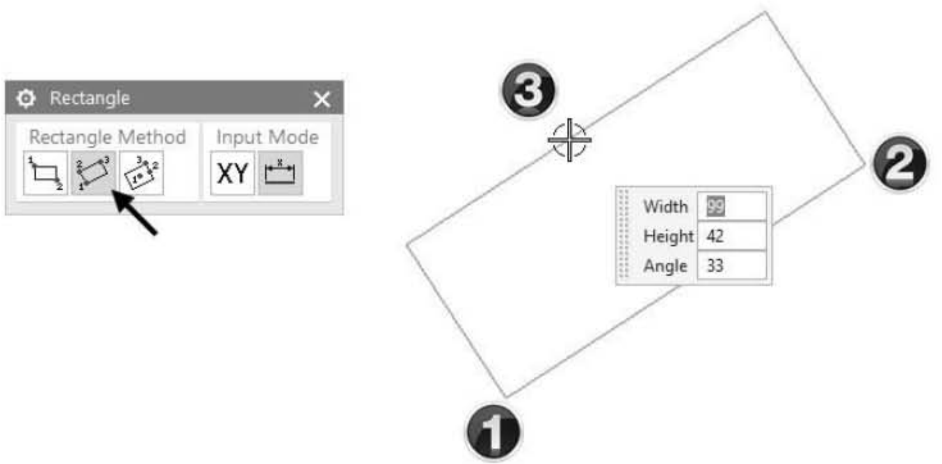
The 2 Points method

This method creates a rectangle by defining its diagonal corners. Activate the **Rectangle** command (On the ribbon, click **Home > Direct Sketch > Rectangle**). On the **Rectangle** dialog, click **By 2 Points** and click to define the first corner. Drag the pointer and click to define the second corner. It is also possible to type values in the **Width** and **Height** boxes attached to the pointer.



The 3 Points method

This method creates an inclined rectangle. The first two points define the width and inclination

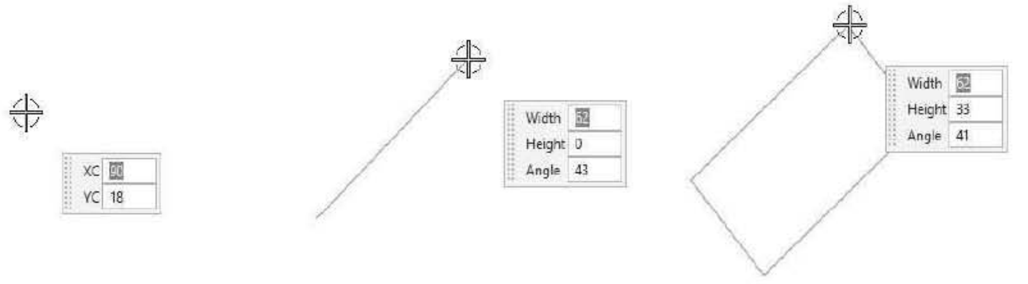


angle of the rectangle. The third point defines its height.

The other procedure to create a 3 Points rectangle is to specify the first corner. Next, type in a value in the **Angle** box. Press the Tab key and type values in the **Width** and **Height** boxes. Click in the graphics window to create the rectangle.

The From Center command

This method creates a rectangle by defining three points: center of the rectangle, and mid and endpoints of the height. Activate the **Rectangle** command and select the **From Center** button on the **Rectangle** toolbar. Specify the center point of the rectangle, move the pointer, and click to define the width and orientation angle of the rectangle. Move the pointer and click to define the height of the rectangle.

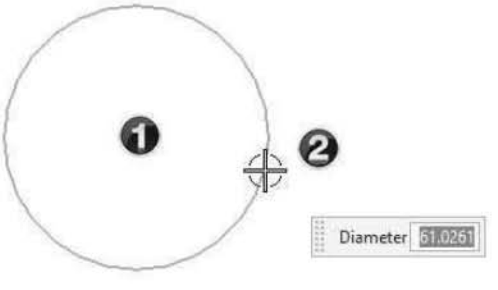


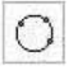
1.2.1.6 The Circle command

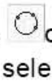
This command creates a circle using two methods: **Circle by Center and Diameter** and **Circle by 3 Points**.

 The Circle by Center and Diameter method

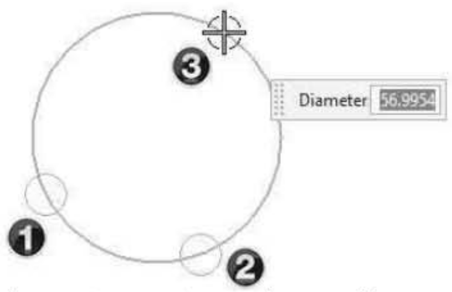
This is the most common way to draw a circle. Activate the **Circle** command (click **Home > Direct Sketch > Circle** on the ribbon) and click **Circle by Center and Diameter** on the **Circle** dialog. Click to define the center point of the circle. Drag the pointer, and then click again to define the diameter of the circle.



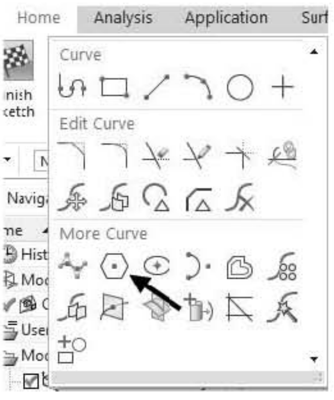
 The Circle by 3 Points method

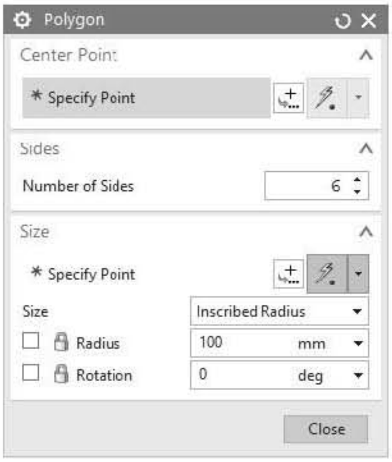
This method creates a circle by using three points. Activate the **Circle** command and click **Circle by 3 Points**  on the **Circle** dialog. Select three points from the graphics window. It is also possible to select existing points from the sketch geometry. The first two points define the location of the circle and the third point defines its diameter.

1.2.1.7 The Polygon command

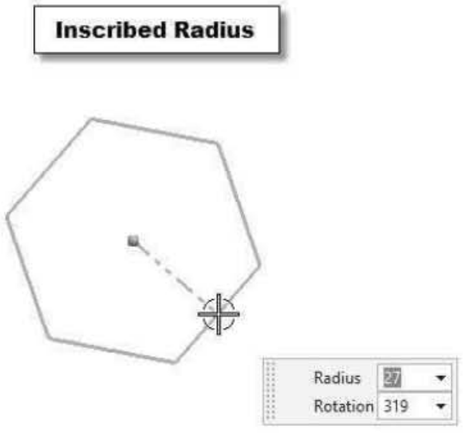


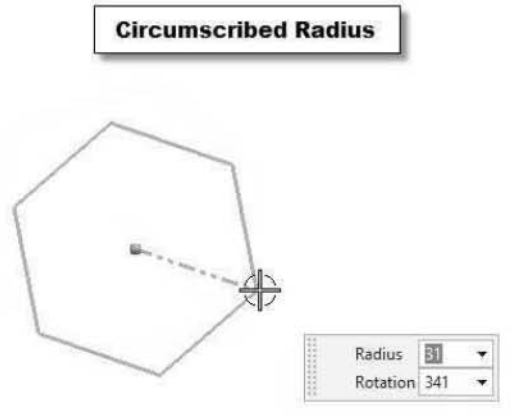
This command provides a simple way to create a polygon with any number of sides. Activate this command (On the ribbon, click **Home > Direct Sketch > More Curve > Polygon**) and click in the graphics window to define the center of the polygon. As you move the pointer away from the center, you will see a preview of the polygon. To change the number of sides of the polygon, just click in the **Number of Sides** box on the dialog and enter a new number. Next, press the ENTER key to update the preview.





Now, The size of the polygon must be defined. On the dialog, the **Size** menu has three options to define the sizeof the polygon: **Inscribed Radius**, **Circumscribed Radius**, and **Side Length**. If you select **Inscribed Radius**, the pointer will be on one of the flat sides of the polygon. If **Circumscribed Radius is selected**, a vertex of the polygonwill be attached to the pointer. Click in the window to define the size and angle of the polygon. The size and angle of the polygon can also be defined by entering values in the **Radius** and **Rotation** boxes on the dialog.

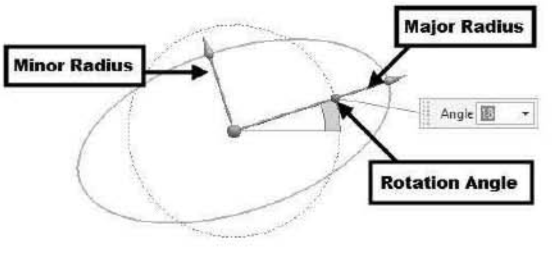




If you select **Side Length**, then you have to define the length and angle of one side a polygon. Type-in values in the **Length** and **Rotation** boxes and press Enter to create the polygon.

1.2.1.8 The Ellipse command

This command creates an ellipse using a center point, and major and minor axes. Activate this command (On the ribbon, click **Home > Direct Sketch > More Curve > Ellipse**) and click to define the center of the ellipse. On the **Ellipse** dialog, type values in the **Major Radius** and **Minor Radius** boxes. The arrows attachedto the ellipse can also be dragged to define the major and minor radius.

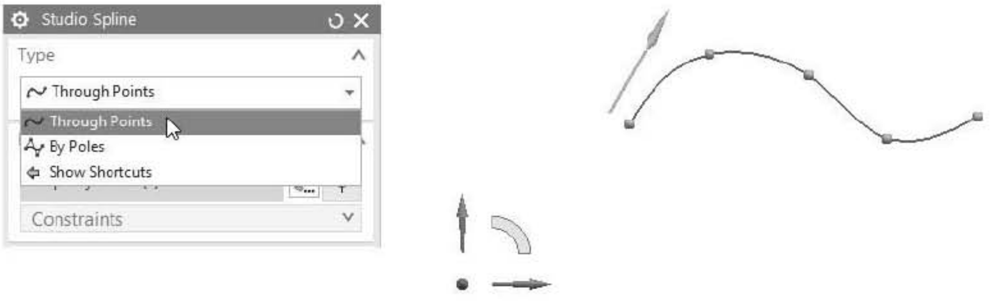


On the dialog, type-in a value in the **Angle** box or drag the angle modifier on the ellipse to define the rotation angleof the ellipse. On the dialog, click **OK** to create the ellipse.

1.2.1.9 The Studio Spline command

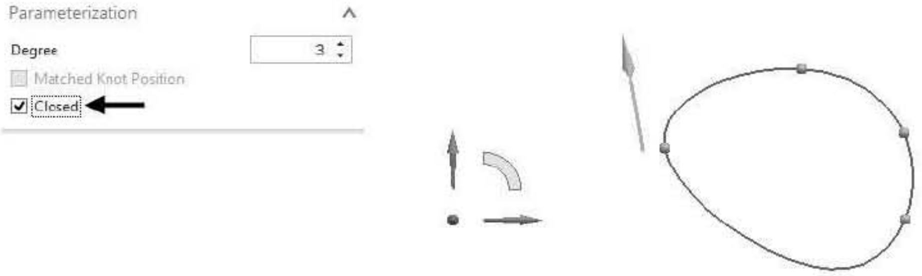
This command creates a smooth B-spline curve using two different methods: **Through Points** and **By Poles**. B- Splines are non-uniform curves, which are used to create smooth shapes.

The **Through Points** method creates a smooth spline passing through the points selected.



In the **By Poles** method, various points called as poles will be defined. As the poles are defined, grey lines are created connecting them. The spline will be drawn tangent to these lines.

The **Close Curve** option can be also used to create a closed curve.

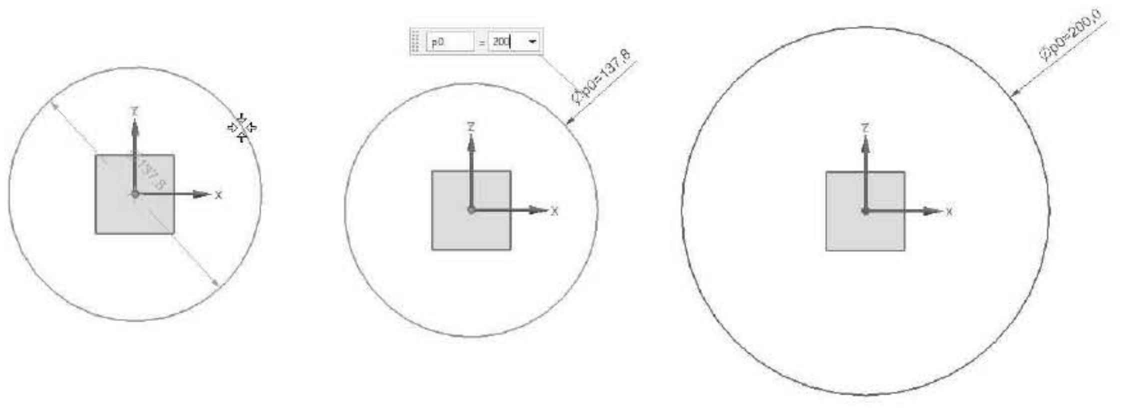


On the dialog, click **OK** to complete the studio spline.

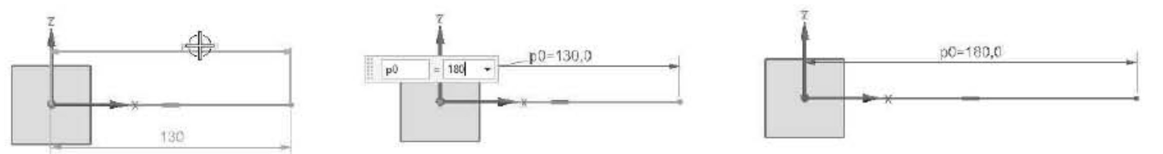
1.2.1.10 The Rapid Dimension command

It is generally considered a good practice to ensure that every sketch created is fully constrained before creating solid features. The term, 'fully-constrained' means that the sketch has a definite shape and size. A sketch can be fully constrained by using dimensions and constraints. As sketches are created in NX, some dimensions are added to the sketch elements. These dimensions are called Driven dimensions and they do not have any control over the sketch geometry. To allow these dimensions to control the shape and size of the sketch geometry, they have to be converted into Driving dimensions. Driving dimensions are so named because they drive the geometry of the sketch.

Driving dimensions can be added to a sketch by using the **Rapid Dimension** command. This command can be used to add all types of dimensions such as length, angle, and diameter and so on. This command creates a dimension based on the geometry selected. For instance, to dimension a circle, activate the **Rapid Dimension** command, and then click on the circle. Next, move the pointer and click again to position the dimension; a box will pop up. Type a value in this box, and then press Enter to update the dimension.



If a line is clicked, the **Rapid Dimension** command automatically creates a linear dimension. Click once more to position the dimension, and then type-in a value and press Enter; the

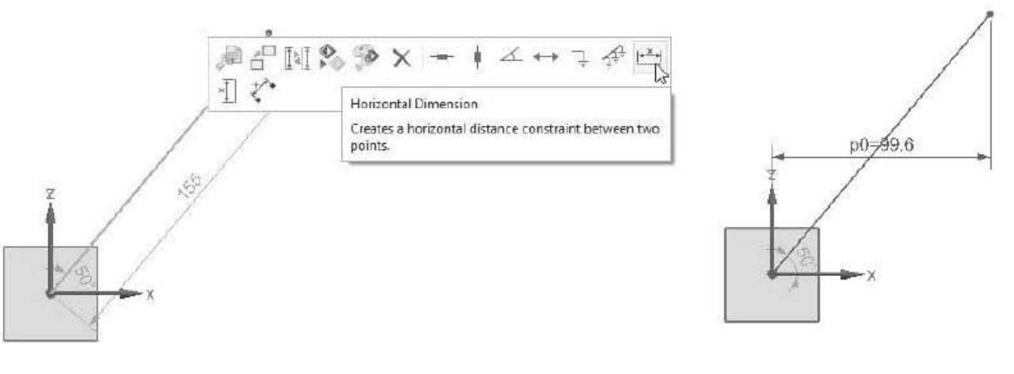


dimension will be updated.

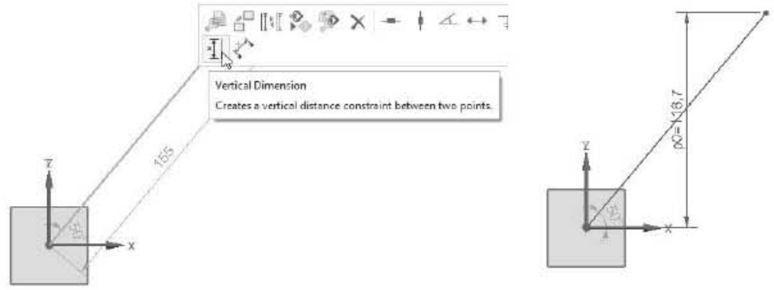
On the **Rapid Dimension** dialog, click **Close** to deactivate the **Rapid Dimension** command.

Linear Dimensions

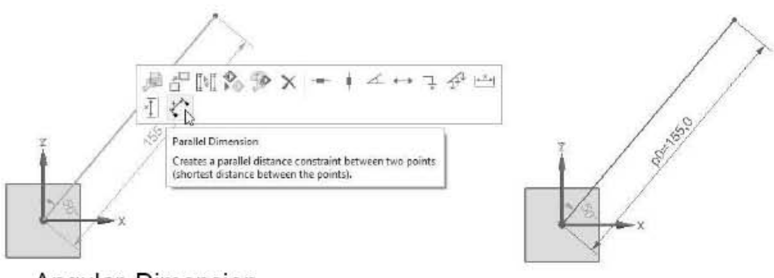
NX allows you to create various types of linear dimensions. Select a line and click on the **Horizontal Dimension** icon on the Contextual toolbar; a horizontal dimension is created.



Select a line and click the **Vertical Dimension** icon on the Contextual toolbar; a vertical dimension will be created.

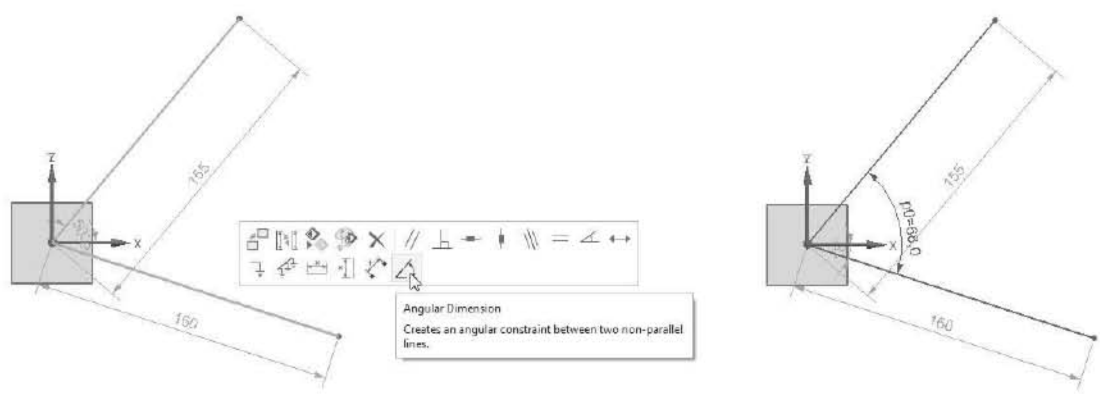


To know the true length of the line, select the line and click the **Parallel Dimension** icon on the **Contextual Toolbar**. Next, double-click on the dimension, type a value in the box, and press Enter; the dimension is updated.



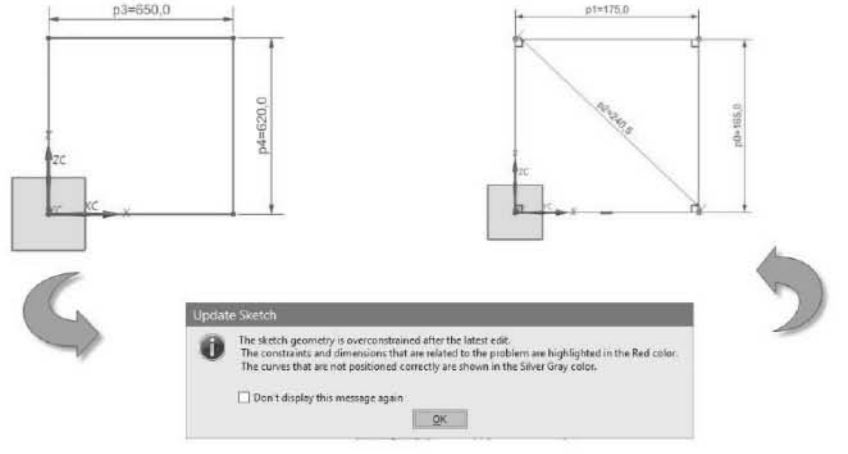
Angular Dimension

The procedure to create an angle dimension is similar to that of linear dimensions. Select two lines that are positioned at an angle to each other. Click the **Angular Dimension** icon on the Contextual Toolbar. Double-click on the angular dimension, type-in a value, and press Enter.

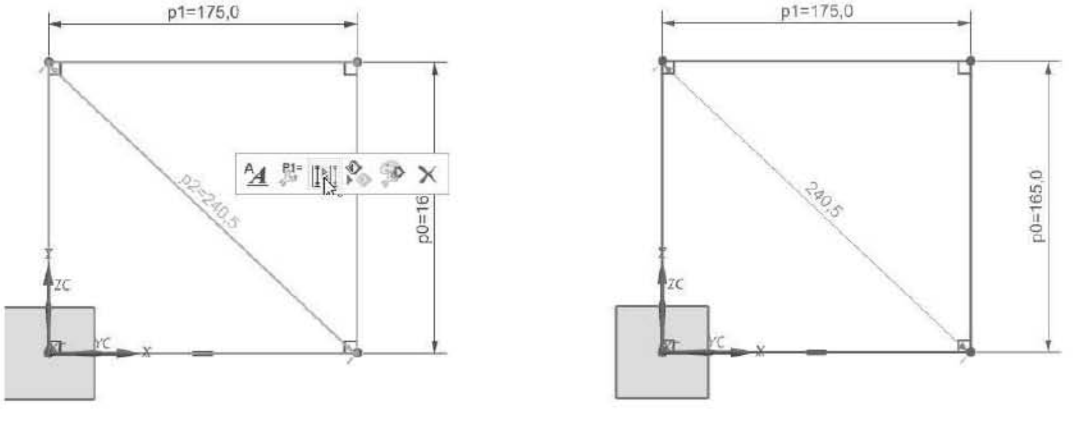


Over-constrained Sketch

When creating sketches for a solid or surface feature, NX will not allow over-constraining the geometry. The term 'over-constrain' means adding more dimensions than required. The following figure shows a fully constrained sketch. If another dimension is added to this sketch (e.g. diagonal dimension), the **Update Sketch** message pops up. It shows that the dimension over constrains the sketch. when clicking **OK**, all the dimensions in the sketch will become red.



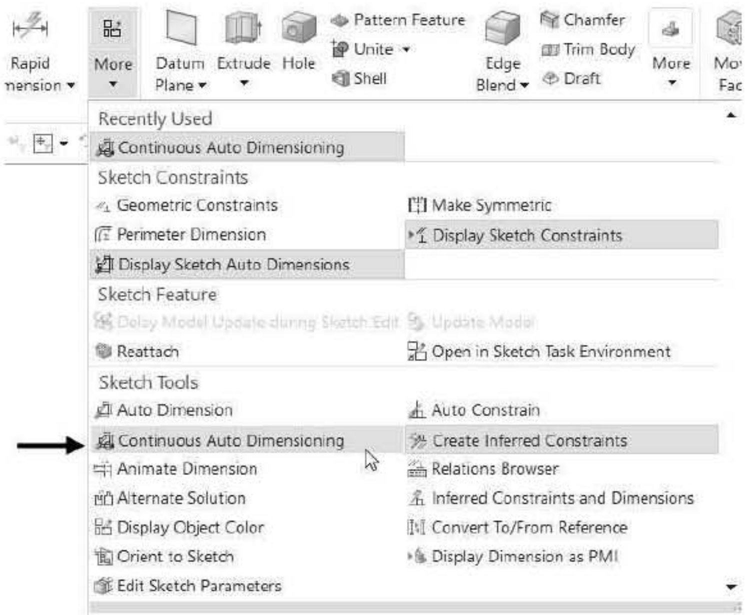
Now, one of the dimensions has to be made as a Reference dimension. Click on the diagonal dimension and select **Convert to Reference** on the contextual toolbar. The **Convert Dimension to Reference** message appears. Click **OK** to convert the dimension to reference. The reference dimensions will be in brown. Now, if the value of the width is changed, the reference dimension along the diagonal updates automatically. Also, note that the dimensions which are initially created will be driving dimensions, whereas the dimensions created after fully defining the sketch are driven dimensions.



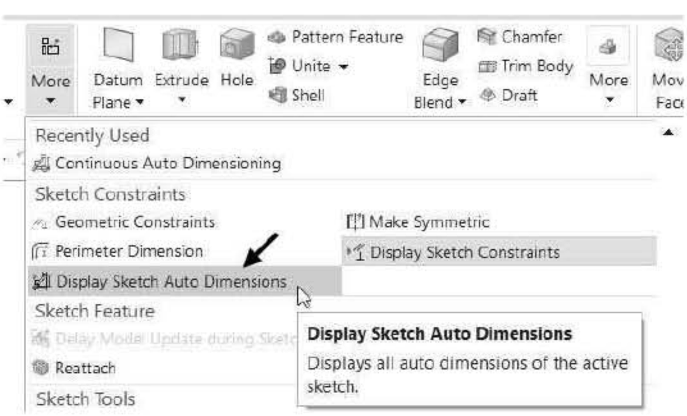
Continuous Auto Dimensioning

NX creates dimensions automatically while a sketch is being drawn. However, if you do not want dimensions to be created automatically, then deactivate the **Continuous Auto Dimensioning** command (on the ribbon, click **Home > Direct Sketch > More > Continuous Auto Dimensioning)**

> .



It is also possible to hide the already created continuous auto dimensions by deactivating the **Display Sketch AutoDimensions** button (Click **Home > Direct Sketch > More > Sketch Constraints > Display Sketch Auto Dimensions**). This button can be activated to display the auto dimensions.

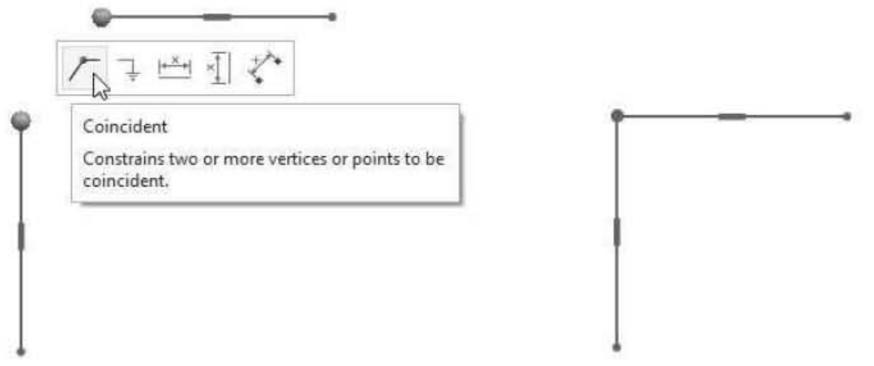


1.2.1.11 Geometric Constraints

Geometric Constraints are used to control the shape of a drawing by establishing relationships between the sketch elements. These geometric constraints can be applied using the **Geometric Constraints** command (On the ribbon, click **Home > Direct Sketch > Geometric Constraints**) or with the help of Contextual Toolbar.

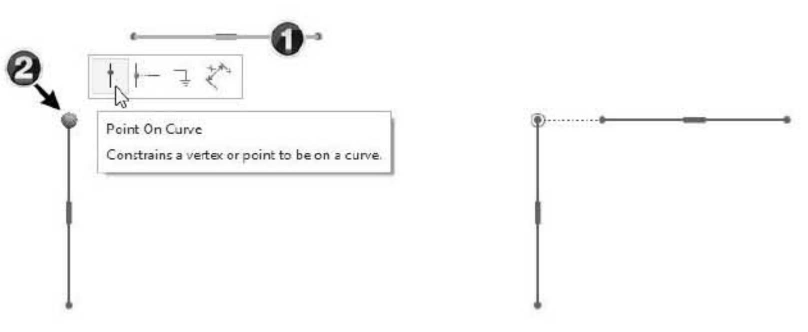
Coincident

This constraint connects one point with another point. Select the points to be made coincident and click the **Coincident** icon on the Contextual Toolbar. The selected points will be connected.



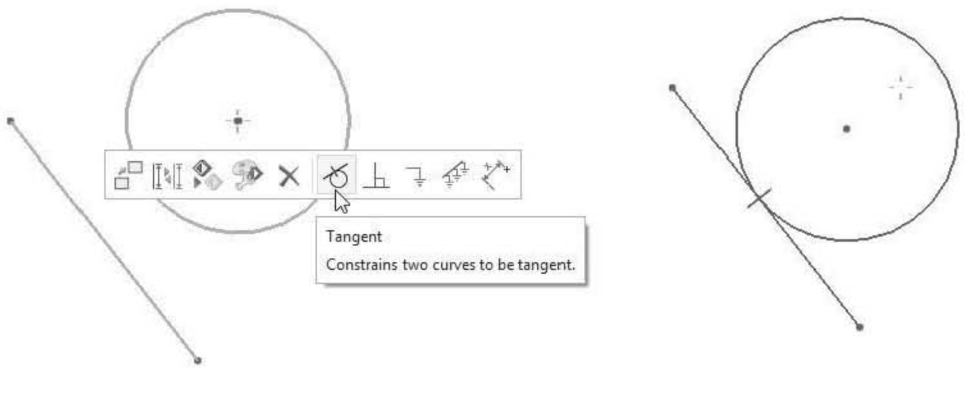
Point on Curve

This constraint makes a vertex or a point to be on a line, curve, arc, or circle. Select a curve and point, and click **Point on Curve** on the Contextual Toolbar. The point will lie on the curve or the curve extension.



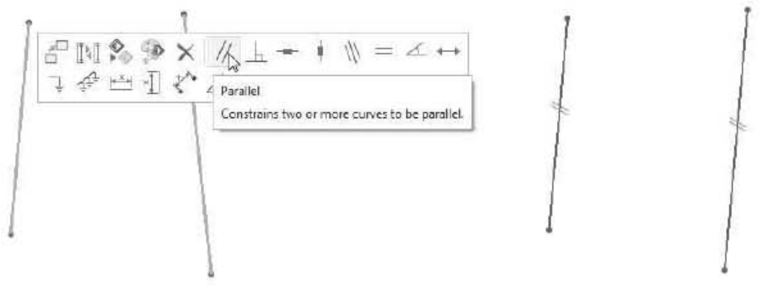
Tangent

This constraint makes an arc, circle, or line tangent to another arc or circle. Select a circle, arc, or line. Select another circle, arc, or line. On the **Contextual Toolbar**, click the **Tangent** button; both the elements become tangent to each other.



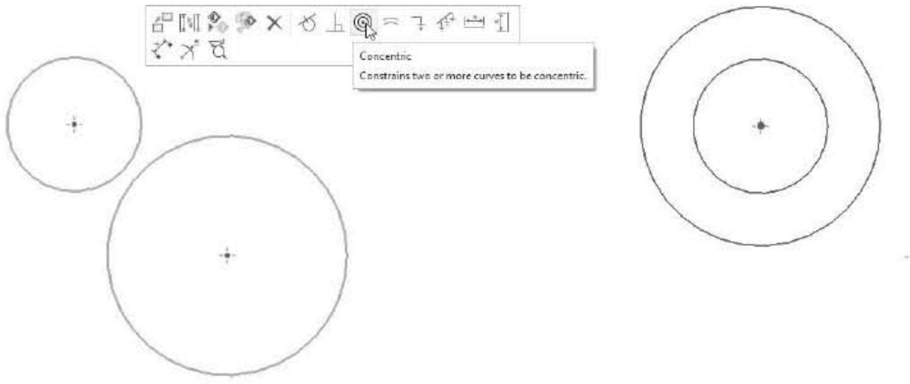
Parallel

This constraint makes two lines parallel to each other. Select two lines from the sketch and click **Parallel** on the Contextual Toolbar. The under constrained line is made parallel to the constrained line. For example, if a line is selected with vertical constraint and a free to move line, the free-to-move line becomes parallel to the vertical line.



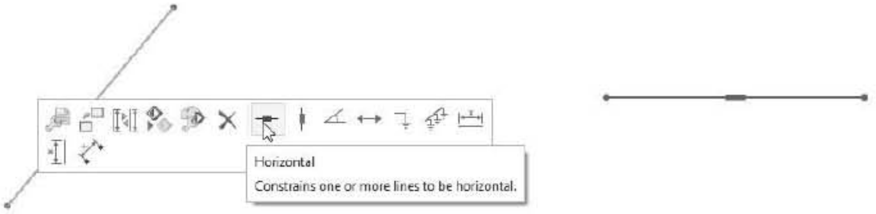
Concentric

This constraint makes the center points of arcs, circles or ellipses coincident. Select a circle or arc from the sketch. Select another circle or arc, and then click **Concentric** on the Contextual Toolbar. The first circle/arc will be concentric with the second circle/arc.



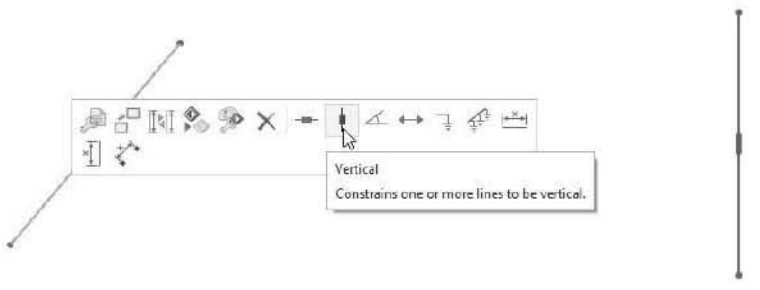
Horizontal

This constraint makes a line horizontal. Select a free-to-move line, and then click **Horizontal** on the Contextual toolbar; the line is made horizontal.



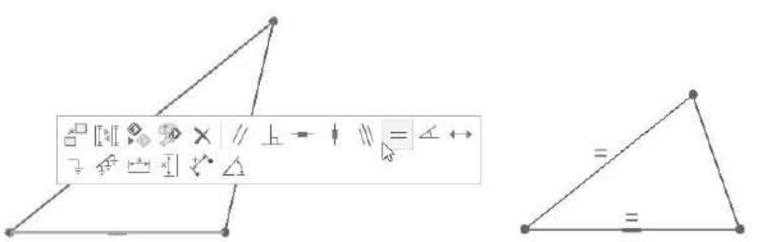
Vertical

This constraint makes a line vertical. Select an under-constrained line, and then click **Vertical** on the Contextual toolbar.



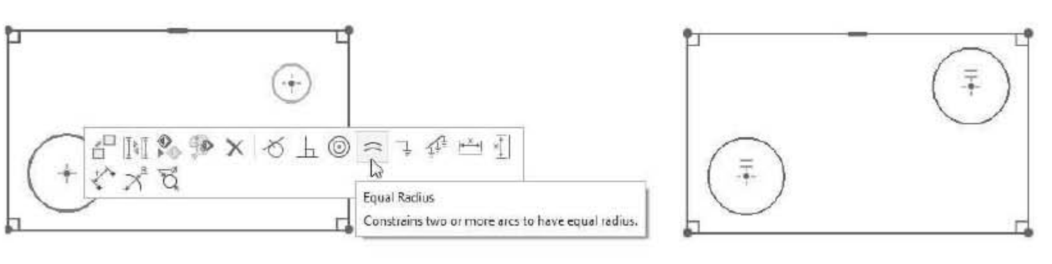
Equal Length

This constraint makes two lines equal in length.



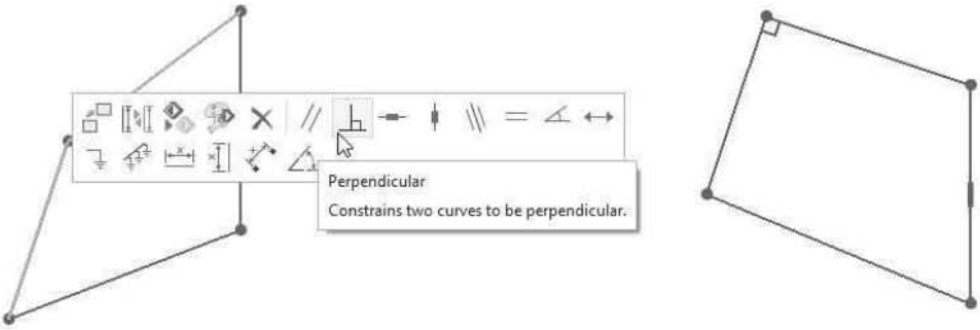
Equal Radius

This constraint makes circles or arcs equal in radius.



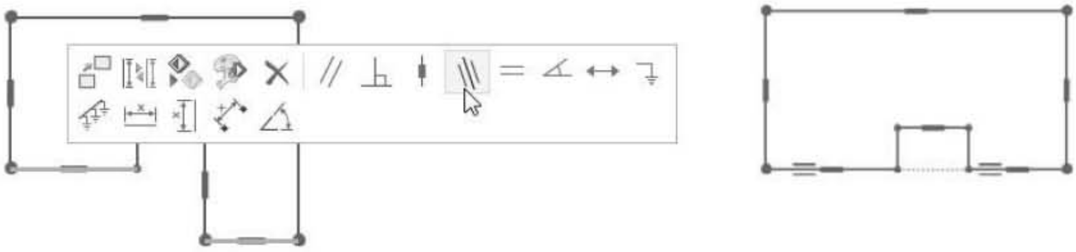
Perpendicular

This constraint makes two lines perpendicular to each other. Select two lines from the sketch, and click the **Perpendicular** icon on the Contextual Toolbar. The two lines will be made perpendicular to each other.



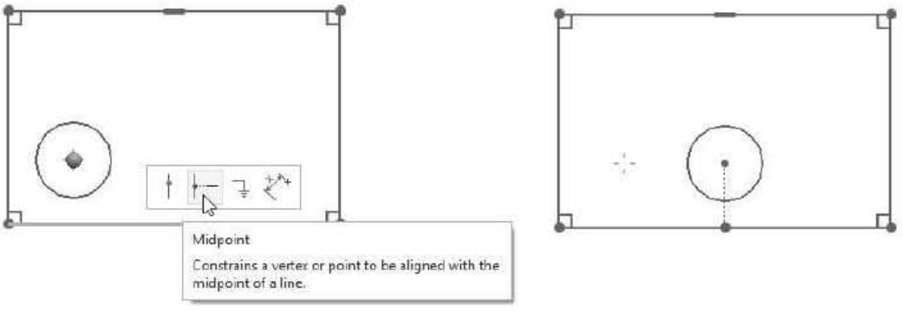
Collinear

This constraint forces a line to be collinear to another line. The lines are not required to touch each other.



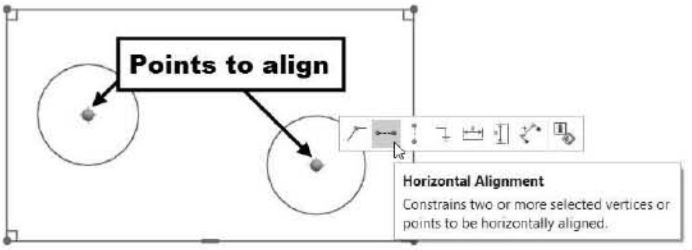
Midpoint

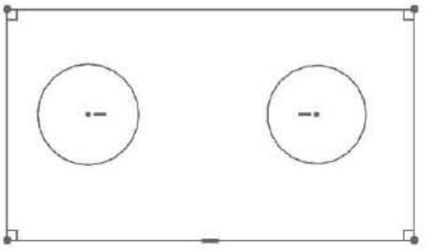
This constraint forces a point or vertex to be aligned with the midpoint of a line. Click on a point or vertex, and then click on a line. Select **Midpoint** from the Contextual Toolbar.



Horizontal Alignment

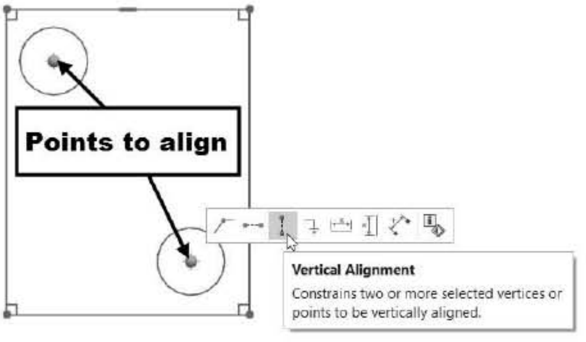
The **Horizontal Alignment** constraint aligns the two selected points horizontally. Select the points to align horizontally, and then click the **Horizontal Alignment** button on the Contextual Toolbar.

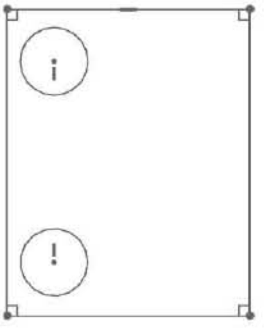




Vertical Alignment

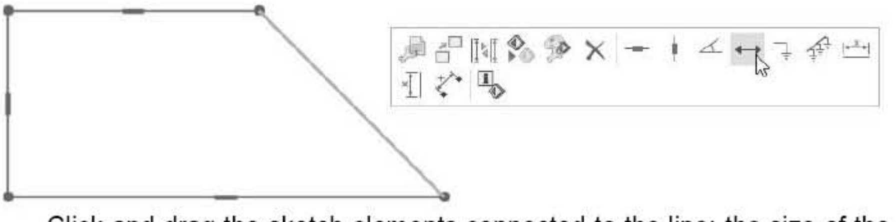
The **Vertical Alignment** constraint aligns the two selected points vertically. Select the points to align vertically, andthen click the **Vertical Alignment** button on the Contextual Toolbar.



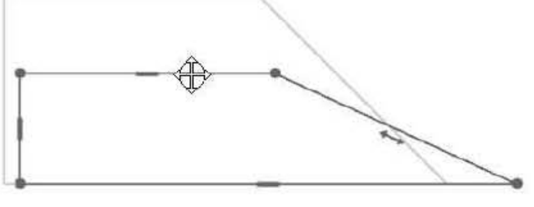


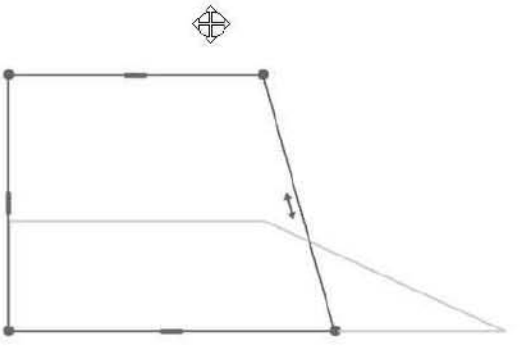
Constant Length

This constraint fixes the length of a selected line. Select a line and click the **Constant Length** button on theContextual Toolbar.



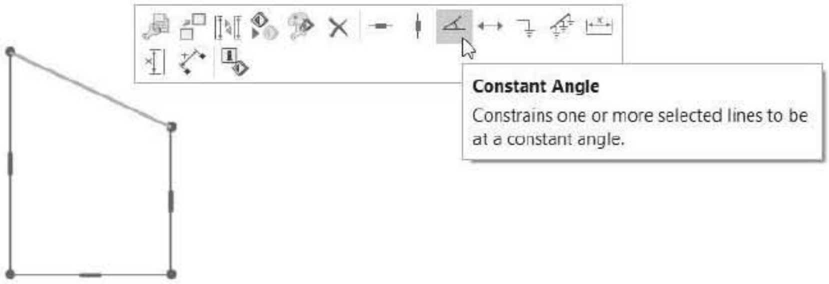
Click and drag the sketch elements connected to the line; the size of the connected elements changes but thelength of the line remains constant.



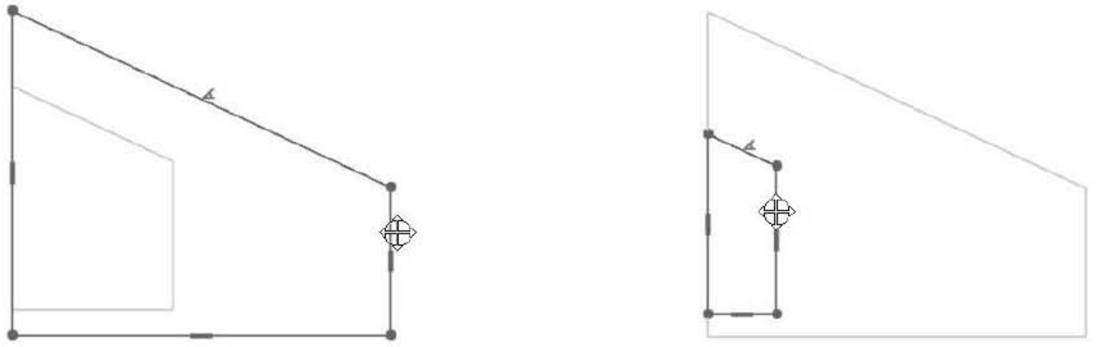


Constant Angle

This constraint fixes the angle of a selected line. Select a line and click the **Constant Angle** button on the Contextual Toolbar.

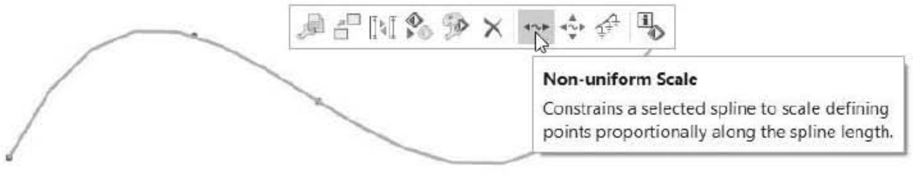


Click and drag the sketch elements connected to the line; the size of the sketch elements changes but the angle ofthe line remains constant.

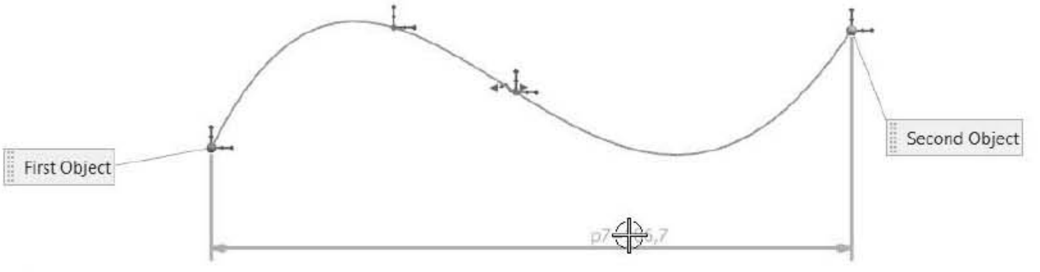


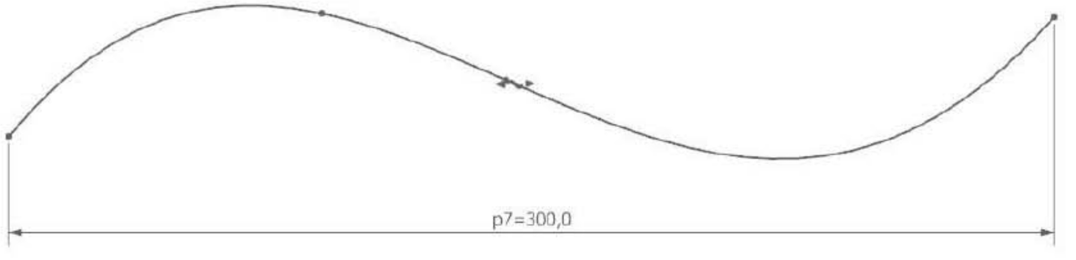
Non-uniform scale

This constraint scales a spline in horizontal or vertical direction. Select a spline and click **Non-uniform scale** onthe Contextual toolbar.



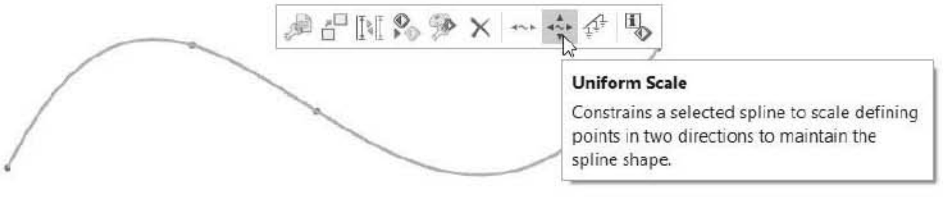
Activate the **Rapid Dimension** command, and then select the two end points of the spline. Change the dimensionvalue; the spline is scaled along the direction of the dimension.



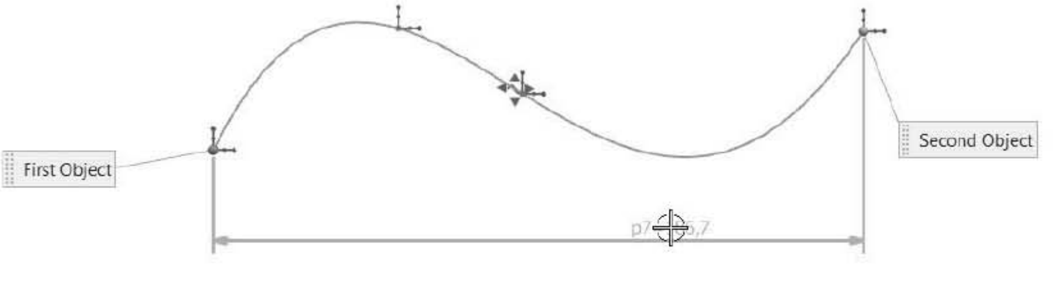


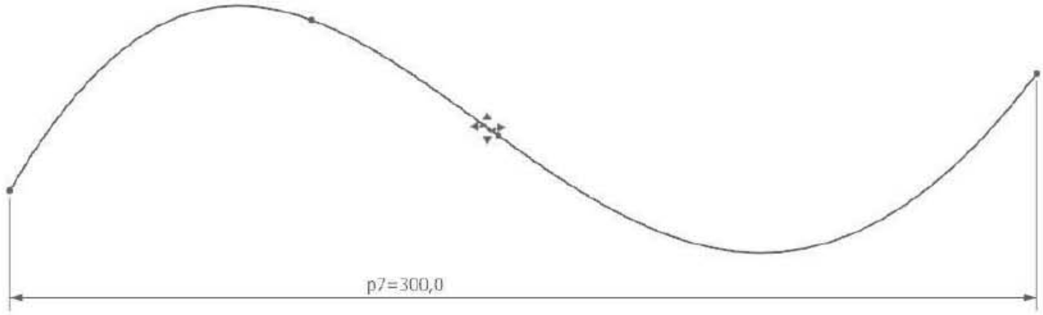
Uniform scale

This constraint scales a spline in both horizontal and vertical direction. Select a spline and click **Uniform scale** onthe Contextual toolbar.



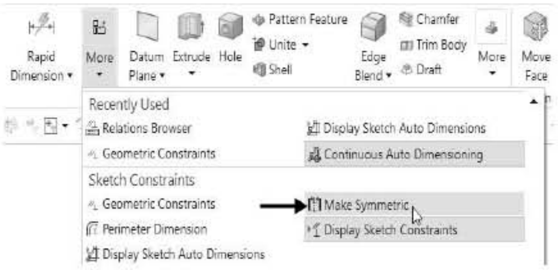
Activate the **Rapid Dimension** command, and then select the two end points of the spline. Change the dimension value; the spline is scaled uniformly in both horizontal and vertical directions.

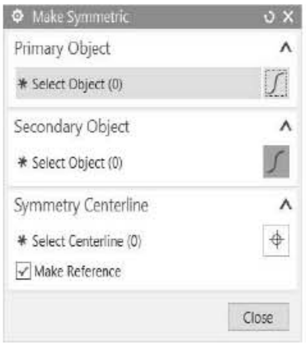


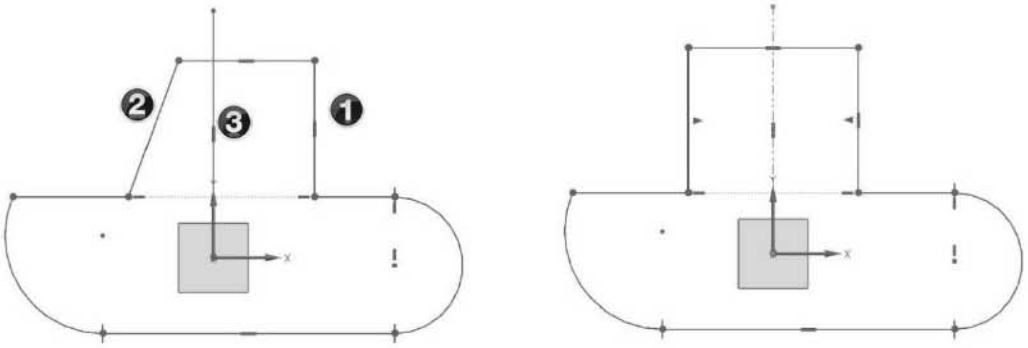


Make Symmetric

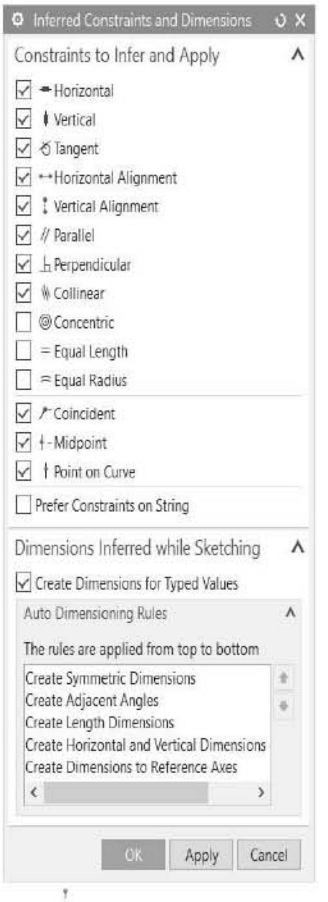
This command makes two objects symmetric about a line. The objects will have same size, position and orientation about a line. Activate this command (On the ribbon, click **Home > Direct Sketch > More > Make Symmetric**) and click on the first object. Click on the second object, and then define a symmetrical centerline. The two objects will be made symmetric about the centerline. The Make Reference option on the Make Symmetric dialog converts the centerline into a reference object.

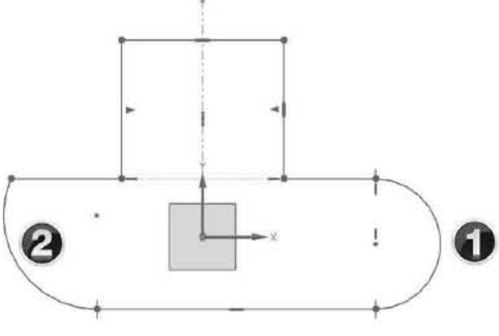


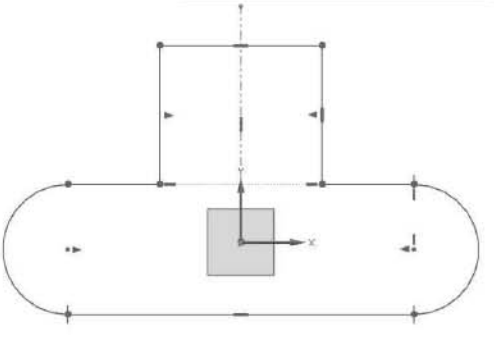




It is possible to continue selecting the objects to be made symmetric object the previously selected centerline. Close the dialog after applying the symmetric constraint.







Create Inferred Constraints

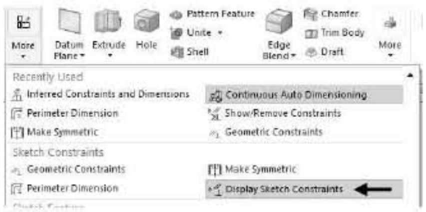
Constraints can also be applied automatically by activating the **Create Inferred Constraints** command. Activate ordeactivate **Create Inferred Constraints** by clicking **Direct Sketch > More > Create Inferred Constraints** on the

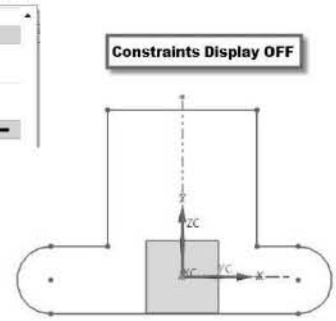
ribbon. With this command turned on, constraints are applied automatically when the sketch elements are created. It is possible to define which constraints to apply automatically by using the **Inferred Constraints and Dimensions** dialog. On the ribbon, click **Home > Direct Sketch > More > Inferred Constraints and Dimensions**. On the **Inferred Constraints and Dimensions** dialog, select the constraints to be created while sketching elements.

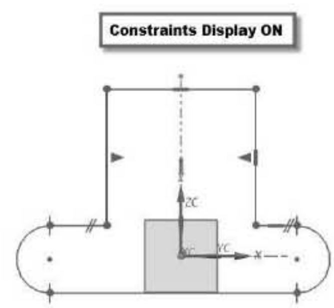
Under the **Dimensions Inferred while Sketching** section, define the order in which the automatic dimensions are applied. Check the **Create Dimensions for Type Values** to create driving dimensions when creating sketches by typing exact values. Click **OK** to close the dialog.

Display Sketch Constraints

As constraints are created, they can be viewed using the **Display Sketch Constraints** button (On the ribbon, click **Home > Direct Sketch > More > Display Sketch Constraints**). When dealing with complicated sketches involving numerous constraints, you can deactivate this button to turn off the display of all constraints.

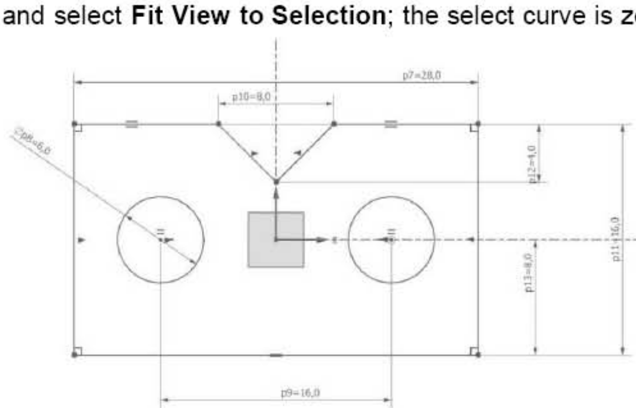


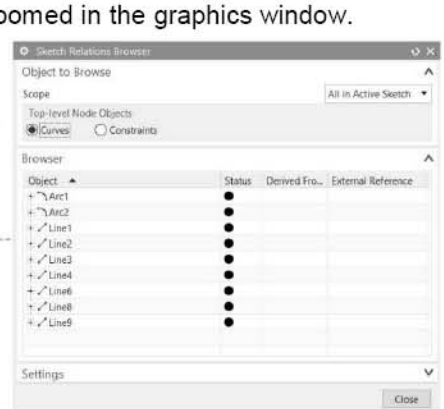




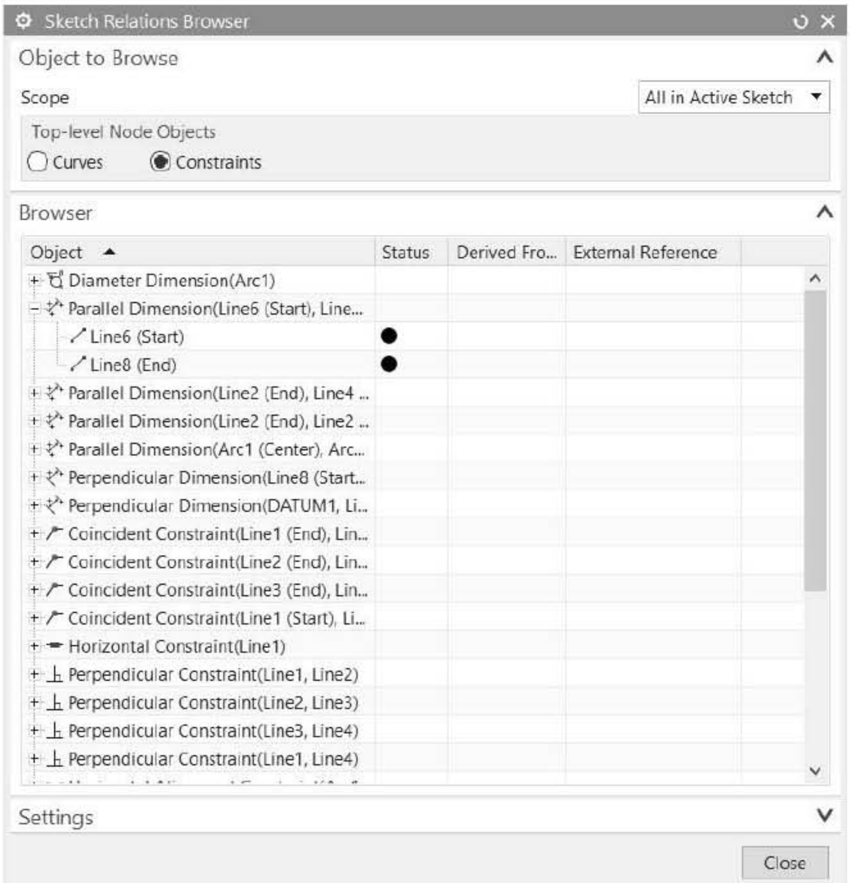
Sketch Relations Browser

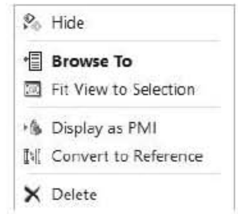
The **Sketch Relations Browser** dialog helps to view all the constraints in the sketch, their status, and elements associated with them. Activate the **Sketch Relations Browser** (On the ribbon, click **Home > Direct Sketch > More > Relations Browser**). On the **Sketch Relations Browser** dialog, select **Scope > All in Active Sketch** to view all the constraints and dimensions in the active sketch. Next, select **Top-level Node Objects > Curves** to display all the curves available in the sketch in the form of nodes. Next, expand each curve node to view all the constraints related to it. The **Status** column displays the status of each curve: **Fully Constrained** ● , **Partially Constrained** ◐ ⊗ , or **Over Constrained**. Right click on the curve node and select **Fit View to Selection**; the select curve is zoomed in the graphics window.





Select **Top-level Node Objects > Constraints** to display all the constraints available in the sketch in the form of nodes. Next, expand each constraint node to view the curves related to it. Right click on a constraint and notice a shortcut menu with different options. These options are self-explanatory.



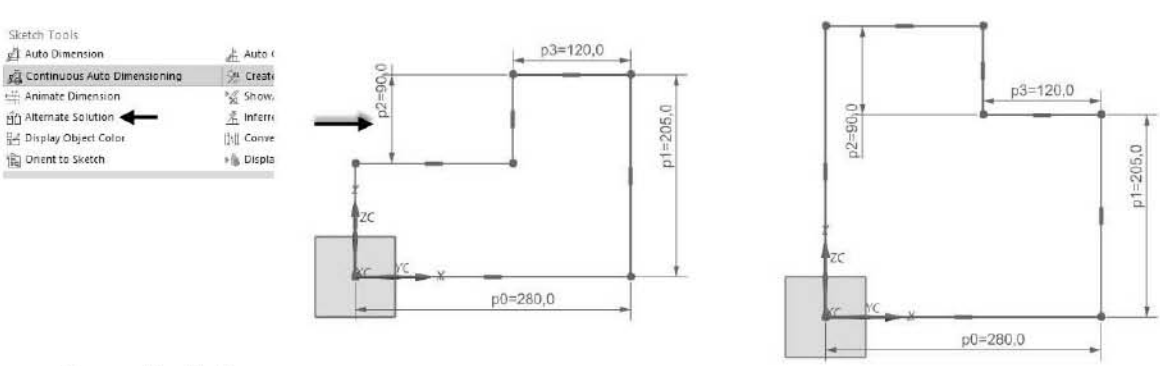


Shortcut Menu

Close the **Sketch Relations Browser** dialog.

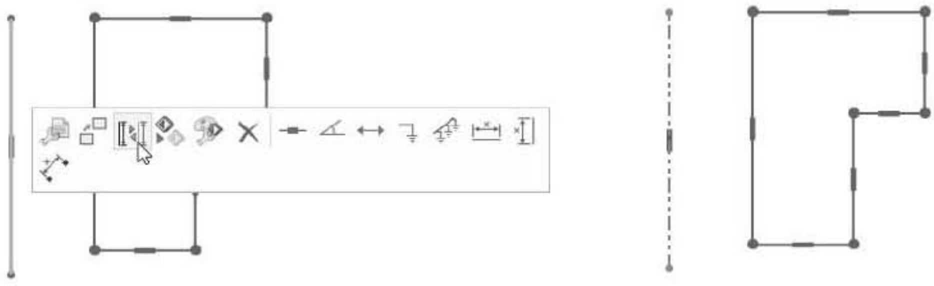
Alternative Solution

This command gives alternate solutions for different dimensions and constraints applied between sketch elements. For example, to change the side of the linear dimension shown in figure, activate the **Alternate Solution** command (On the ribbon, click **Home > Direct Sketch > More > Alternate Solution**). Click on the linear dimensionto see the alternate solution. Likewise, alternate solutions for geometric constraints can be seen.



Convert to Reference

This command converts a sketch element into a reference element. Reference elements support you to create a sketch of desired shape and size. To convert a sketch element to the reference element, click on it and select **Convert to Reference** on the contextual toolbar. It can also be converted back to a sketch element by clicking on it and selecting **Convert to active**.



1.2.2 Correction commands in 2D part design

Only one sketch can be active at a time. While a sketch is active, NX adds any geometry created to that sketch. How to activate a sketch depends on whether the sketch is internal or external.

Internal Sketches

- In the **Part Navigator**, right-click the owning feature and choose **Edit Sketch**.
- In the **Part Navigator** or the graphics window, double-click the owning feature. On the feature dialog box, click the **Sketch Section** step to activate the Sketch task environment.

External Sketches

From Modeling, external sketches can be opened in a number of ways, depending on the work preference:

- Double-click a sketch curve in the graphics window.
- From the **Part Navigator**:
    - Double-click a sketch feature.
    - Right-click a sketch and choose **Edit**.

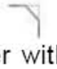
Edit Multiple Sketches

Multiple sketches can be edited when just one model update is required. To edit multiple sketches, it is required to:

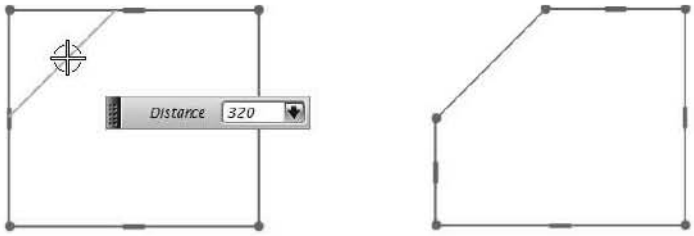
- Open a sketch in the **Sketch task environment** .

- Select a sketch name from the **Home** tab→**Sketch** group→**Sketch Name** Drop-down list.

To edit subsequent sketches, double-click a sketch in the **Part Navigator** or select another sketch name from the **Sketch Name** list.

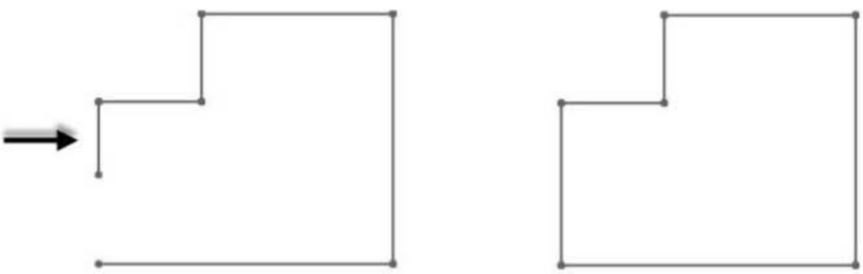
1.2.2.1 The Chamfer command 

This command replaces a sharp corner with an angled line. Activate this command (On the ribbon, click **Home > Direct Sketch > Chamfer**) and select the elements' ends to be chamfered. Type the chamfer angle inthe **Distance** box and press Enter.



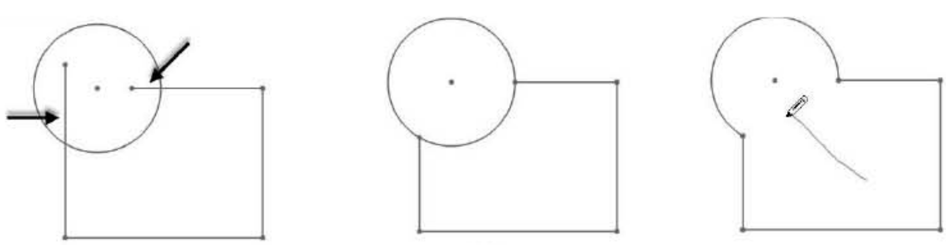
1.2.2.2 The Quick Extend command

This command extends elements such as lines, arcs, and curves until they intersect another element called the boundary edge. Activate this command (On the ribbon, click **Home > Direct Sketch > Quick Extend**) and click on the element to extend. It will extend up to the next element.



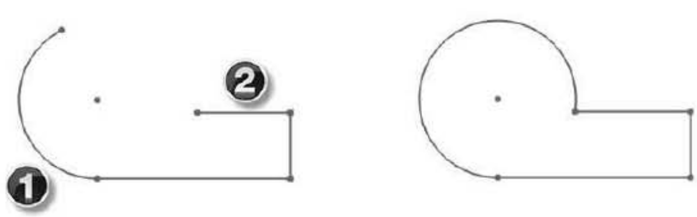
1.2.2.3 The Quick Trim command

This command trims the end of an element back to the intersection of another element. Activate this command (On the ribbon, click **Home > Direct Sketch > Quick Trim**) and click on the element or elements to trim. It is also possible to drag the pointer across the elements to trim.



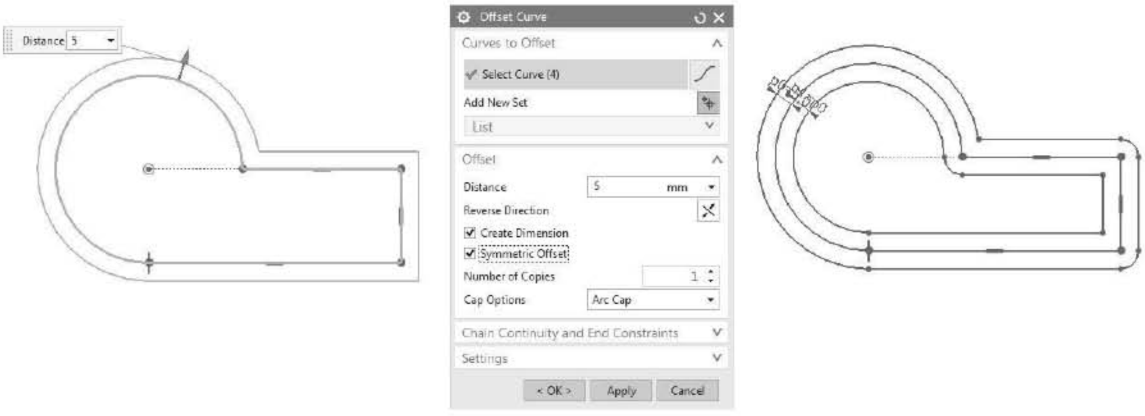
1.2.2.4 The Make Corner command

This command trims and extends elements to form a corner. Activate this command (On the ribbon, click **Home > Direct Sketch > Make Corner**) and select two intersecting elements. The elements will be trimmed and extended to form a closed corner.



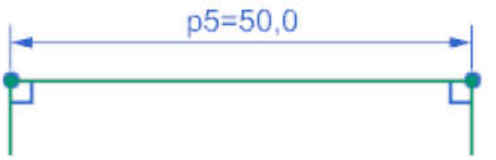
1.2.2.5 The Offset Curve command

This command creates a parallel copy of a selected element or chain of elements. Activate this command (On the ribbon click **Home > Direct Sketch > Offset Curve**) and select an element or chain of elements to offset. After selecting the element(s), type a value in the **Distance** box. On the **Offset Curve** dialog, click the **Reverse Direction** button to reverse the side of the offset. Check the **Symmetric Offset** option to create a parallel copy onboth sides. Set the **Cap Options** to **Arc Cap** to create arcs at the corners. On the dialog, click **OK**. The parallel copyof the elements will be created.



1.2.3 Measuring tools in 2D design

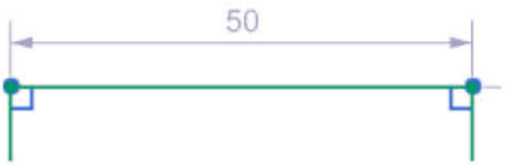
1.2.3.1 Driving dimensions



Driving dimensions control the design intent in the sketch. These dimensions drive the position, shape, and size of the sketch.

Each dimension creates an expression that can be edited.

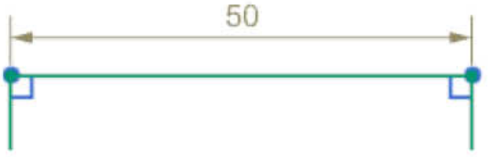
1.2.3.2 Automatic dimensions



Automatic dimensions show where design intent has not been added. NX creates automatic dimensions by default as you sketch.

These dimensions can be converted to driving dimensions. As The design intent is added, NX automatically deletes redundant auto dimensions.

1.2.3.3 Reference dimensions



Reference dimensions display information only. They do not constrain the sketch.

It is possible to create reference dimensions or convert driving dimensions to reference dimensions. There are no specific commands to create reference dimensions. To create them, use the **Reference** option in any dimension command.

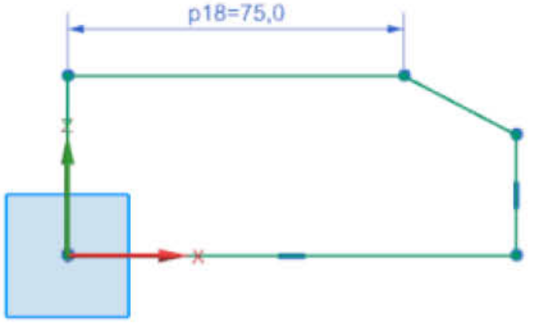
1.2.3.4 Dimensional constraint terminology

To correctly add the design intent to a sketch, the terminology and the process of creating dimensional constraints must be understood.

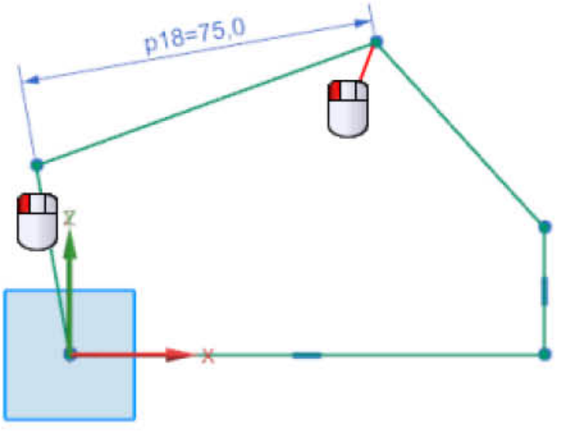
These terms apply to all NX sketch dimension types.

1.2.3.5 Measurement Method

The measurement method controls the design intent of a dimensional constraint. The design intent may not be apparent until the geometry of the sketch is changed. For example, the linear dimension shown could be created using one of three different measurement methods.

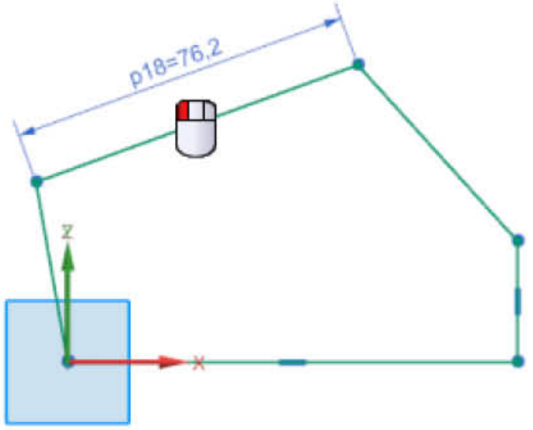


This linear dimension is created using one of three measurement methods shown below. When the geometry is changed, the dimension appears as one of the three conditions shown, depending on the measurement method used.



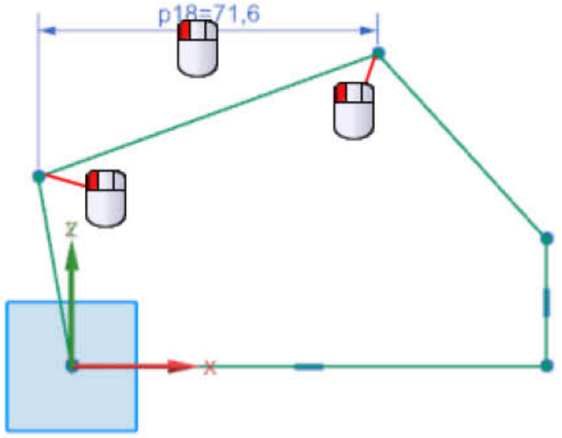
Perpendicular

Create a perpendicular dimension by selecting a line and a point as reference objects.



Point-to-Point

Create a point to point dimension between any two points or line end points. When you select a line, NX initially infers the two end points of the line as reference objects.



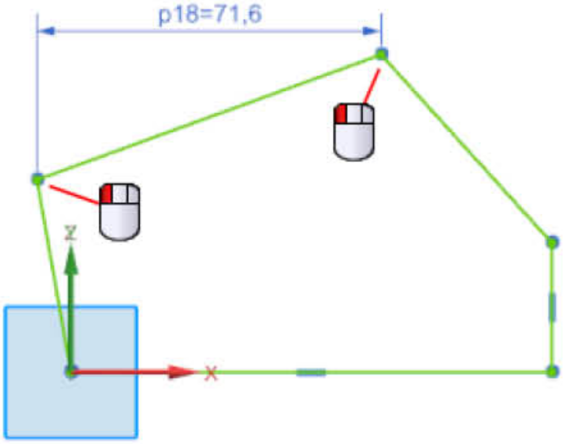
Horizontal

A horizontal or vertical dimension uses two points as reference objects, but measures the distance vertically or horizontally relative to the sketch coordinate system. NX infers horizontal or vertical by where the place of the dimension text is clicked.

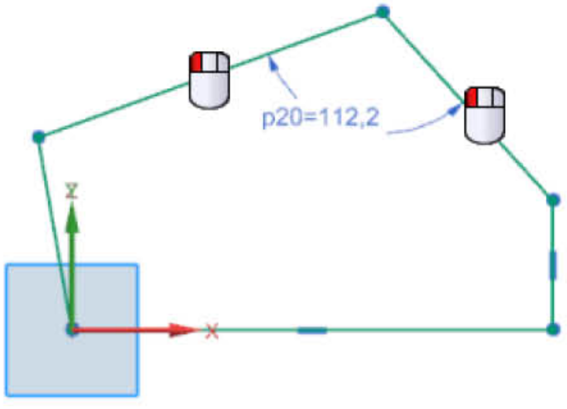
It is possible to explicitly choose a measurement method when creating the dimension, NX can infer a measurement method itself. NX infers a measurement method based on what is selected for the reference objects, and where the origin is located.

1.2.3.6 Reference Objects

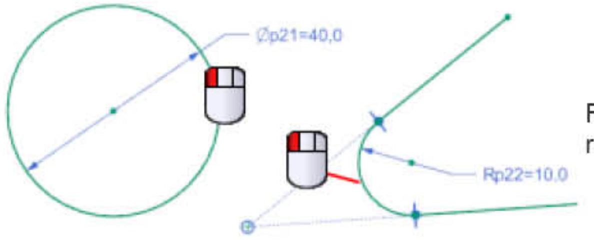
Sketch dimensions create a constraint between reference objects in the sketch. Reference objects can be curves or points.



Point-to-point, horizontal, vertical, and perpendicular dimensions require two reference objects. These can be two points, or a line and a point.



Angular dimensions require two lines as reference objects.



Radial and diametral dimensions use only one reference object.

1.2.3.7 Inferred reference objects

When the first reference object is selected, NX may initially infer the second reference object, depending on the measurement method and the object selected. For example, if a line is selected, NX may infer that a dimension between the two end points of the line need to be created.

-

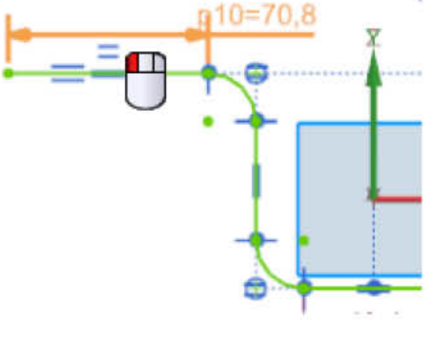
-

1.2.3.8 Inferred selection gesture

When a second reference object that is different from what NX infers needs to be selected, an inferred selection gesture can be used.

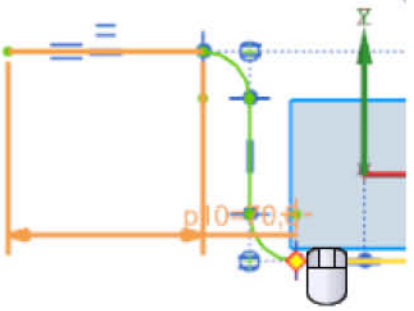
1. When a line is selected as the first reference object, NX may infer that the two reference objects for the dimension are the two end points.

   The **References** group of the dialog box indicates that two reference objects are selected, and the graphics window previews the dimension between the two inferred reference objects.

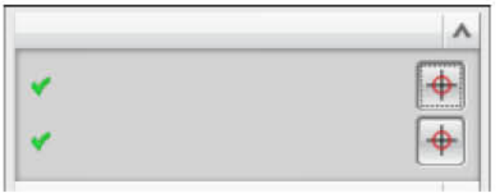


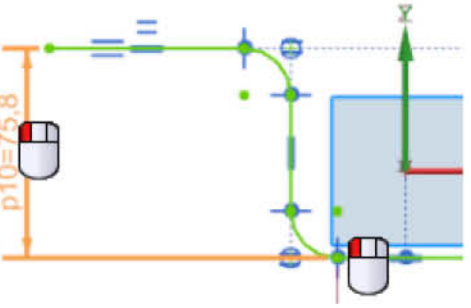
2. To infer a different type of dimension, hover the cursor over the object you would like to select for the second reference object.

   The **References** group of the dialog box highlights the second reference object selection line.



3. Click to select this alternate second reference object, then click to locate the dimension text.



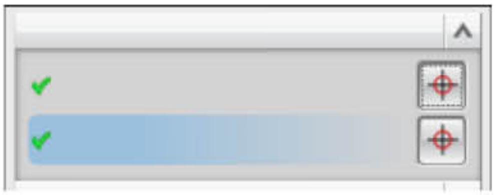


1.2.3.9 Reselecting reference objects

The first and second reference objects can be explicitly selected in order by clicking the selection line in the dialog box.

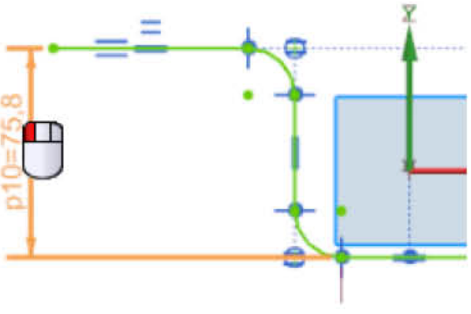
Selected reference objects can be changed at any time until the dimension is created by locating the text.

When a dimension is edited, the two reference objects can be reselected to reattach the dimension to different reference objects.



1.2.3.10 Origin

The origin of a dimension is the location of the dimension text. To change this location, drag the dimension text.



Driving dimensions

Most sketch dimensions in NX Modeling are added for the purpose of controlling the shape of the sketch. Reference dimensions may also be created to measure distance without driving the geometry.

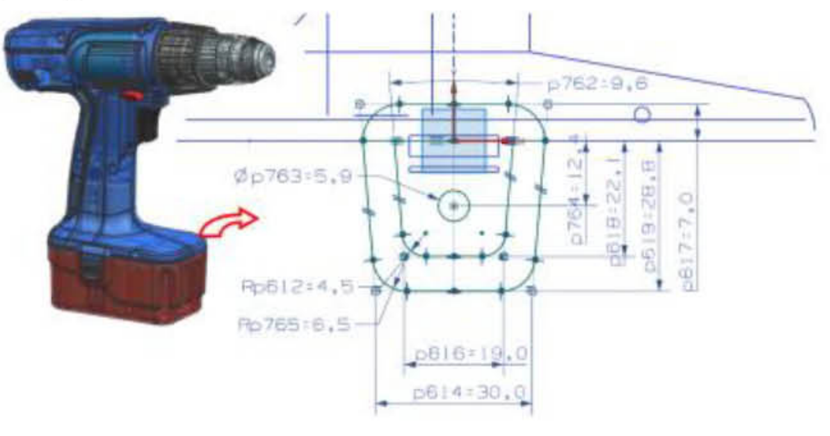
1.2.3.11 Dimension commands

There are five dimension commands. Each command supports a related family of measurement methods. When a dimension is edited, the measurement method can be changed between the family of measurement methods.

Dimension Command	Description	Measurement Methods
Rapid Dimension	Creates a dimensional constraint between one or two objects selected. This command will infer one of these measurement types based on the objects selected, or one of these dimension measurement methods can be selected explicitly. A linear, radial, or angular dimension is created.	• **Horizontal** • **Vertical** • **Point to Point** • **Perpendicular** • **Cylindrical** • **Angular** • **Radial** • **Diametral**
Linear Dimension	Creates a dimensional constraint between the objects selected using one of these dimension measurement methods. When a dimension is edited using one of these measurement methods, the measurement method can be changed between the methods listed.	• **Horizontal** • **Vertical** • **Point to Point** • **Perpendicular** • **Cylindrical**
Radial Dimension	Creates a radial or diametral dimensional constraint on an arc or circle selected. When a radial or diametral dimension is edited, the measurement method can be changed between the methods listed.	• **Radial** • **Diametral**
Angular Dimension	Creates an angular dimensional constraint between two lines you select.	This measurement method cannot be changed to another type.

Rapid Dimension

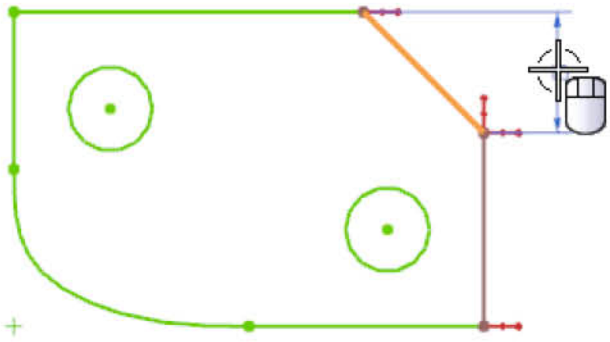
Use the **Rapid Dimension** command to create a dimensional constraint. NX infers the dimension type based on the objects you select and the location of the cursor.



Command Finder	**Rapid Dimension**

Linear Dimension

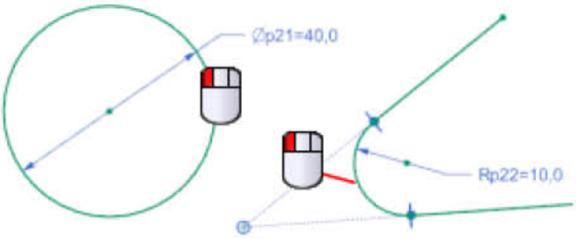
Use the **Linear Dimension** command to create a dimensional constraint. NX infers the dimension type based on the objects selected and the location of the cursor.



Command Finder	Linear Dimension

Radial Dimension

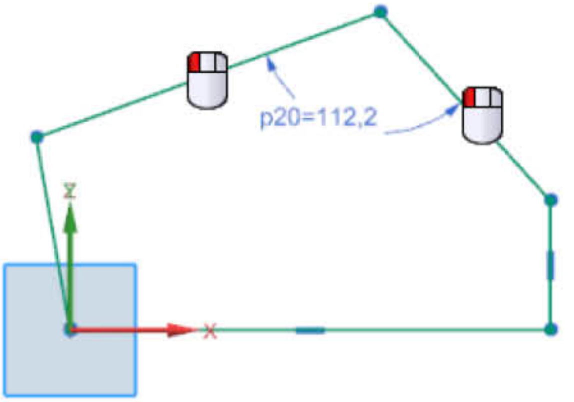
Use the **Radial Dimension** command to create a radial or diametral dimensional constraint. NX infers the dimension type based on the object selected.



Command Finder	Radial Dimension

Angular Dimension

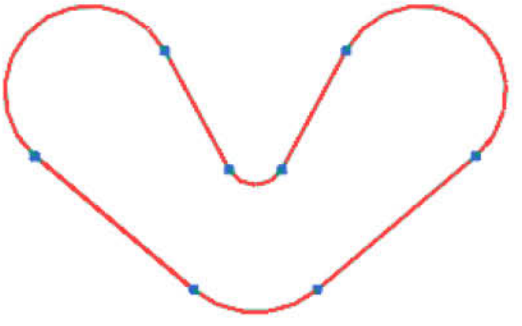
Use the **Angular Dimension** command to create an angular dimensional constraint between two lines.



Command Finder	Angular Dimension

Perimeter Dimension

Use the **Perimeter** command to constrain the total lengths of selected lines and arcs in an open or closed profile. Ellipses, conics or splines cannot be selected.

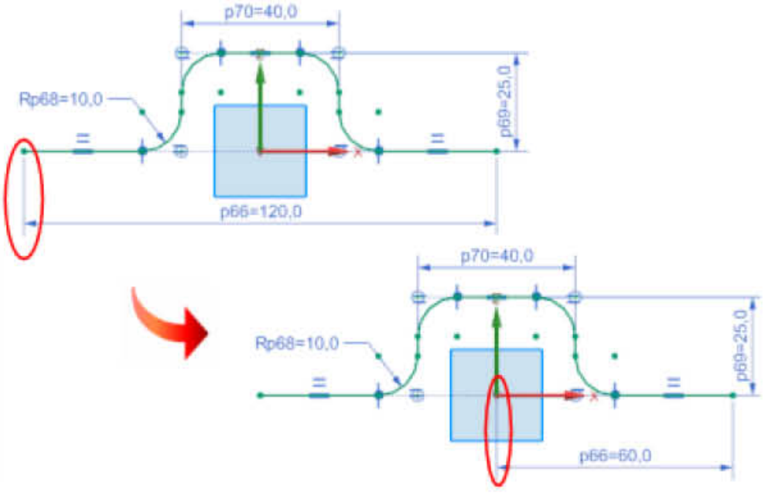


Command Finder	Perimeter Dimension

Edit dimension associativity

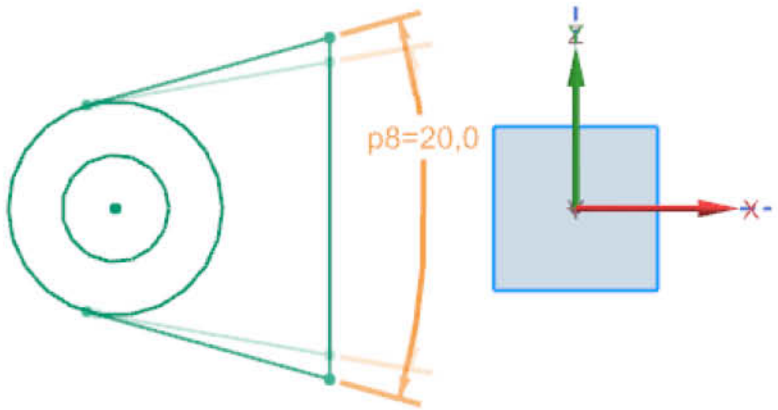
A dimension can be edited to reattach the dimension leader lines to different reference objects using the same dialog box used to create the dimension. When reattaching a dimension, it is possible to:

- Retain the expression value and resize the target geometry to match it.
- Change the expression value to match the target geometry.



Animate Dimension

Use the **Animate Dimension** command to dynamically display the effect of varying a given dimension over a specified range. Any geometry affected by the selected dimension is also animated.



Animate does not change the sketch dimensions. When the animation is finished, the sketch returns to its original state.

Tip You can also use the **Edit Sketch Parameters** command to dynamically animate dimensions using a slider bar.

Command Finder	**Animate Dimension**

Auto Dimension

Use the **Auto Dimension** command to create dimensions on selected curves and points according to a set of rules.

The following rules can be applied in any order.

- Create Horizontal and Vertical Dimensions on Lines
- Create Dimensions to Reference Axes
- Create Symmetric Dimensions
- Create Length Dimensions
- Create Adjacent Angles

Driving and Automatic Dimensions

Driving

Creates a dimension based on an expression. Modify the dimension expression value to size the geometry.

Automatic

Automatic dimensions show where design intent to a sketch has not been entered.

- When dragging sketch curves, automatic dimensions are relaxed, allowing the sketch to be shaped. After the drag, the new shape will be constrained by the automatic dimensions again.
- If a constraint that conflicts with an automatic dimension is added, the automatic dimension will be deleted.
- Automatic dimensions can be converted into driving or reference dimensions.

All dimensions are displayed in the **Part Navigator** when **Timestamp Order** is turned off in the **Unused Items→Sketch→Curves and Dimensions→Dimensions** folder.

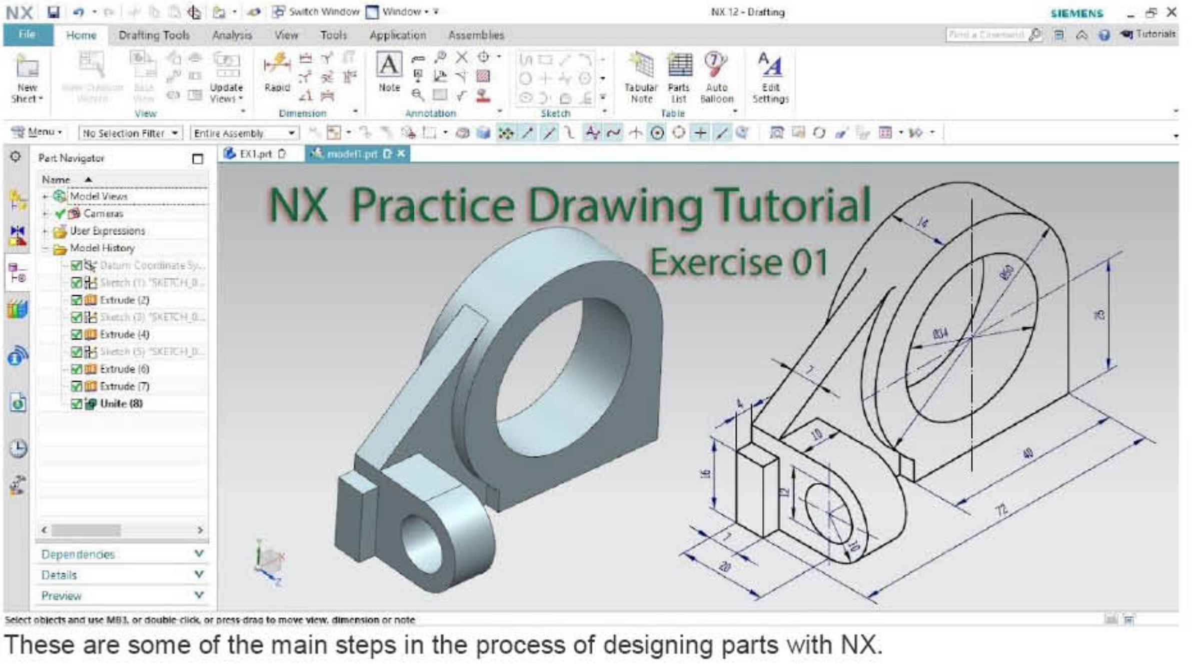
In Modeling, use this command to assist in creating a fully constrained sketch by removing all degrees of freedom from the selected curves.

In Drafting, use this command to fully dimension selected sketch curves in a drawing.

Command Finder	**Auto Dimension**

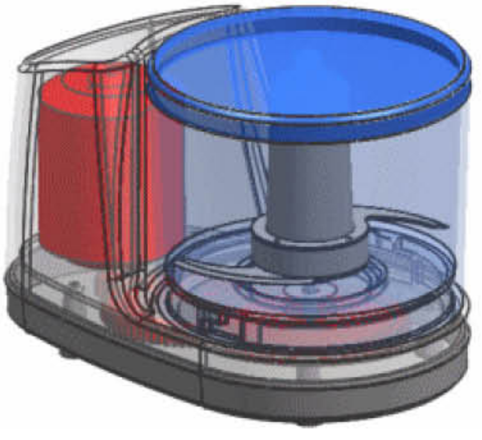
1.3. Design 3D with NX

The modeling process to design a part is the same whether it is the design of stand-alone parts or parts within an assembly. The decisions made at each step depend on the design goals.



These are some of the main steps in the process of designing parts with NX.

Start with a new file.

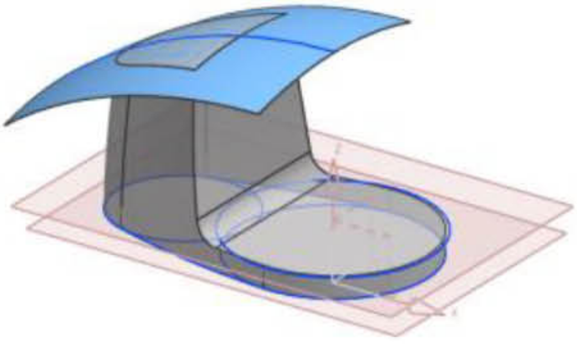


1. Create an empty file for the part model.
2. Add the empty file to the assembly as a new component to design the part within an assembly context.

Design parts within an assembly to create proper fit and alignment to other parts, and to avoid unintentional interferences.

Define the modeling strategy.

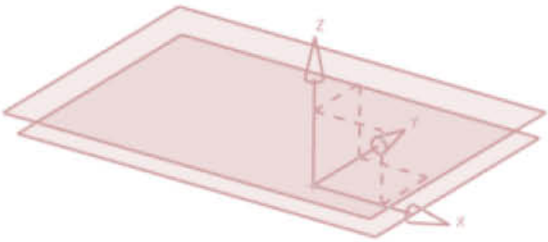
Decide if the final part will be a solid body or a sheet body. This will impact



the modeling strategy of which features to build first.

- Solid bodies are preferred for most models because they provide an unambiguous definition of the volume and mass.
- Sheet bodies are sometimes used for manufacturing or simulation. They may also be used as trimming tools for solid body models

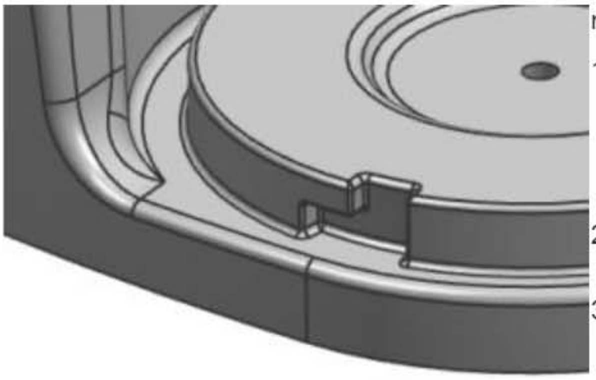
Create datums.



Create datum coordinate systems and datum planes to position modeling features.

These datums form the beginning of a chain of associativity for the features added next.

Create features.



Create features according to the modeling strategy.

1. Start with design features such as extrude, revolve, or sweep to define basic shapes. These features typically use sketches to define sections.
2. Continue adding other features to design the model.
3. Finish with detailed features such as edge blends, chamfers, and draft to add the final details.

1.3.1 Switching from 2D drawing to 3D

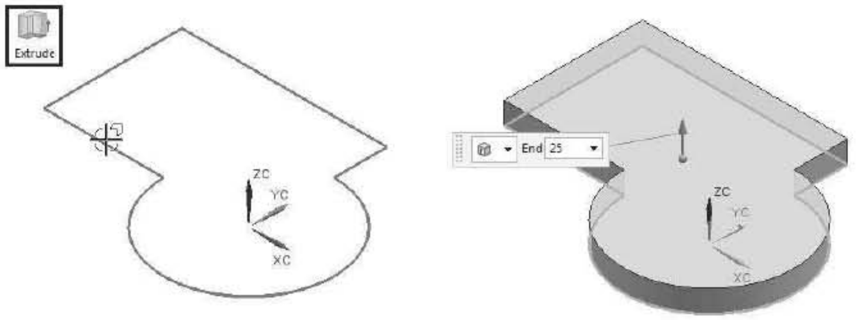
Extrude and revolve features are used to create basic and simple parts. Most of the time, they form the base for complex parts as well. These features are easy to create and require a single sketch. Now, let's find out the commands to create these features.

The topics covered in this part are:

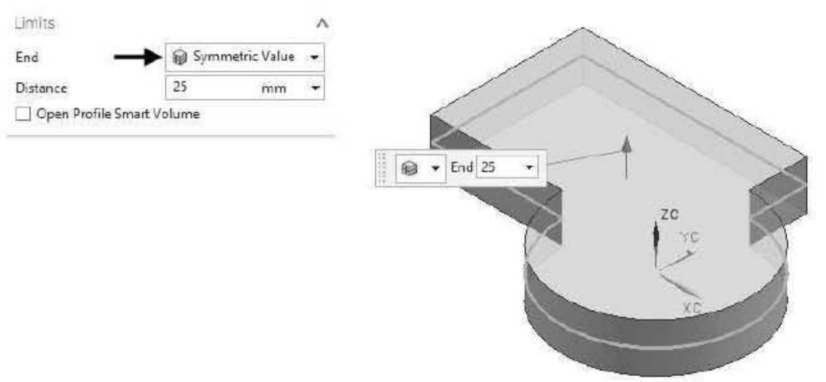
- *Constructing Extrude and Revolve features in the Modeling template*
- *Creating Reference Planes*
- *Additional Options in the Extrude and Revolve commands*

1.2.1.1 Extrude Features

Extrude is the process of taking a two-dimensional profile and converting it into 3D by giving it some thickness. A simple example of this would be taking a circle and converting it into a cylinder. Once a sketch profile or profiles wanted to *Extrude* have been created, activate the **Extrude** command (On the ribbon click **Home > Feature > Extrude**). Click on the sketch profile to add thickness to the sketch. Type-in a value in the **End** box and press Enter to create the *Extrude* feature.



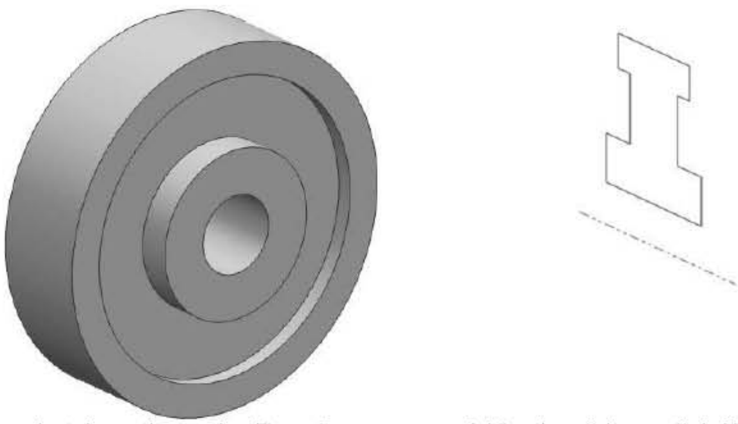
Use the **Symmetric Value** option on the dialog to add equal thickness on both sides of the sketch.



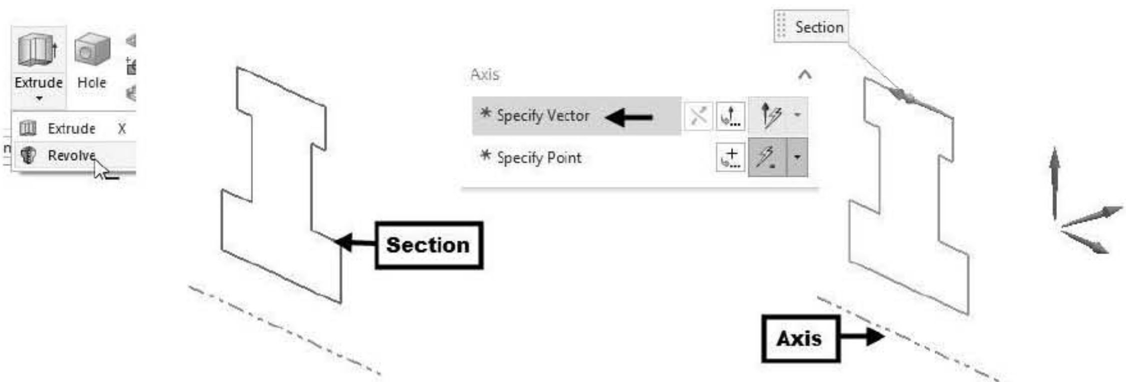
On the dialog, click **OK** to complete the *Extrude* feature.

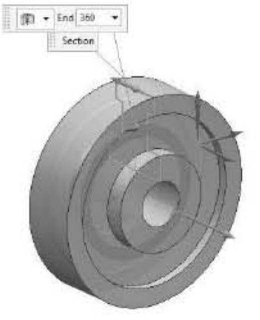
1.3.1.2 Revolve Features

Revolve is the process of taking a two-dimensional profile and revolving it about a centerline to create a 3D geometry (shapes that are axially symmetric). While creating a sketch for the *Revolve* feature, it is important to think aboutthe cross-sectional shape that will define the 3D geometry once it is revolved about an axis. For instance, the following geometry has a hole in the center. This could be created with a separate *Cut* or *Hole* feature. But in orderto make that hole part of the *Revolve* feature, it is necessary to sketch the axis of revolution so that it leaves a space between the profile and the axis.



After completing the sketch, activate the **Revolve** command (On the ribbon, click **Home > Feature > Design Feature Drop-down > Revolve**). Click on the sketch to define the section of the *Revolve* feature. On the dialog, click **SpecifyVector** under the **Axis** section. Click on a line to define the axis of revolution. The sketch will be revolved at a full turn of 360. To enter an angle of revolution, type a value in the **End** box attached to the preview andpress Enter. On the dialog, click **OK** to complete the *Revolve* feature.

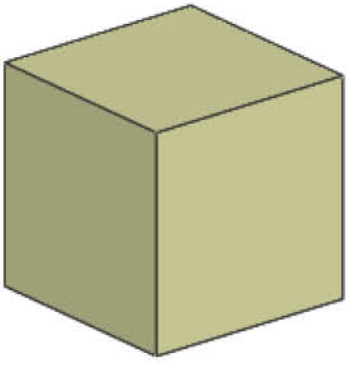




1.3.2 Basic commands in 3D part design

1.3.1.1 Block

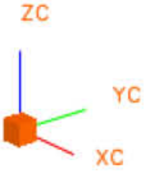
Use this command to create basic block solid bodies. Blocks are associated with their positioning objects.



It is possible to use one of three methods (types) to create a block.

Create a block using a point and edge lengths

1. Choose **Home** tab→**Feature** group→**Block** .
2. In the **Type** group, select **Origin and Edge Lengths** from the Type list.
3. Select an object to infer an origin point or specify a point using the **Specify Point** options in the **Origin** group. Snap Point options are available.

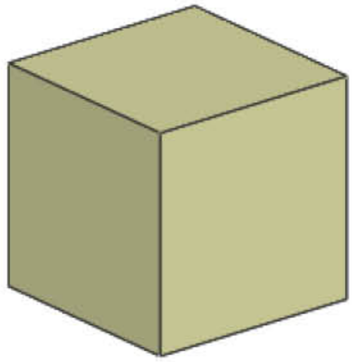


4. Type the block's measurement values in the **Dimensions** group. All values must be positive.
5. (Optional) If the block intersects another solid, you can choose a Boolean option from the Boolean list.

   Select Body  becomes available to specify a target for the Boolean operation.

6. Click **OK** or **Apply** to create the block.

   The block is created from the origin point, with edges parallel to the axes of the WCS.



1.3.1.2 Cylinder

Use this command to create basic cylindrical solid bodies. Cylinders are associated to their positioning objects.

It is possible to use one of two methods (types) to create a cylinder.

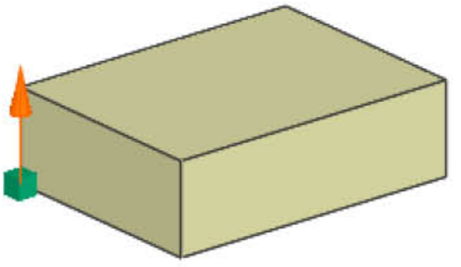
Prerequisite	Available with the **Advanced with full menus** and **Essentials with full menus** roles.
Command Finder	**Cylinder**

Create a cylinder using an axis, a diameter, and a height

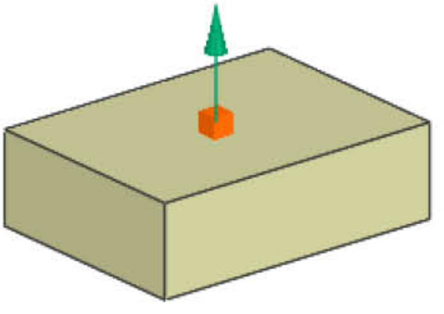
1. Choose **Home** tab→**Feature** group→**Cylinder** .
2. From the **Cylinder Type** list, select **Axis, Diameter, and Height**.

   Note A vector, direction, and origin point for the cone's axis are specified by default. If you want to change the defaults, see steps 3, 4, and 5.

3. Select an object to infer a vector, or specify a vector using the **Specify Vector** options in the **Axis** group.

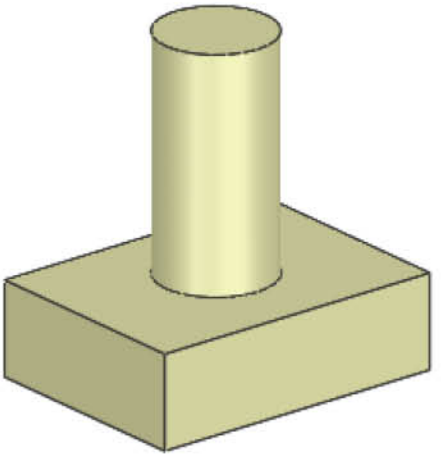


4. (Optional) Create a cylinder in the opposite direction by clicking **Reverse Direction** .
5. Click **Specify Point** and select an object to infer a point, or use the **Specify Point** options to specify one. Snap Point options are available.



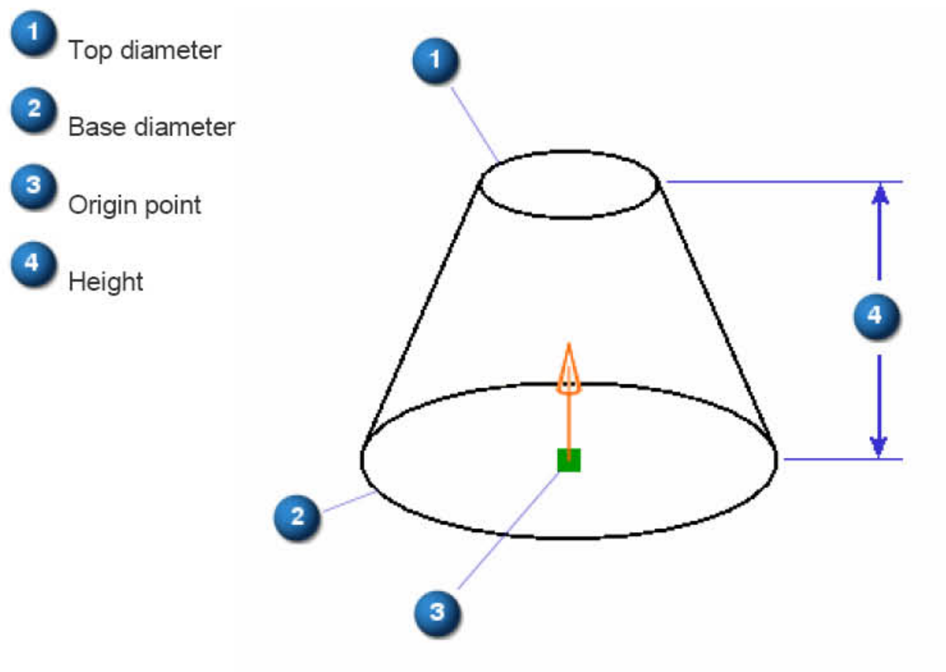
6. In the **Dimensions** group, type values for the diameter and height of the cylinder.
7. (Optional) Choose a Boolean option from the Boolean list if the cylinder intersects another solid.

   Select Body  allows selecting a body for the **Unite**, **Subtract**, or **Intersect** Boolean operation.
8. Click **OK** or **Apply** to create the cylinder.



1.3.1.4 Cone

Use this command to create basic conical solid bodies. Cones are associative to their positioning objects.



It is possible to use one of five methods (types) to create a cone.

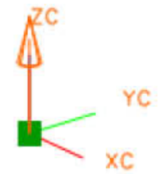
Prerequisite	Available with the **Advanced with full menus** and **Essentials with full menus** roles.
Command Finder	**Cone**

Create a cone using arc diameters and height

1. Choose **Home** tab→**Feature** group→**Cone** .
2. In the **Type** group, select **Diameters and Height** from the Type list.

   Note A vector, direction, and origin point for the cone's axis are specified by default. To change the defaults, see steps 3, 4, and 5.

3. Select an object to infer a vector, or specify a vector using the **Axis** group **Specify Vector** options.
4. (Optional) Create a cone in the opposite direction by clicking **Reverse Direction** .
5. Click **Specify Point** and select an object to infer a point, or use the **Specify Point** options to specify one. Snap Point options are available.



   For this example, the default WCS origin point and the Z-axis direction are used.

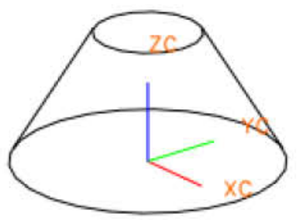
6. In the **Dimensions** group, type values for the following dimensions:

- **Base Diameter**
- **Top Diameter**
- **Height**

For this example, the **Base Diameter** is 50, the **Top Diameter** is 20 and the **Height** is 25.

7. (Optional) If the cone intersects another solid, it is possible to select a Boolean option from the Boolean list.

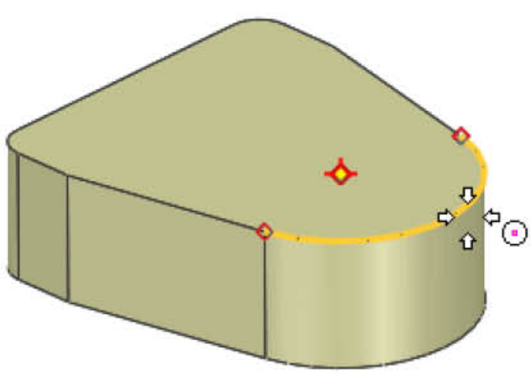
   Select Body  becomes available to specify a target for the Boolean operation.
8. Click **OK** or **Apply** to create the cone.

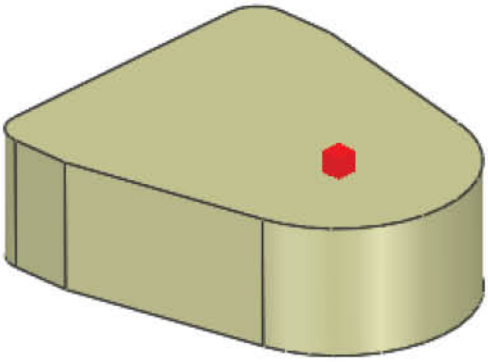


1.3.1.5 Sphere

Create a sphere using a center point and a diameter

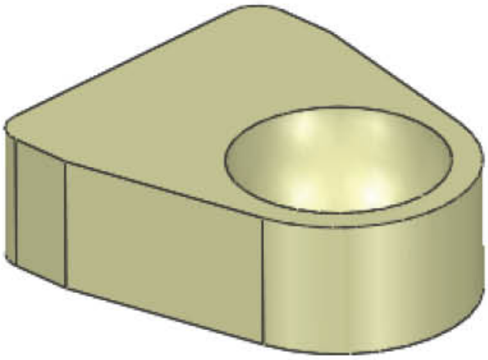
1. Choose **Home** tab→**Feature** group→**Sphere** .
2. In the **Type** group, select **Center Point and Diameter** from the Type list.
3. Select an object to infer a center point for the sphere or create a center point using the **Specify Point** options in the **Center Point** group.





4. In the **Dimension** group, in the **Diameter** box, type a value for the diameter of the sphere.
5. (Optional) If the sphere intersects another solid, select a Boolean option from the Boolean list.

   Select Body  becomes available to specify a target for the Boolean operation.
6. Click **OK** or **Apply** to create the sphere.

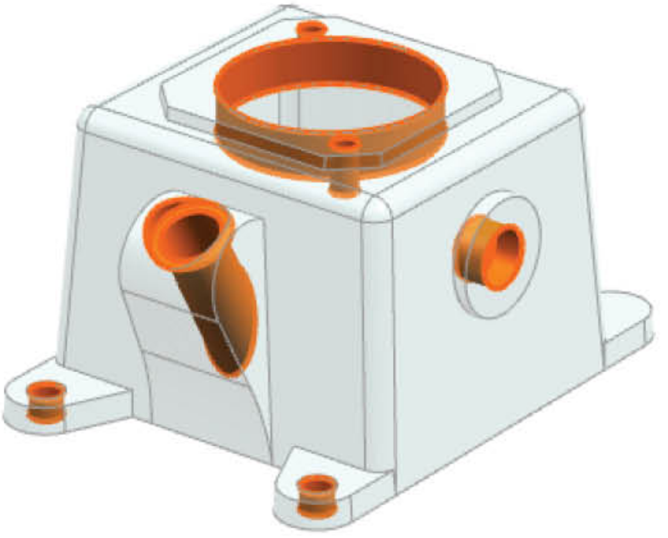


Sphere created using the Subtract Boolean

1.3.1.6 Hole

Use the **Hole** command to add the following types of hole features in a part or assembly:

- General holes (simple, counterbored, countersunk, or tapered form)
- Drill Size holes
- Screw Clearance holes (simple, counterbored, or countersunk form)
- Threaded holes
- Holes on non-planar faces
- Holes through multiple solids as a single feature
- Multiple holes as a single feature



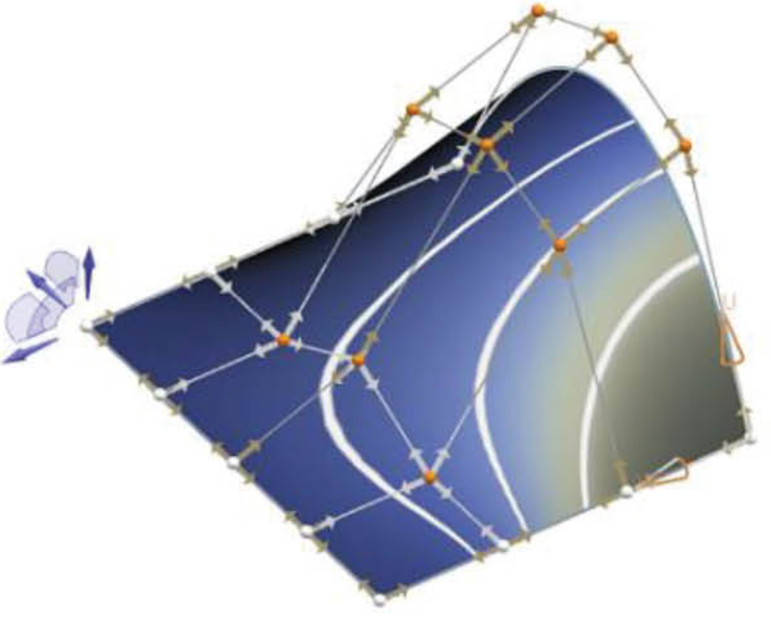
Application	Modeling
Command Finder	**Hole** 

.

1.3.3 Advanced commands in 3D part design (Surface Modeling)

1.3.3.1 X-Form

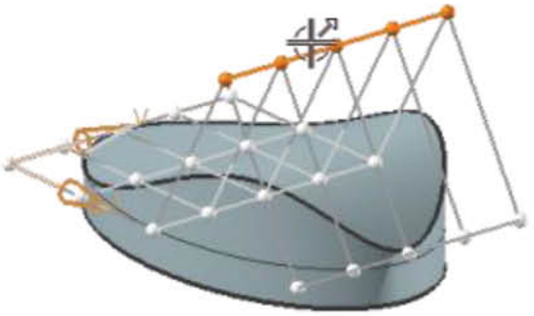
Use the **X-Form** command to edit surfaces or spline curves by dynamically manipulating the pole locations.



It is possible to do the following:

- Select any type of face (B-surface or non-B-surface).

- Select individual or multiple poles using standard NX selection methods such as selecting inside a rectangle.
- Select rows of poles by selecting the polyline that connects pole handles.



The **X-Form** command also includes advanced options to edit a B-surface.

It is possible to do the following:

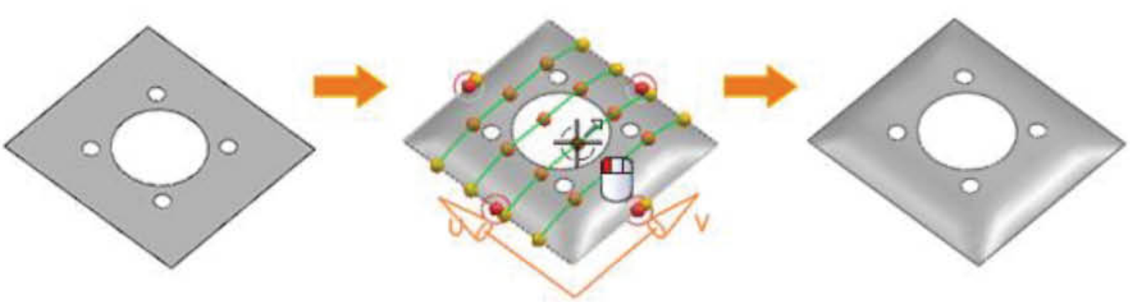
- Increase or decrease the number of poles and patches.
- Insert knot points.
- Proportionally move adjacent poles.
- Maintain continuity at surface edges when editing a surface.
- Lock regions of a surface to remain constant when editing the surface.
- Identify faces with symmetry and offset conditions. The modifications made to the selected face will automatically be applied to the faces that meet these conditions.

The **X-Form** command creates an X-Form feature in the **Part Navigator** when the **Associative Freeform Editing** is enabled in Modeling Preferences.

Application	Modeling, and Shape Studio
Command Finder	**X-Form**

1.3.3.2 I-Form

Use the **I-Form** command to dynamically modify a face by adding isoparametric curves to it and by editing their direction.



It is possible to select and modify the control poles or polygons of any type of face (B-surface and non-B-surface) without having to extract, untrim, refit, replace, or otherwise convert them.

Changes made to a face are cumulative as its shape is altered. For example, a face can be modified by first changing its shape along its U-direction isoparametric curves, and then modified further by changing its V-direction isoparametric curves.

Note The **I-Form** command creates an I-Form feature in the **Part Navigator** when the **Associative Freeform Editing** preference is enabled in the **Modeling Preferences** dialog box on the **Freeform** tab.

Application	Shape Studio, Modeling
Command Finder	**I-Form** 

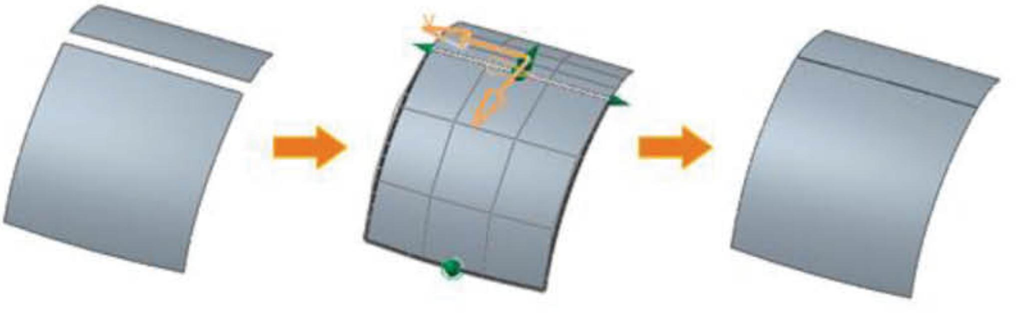
1.3.3.3 Match Edge

Use the **Match Edge** command to modify a face so it is geometrically continuous with one or more reference objects.

It is possible to:

- Match the edge of a face to the edge of another face, the interior of another face, or a curve. When the **Associative Freeform Editing** preference is enabled, the relationship with the parent object is maintained when the parent object is repositioned.
- Maintain or edit the surface degrees and patches.
- Edit the surface poles while maintaining matching continuity.
- Use Shape Control to influence the magnitude of change in the Matched surface.

This example shows two input surfaces during a **Match Edge** operation, and the matched surfaces.

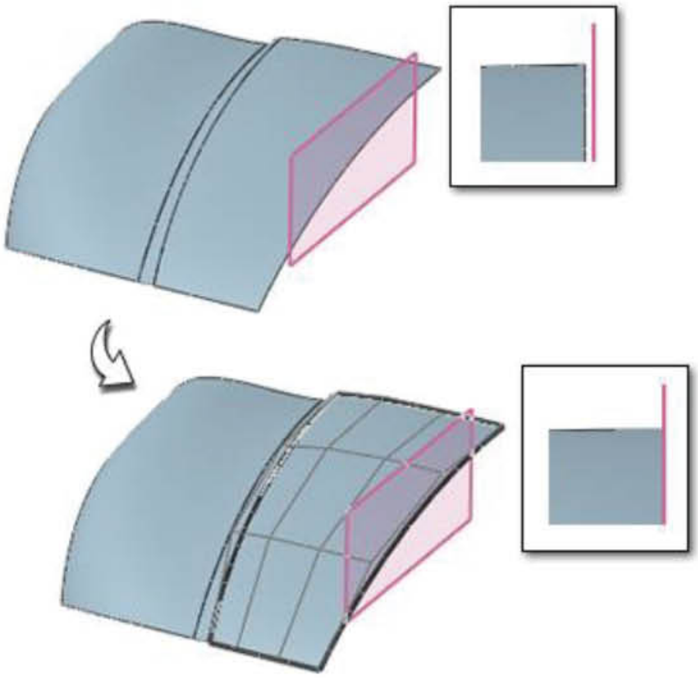


The **Match Edge** command creates a Match Edge feature in the **Part Navigator** when the **Associative Freeform Editing** preference is enabled.

Application	Modeling and Shape Studio
Command Finder	**Match Edge** 

1.3.3.4 Edge Symmetry

Use the **Edge Symmetry** command to match a surface edge to a centerline plane so that the surface will be symmetrical and continuous to its mirror image across the centerline plane.



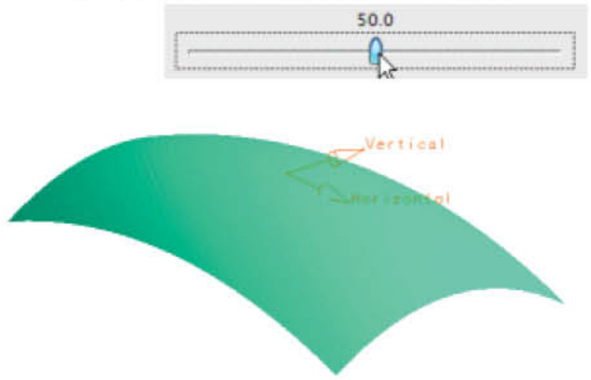
Application	Modeling and Shape Studio
Command Finder	**Edge Symmetry**

1.3.3.5 Deform Surface

Use the **Deform Surface** command to dynamically modify a surface using stretch, bend, skew, twist and shift operations.

This example shows how a surface is deformed using Stretch and Bend pivot controls.

Pivot Control = Horizontal → Stretch



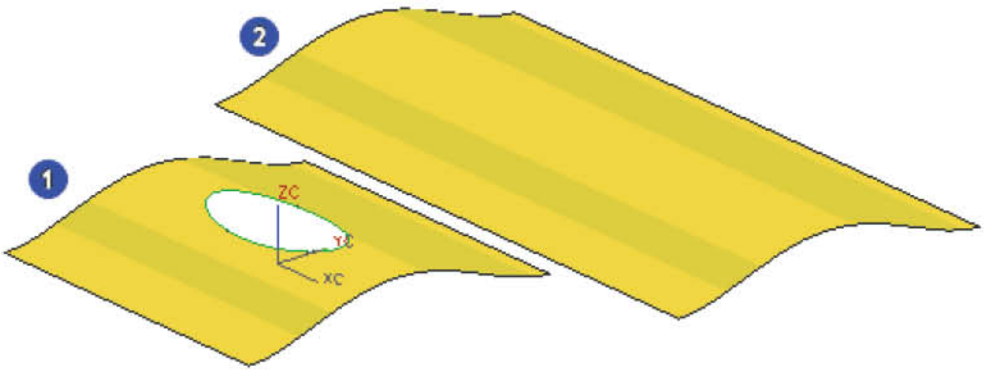
Pivot Control = V-Low → Bend

The command creates a Deform feature in the **Part Navigator** when the **Associative Edit Freeform** preference in the **Modeling Preferences** dialog box is enabled.

Application	Modeling, Shape Studio
Command Finder	**Deform Surface** 

1.3.3.6 Transform Surface

Use the **Transform Surface** command to dynamically scale, rotate, and translate individual B-surfaces.



1 Original sheet body

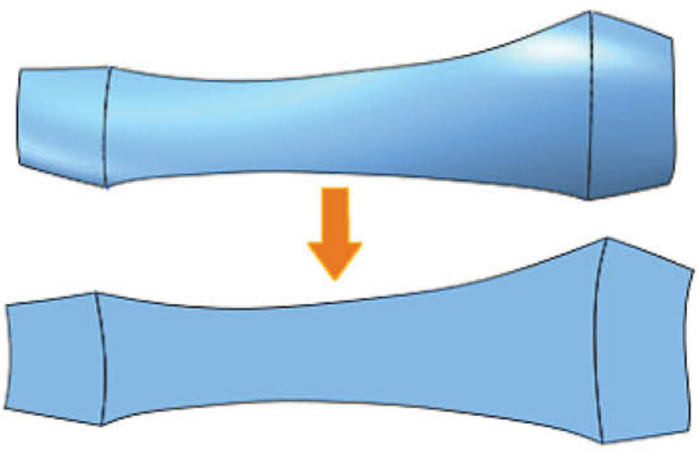
2 Copy of the original sheet body scaled along XC axis and translated along YC axis

Note If the original sheet is chosen to edit, and the selection is parametric, a warning is displayed informing that transforming the surface will result in it becoming unparameterized.

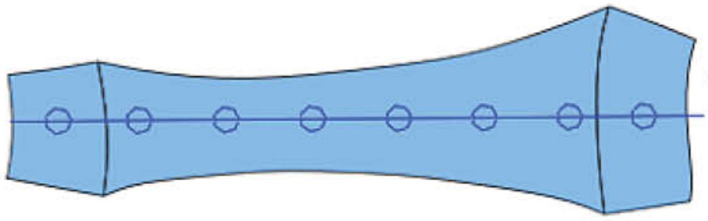
Application	Modeling, Shape Studio
Menu	**Transform Surface** 

1.3.3.7 Flattening and Forming

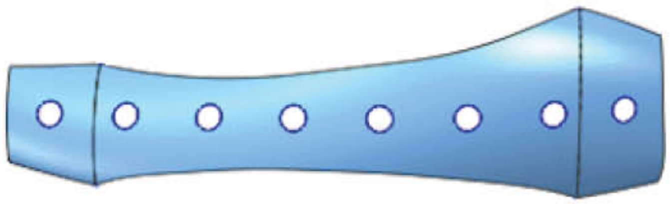
You can unwrap and flatten 3D faces and related curves and points from one or more bodies to a 2D plane, where it may be easier to define and create new curve and point structures. It is possible to then *form* the new curves and points back onto the original 3D faces.



- 3D objects can be flattened to a 2D plane. Object types that can be flattened are:
  - Faces
  - Curves and Points



- New sketch curves can be added to the flattened faces.



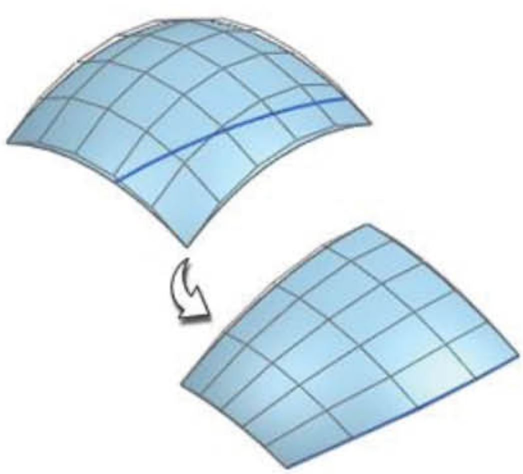
- The Flattened faces, curves, and points, along with any new curves and points specified, can be formed back onto the original 3D faces.
- In this example, openings are trimmed in the faces using the sketch curves.

Application	Modeling and Shape Studio
Command Finder	**Flattening and Forming**

1.3.3.8 Snip Surface

Use the **Snip Surface** command to divide a surface, or to snip off portions of a surface, at specified boundary geometry. **Snip Surface** modifies the underlying pole structure of the target surface.

This example shows a surface snipped along a curve, and the pole structure of both the input surface and the **Snip Surface** feature.



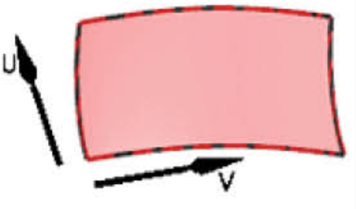
Application	**Modeling** and **Shape Studio**
Command Finder	**Snip Surface**

1.3.3.9 Editing the U and V Directions

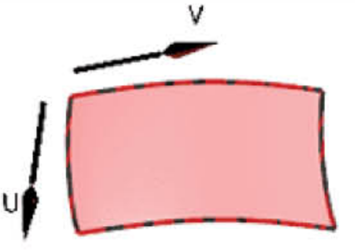
Use the **Edit UV Direction** command to modify the U and V directions of B-surfaces that are not part of a feature.

Any combination of the following options can be used:

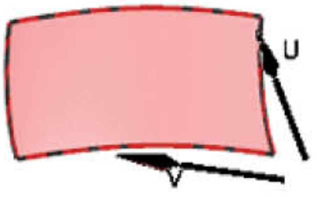
- **Reverse U**
- **Reverse V**
- **Swap U and V**



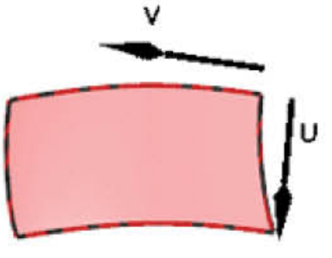
Start



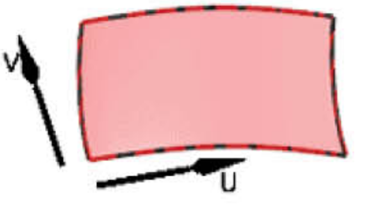
With Reverse U selected



With Reverse V selected



With both Reverse U and Reverse V selected



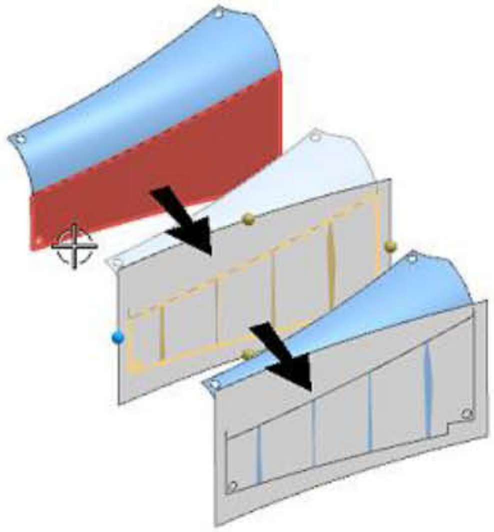
With Swap U and V selected

Application	Modeling and Shape Studio

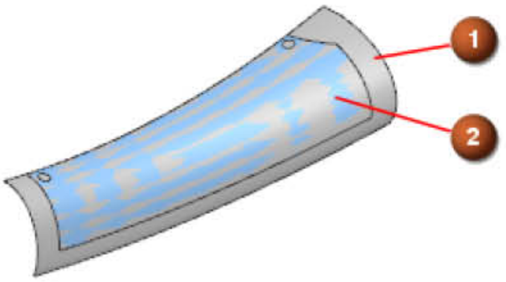
Command Finder	**Edit U/V Direction**

1.3.3.10 Enlarge

Use the **Enlarge** command to create a 4–sided surface feature that is associative to a selected trimmed or untrimmed sheet or face. The size of the feature can be changed by dragging handles or typing U and V percentage values for the four edges.



When creating models using sheets it is good practice to overbuild them, to eliminate downstream solid modeling issues. The **Enlarge** command can be used to do this while retaining the sheet's current parameters. It is also possible to use it to reduce the size of a sheet, to remove degenerate edges, for example.



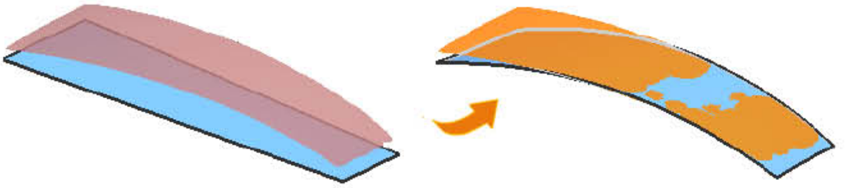
Enlarge may not always follow the complete outline shape of the original surface or its number of sides, because its resize parameters affect only the 4–sided form of the **Enlarge** feature.

1. **Enlarge** feature
2. Originally selected face

Application	Modeling and Shape Studio
Command Finder	**Enlarge**

1.3.3.11 Refit Face

Use the **Refit Face** command to modify an existing surface by changing degree, patch count, or tolerance. A face can also be fitted to target geometry.



Use **Refit Face** to modify existing geometry when the geometry:

- Has a heavy pole structure as a result of previous operations.
- Is reverse-engineered data to be used for detail work.
- Is data translated from other CAD software.
- Is aesthetically unacceptable.

Application	Modeling
Command Finder	**Refit Face**

1.3.3.12 Smooth Poles

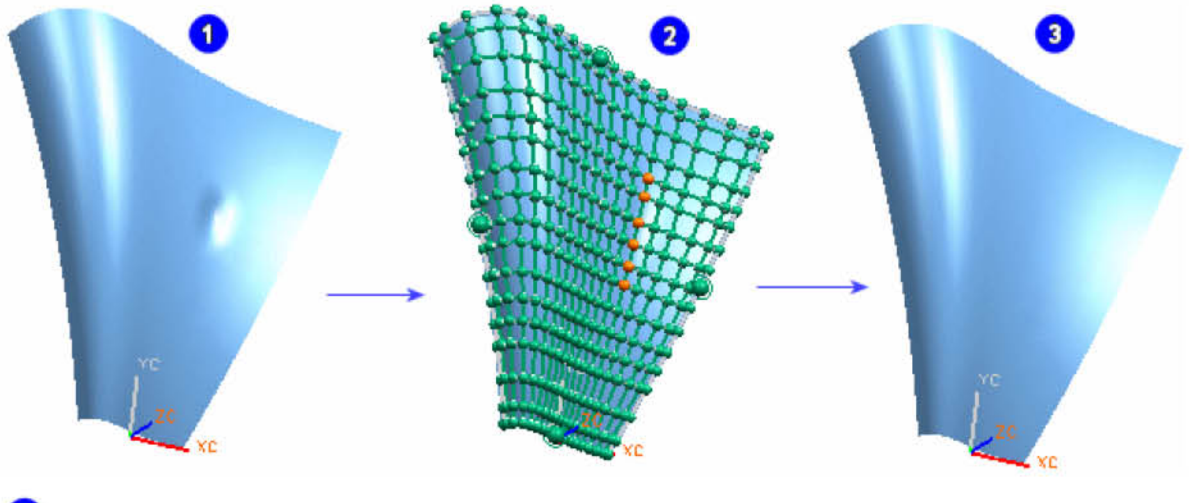
Use the **Smooth Poles** command to modify pole distribution by calculating the proper distribution of selected poles in relation to surrounding surfaces.

This command can be used to finalize Class A production surfaces. After creating a surface that is in compliance with the specification data (such as a scanned image), it is possible do some minor editing to achieve a clean pole distribution and finalize the selected surface.

Removing minor imperfections in a slab surface can help later when trying to build secondary surfaces that are both in compliance with the specification data and that achieve the proper highlight flow.

Note The **Smooth Poles** command removes parameters from associative surface features.

The following graphics show an example of surface pole smoothing.



1 Original surface

2 Selected poles to move

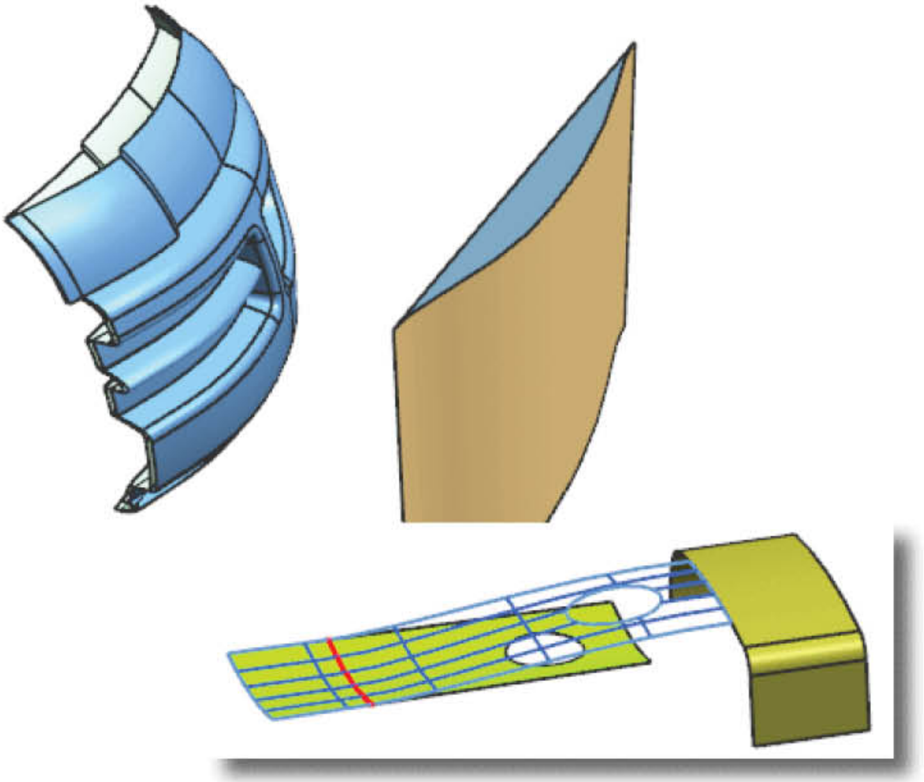
3 Resulting smoothed surface

Application	Modeling, Shape Studio
Command Finder	**Smooth Poles**

1.3.3.13 Global Shaping

Use the **Global Shaping** command to deform a surface region using a combination of geometry and values, the relationship of two curves, or the relationship of two surfaces. Combinations of geometry and values are provided by **Types** of **Global Shaping**.

Global Shaping allows deforming a surface in a predictable manner, with full associativity. It can be used to alter existing surfaces while preserving their aesthetic properties. It is also possible to modify surfaces to account for the effects of springback during metal forming.



Using a shaded preview to gauge Global Shaping results

A shaded result will only be as true as the facets of the sheet body will allow, which is why a preview is not a full proof method to validate the result of a **Global Shaping** action. If the surface is planar in most of the areas, the face resolution tolerances may occasionally have to be tweaked to increase the number of facets to get the correct shaded display.

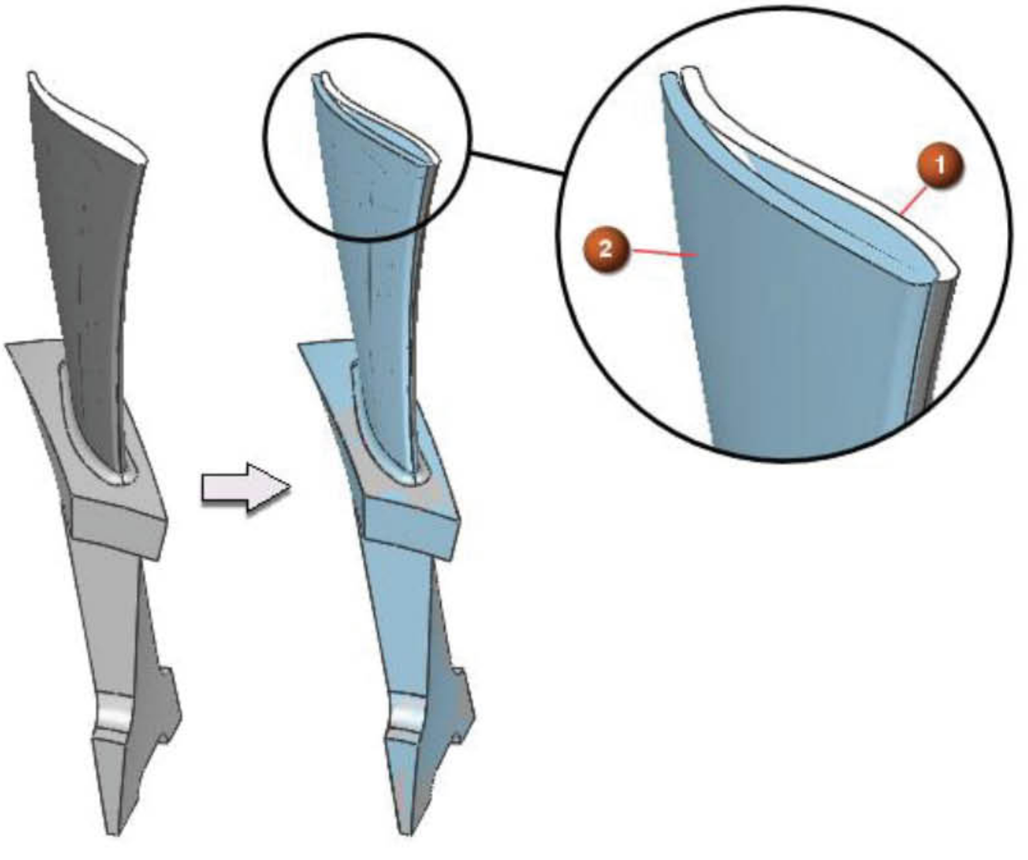
To change the **Resolution Tolerances** for **Face** in either the **Facet Settings** or **Visualization Preferences** dialog boxes. You could also use a **Section Analysis** grid to view the correct result.

Application	Modeling
Command Finder	**Global Shaping**

Global Deformation

Use the **Global Deformation** command to modify an existing product design to accommodate changes from downstream design or production process constraints, such as CAE analysis or physical tryout.

The command can also be used for maintenance, repair, and overhaul processes (MRO). For example, **Global Deformation** could be used for MRO in jet engines, to adapt an as designed engine fan blade in it's hot state to the unwound cold state when the engine is not operating. This representation of the form may be important if it is the state in which the part is manufactured or repaired.



- 1 Original shape of a jet engine fan blade
- 2 Globally deformed shape (shown in blue)

Inputs for global deformation can include:

- The formed shape of the sheet body.
- A set of points representing the non-deformed shape.
- Another set of points corresponding one-to-one to the previous set of points, and representing the deformed shape.

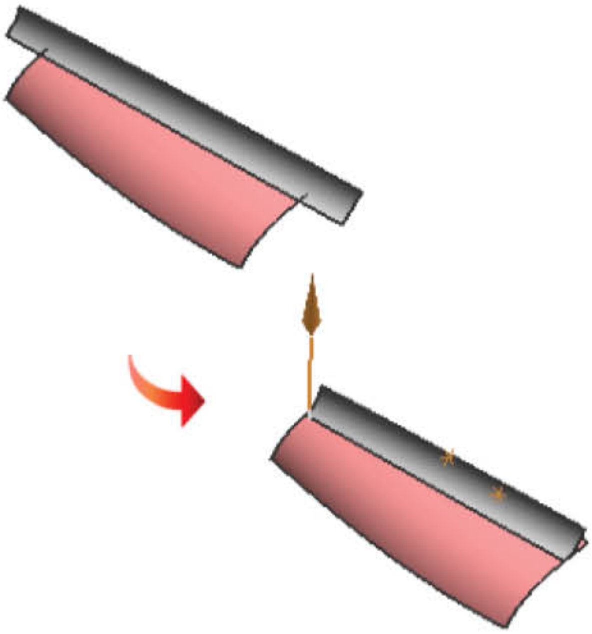
Application	Modeling
Command Finder	**Global Deformation**

1.3.3.14 Replace Edge

Use the **Replace Edge** command to modify or replace an existing boundary of a sheet body.

It possible to:

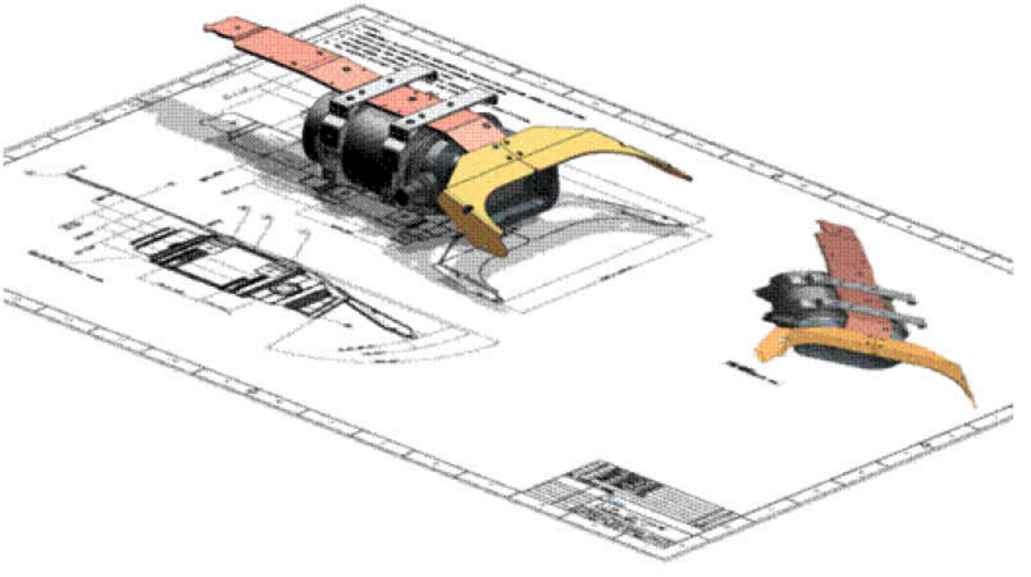
- Extend the boundaries of a sheet body if it consists of a single face.
- Edit the original or a copy of a sheet body.



The **Replace Edge** command trims a sheet body and removes the sheet body parameters. If you want to retain the sheet body parameters, use the **Trimmed Sheet** or **Trim Body** command.

Application	Modeling
Command Finder	**Replace Edge**

1.4. Export 2D drawings from 3D drawings with NX



The Drafting application is designed to produce and maintain engineering drawings which comply with major national and international drafting standards. Drawings can be created in NX using the following processes.

The model-based process	Drawings are created directly from 3D models or assemblies and are fully associated to the model. Any changes made to the model are automatically reflected in the drawing.
The stand-alone process	Drawings are created using 2D geometry and dedicated Drafting tools to create the drawing details. Additional tools are available that allows converting the 2D content into 3D models.

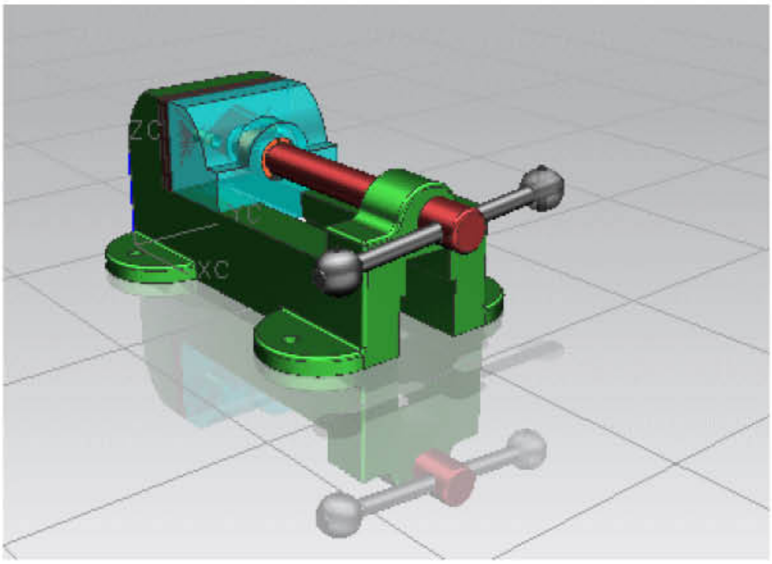
1.4.1 International Drawing Standards

An intuitive, easy to use, graphical user interface with automated tools that help to

create drawings quickly and easily. Immediate, on-screen feedback throughout the drafting process helps to reduce rework and editing.

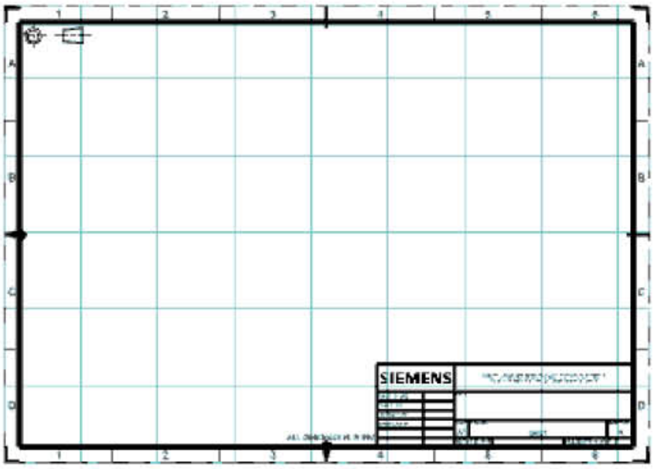
- Support for major national and international drafting standards, including ASME, ISO, DIN, JIS, GB and ESKD.
- Fully associative drafting annotations that update when the model updates.
- A comprehensive set of view creation tools that support advanced display, placement, associative, and update requirements for all view types.
- Support for 2D to 3D workflows in the stand-alone process. The 2D curve data created in the drawing can be used to derive a 3D model.
- Controls for managing drawing updates and large assembly drawings which enhance user productivity.
- NX Open Application Programming Interfaces (APIs) to the Drafting functionality that enable the development of customer and third party custom applications.
- Data migration for I-deas drawings (ASC/DWG), DXF/DWG data, and IGES data.

- The following illustrates the general process for creating a drawing from an existing 3D model This overview is not intended to give a detailed description of specific functions or operations. Details are addressed in later sections of this document.



- Set the drafting standard and drawing preferences
- Before creating a drawing, it is recommended that the drafting standard, drafting view preferences, and annotation preferences for the new drawing should be set. Once set, all views and annotations will be consistently created with appropriate visual characteristics and symbology.

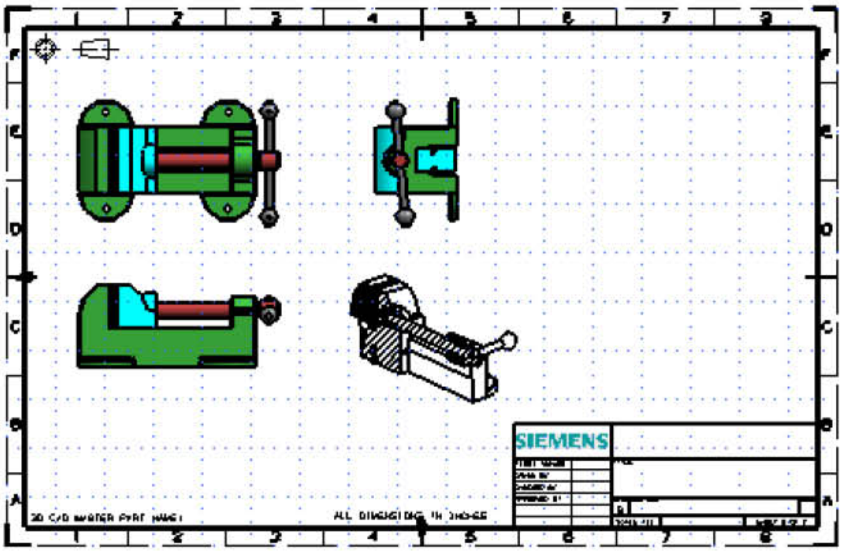
1.4.1.1Create a new drawing

- The first step in creating a drawing is to make a new drawing sheet either directly within the current work part, or by creating a non-master drawing part that contains the model geometry as a component.
- See Creating a new drawing for more information.
- 

1.4.1.1 Add views

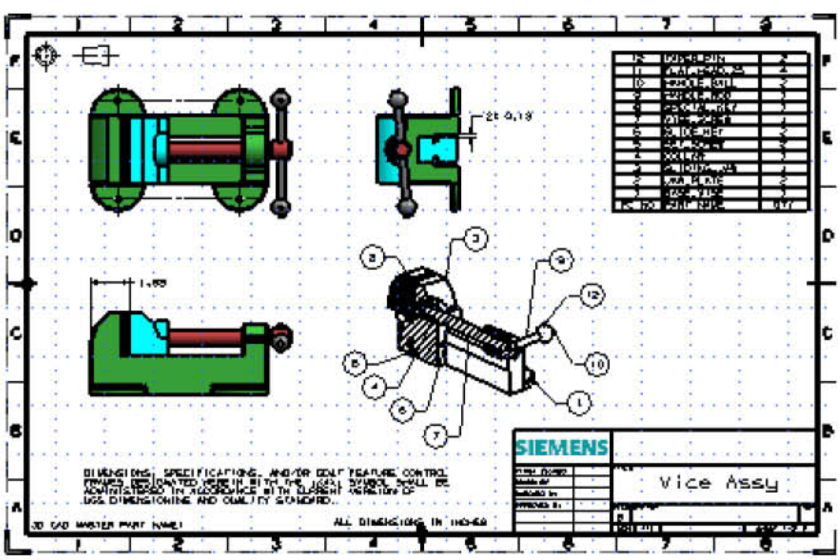
- NX enables to you create a single view or multiple views at the same time. All views are derived directly from the model, and can be used to create other views, such as section and detail views. The base view determines the orthographic space and view alignment for projected views.

- See Drafting views for more information.

- 

1.4.1.3 Add dimensions and annotations

- Once the views on the drawing have been place, dimensions and annotations are ready to be added.
- Dimensions and annotations are associated with the geometry in the views. If a view is moved the associated dimensions and annotations move with the view. If the model is edited, the dimensions and annotations update to reflect the change.
- You may also choose to add notes, labels, and in the case of assembly drawings, parts lists to the drawing

- 

- A completed drawing can be plotted directly from NX, or the part containing the drawing can be used directly by manufacturing to fabricate the part

1.4.1.4 The stand-alone drawing process in NX

The following illustrates the general process for creating stand-alone 2D drawings. NX provides support for two basic standalone approaches:

- Non-view based approach — Applying 2D geometry directly to the drawing sheet.

- Drawing-view based approach — Using **Drawing** views to manage 2D curves and the resultant 3D model.

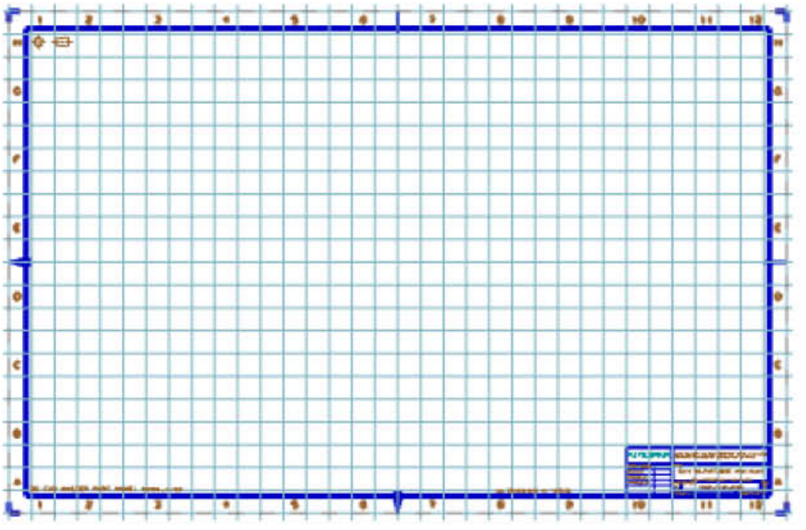
1.4.1.5 Set drafting standard and drawing preferences

Before creating a drawing, it is recommended that the drafting standard, drafting view preferences, and annotation preferences for the new drawing are set. After having set them, all views and annotations will be created consistently with appropriate visual characteristics and symbology.

See Setting your Drafting standard and Drafting Preferences for more information.

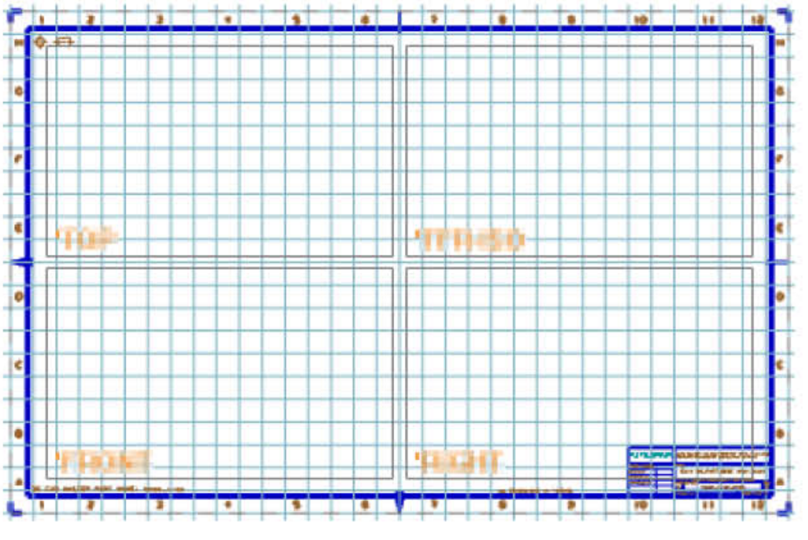
1.4.1.6 Create a new stand-alone 2D drawing

The first step in creating a stand-alone 2D drawing is to create a new part file. This drawing part file does not need to reference 3D model geometry. It is possible to use the drawing templates that do not require the reference of model geometry. See 2D: Create a stand-alone drawing for more information.



1.4.1.7 Add views

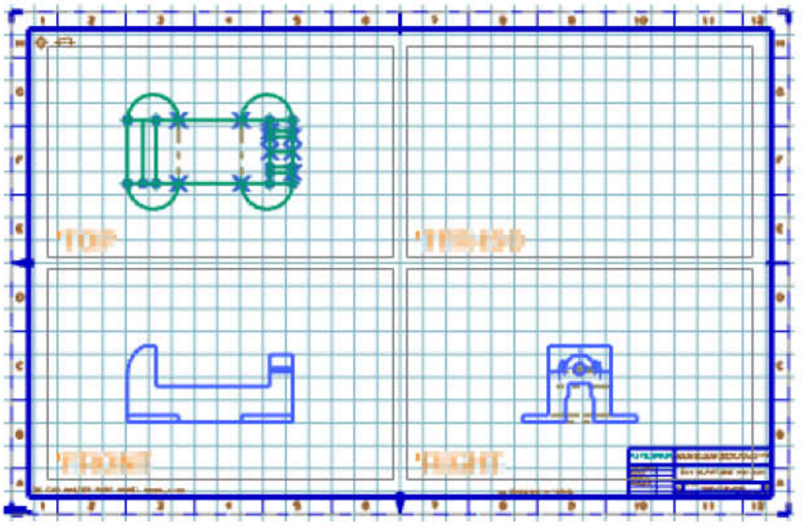
When using the drawing-view based approach, it is possible to create a single empty drawing view or multiple empty drawing views at the same time. The drawing view determines the orthographic space and view alignment for all projected views, and can be used to create detail views. See Drafting views and Drawing View for more information.



1.4.1.8 Create curves

In NX curves can be created directly on the drawing sheet or in the **Drawing** view. See Methods for creating 2D curves in Drafting for more information.

In the drawing-view based approach, the **Project to View** command simplifies 2D design by projecting curves from one view onto other views with appropriate orientations.

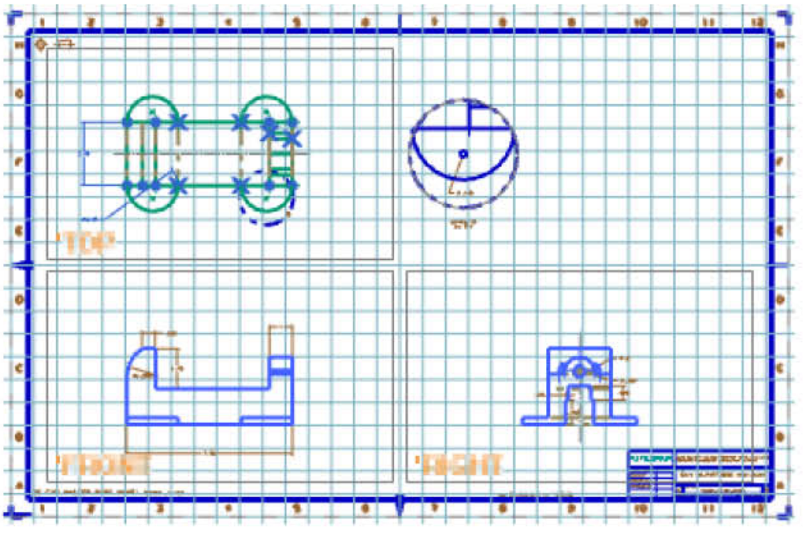


In the non-view based approach, NX enables the infinite use of lines to help to create and align 2D geometry in a drawing. See Infinite sketch lines overview for more information.

1.4.1.9 Add dimensions and annotations

After having placed the views on the drawing, dimensions and annotations are ready to be added.

Dimensions and annotations such are associated with the geometry in the views. If a view is moved the dimensions and annotations move with the view. If the model is edited, the dimensions and annotations are updated to reflect the change.

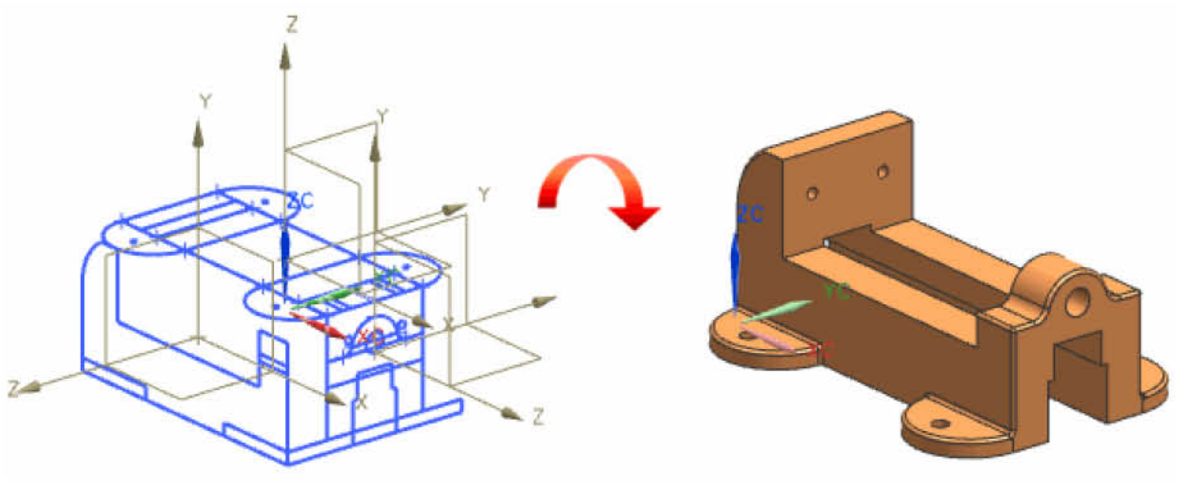


Notes and labels can also be added to the drawing.

A completed drawing can be plotted directly from NX, or the part containing the drawing can be used directly by manufacturing to fabricate the part.

1.4.1.10 Convert 2D designs to 3D models

In the drawing-view based approach, NX provides a streamlined path to convert 2D designs to 3D models without recreating the whole geometry. Use the **Copy to 3D** command to project sketch and view curves from drafting views to the 3D model space as a starting point for the construction of the model geometry.

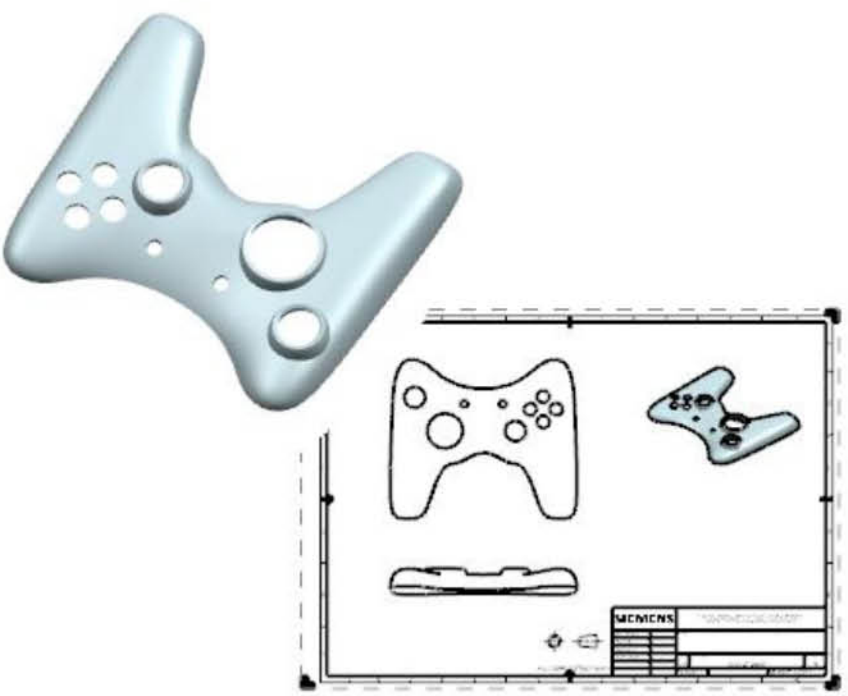


1.4.1.11 Drafting support for convergent bodies

Convergent bodies are faceted geometry that are imported to NX and optimized so it is possible to apply NX features directly to the faceted data. Because convergent bodies are modeling data from multiple sources, they can be used to accelerate the concept-to-production workflows.

Convergent bodies are approximations of a true analytic surface, so support for convergent bodies in drawings is limited to documenting key characteristics of the part, such as overall dimensions, annotations that point out key design features, and important design decision and manufacturing notes.

For more information about the use of convergent bodies in modeling, see What is Convergent Modeling? in the Modeling help.



1.4.1.12 Limitations to using convergent bodies in a drawing

- **Exact (Pre-NX8.5)** drafting views and legacy lightweight views are not supported. If convergent bodies are detected in the master model, a warning is presented indicating that the bodies will not be displayed.

Note In pre-NX 12 drafting views, you must first version up and then update the views for convergent bodies to be visible in the view.

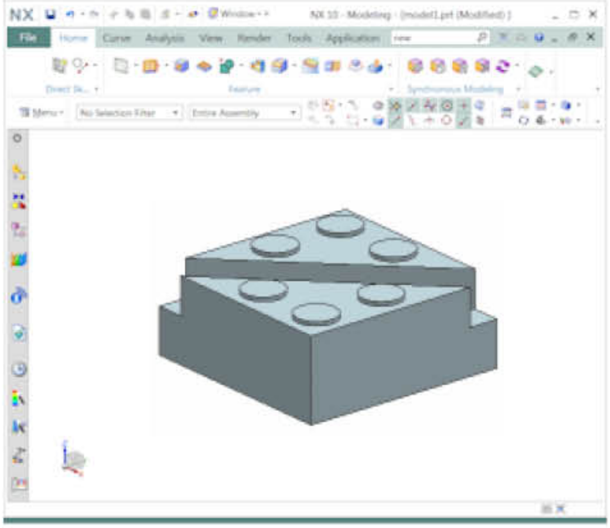
- The following **Hidden Lines** preferences and settings are not supported for convergent bodies in a view:
  - **Process Interfering Solids**
  - **Display Interference Curves**
  - Any **Display Mode** setting for **Small Features**
- It is only possible to dimension convergent bodies using the **Linear Dimension** and the **Rapid Dimension** commands. Only **Horizontal**, **Vertical**, and **Point to Point** linear dimensions and dimension sets are supported.
- When applying annotations, dimensions, and centerlines on convergent bodies, only specific snap point options can be used:
  - For annotations, only **End Point**, **Control Point** (limited to end points only), and **Point on Curve** can be used.
  - For dimensions, only **End Point** and **Control Point** (limited to end points only) can be used.
  - For centerlines, only **Control Point** (limited to end points only) can be used.

    Note It is possible to create all types of centerlines, except 3D centerlines and automatic centerlines.

- For **Smart Lightweight** drafting views, hidden line rendering may display lines for geometry that should be hidden.
- The following limits apply for **Smart Lightweight** section views created from convergent bodies:
  - Automatic centerlines are not applied to convergent edges created in the section view.
  - Automatic anchor points are not created for section views. Neither can anchor points be manually added to the section view. If converting a section view from **Exact** to **Smart Lightweight**, or vise versa, the view position on the drawing may change.
  - It is impossible to apply dimensions to section curves created from convergent bodies in the section view.
- You cannot apply **Flag**, **Datum**, or **Dot Terminated** leader types because convergent bodies do not contain true linear edges.
- When using convergent bodies in the definition of a custom symbol, NX extracts edge geometry from the body and saves it in the custom symbol.
- It is impossible to create crosshatch, area fill, or solid fill patterns on convergent bodies.
- It is impossible to create hole tables from convergent bodies.
- Hybrid bodies (convergent bodies to which NX features are added) are not supported in Drafting.

1.4.1.13 3D Create a new drawing of a master model

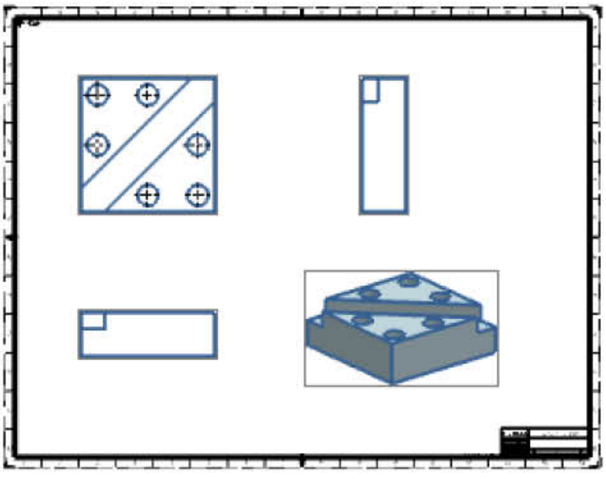
This example shows how to create a non-master drawing from a master part.



- Choose **File** tab→**Open**, and select the master model part or assembly.
- *For native NX only:* Choose **File** tab→**New**.
- *For Teamcenter Integration only:* Choose **File** tab→**New**→**Item**.
- Click the **Drawing** tab.
- *For native NX only:* In the **Templates** group, from the **Relationship** list, make sure **Reference Existing Part** is selected.
- From the **Templates** list, select an appropriate drawing template.

• Tip	• Although not mandatory, typically a drawing will be in the same units as the model. By default, the units will be set to the same units as the model.

- (Optional) In the **Name** box, accept the default name or type a name.
- (Optional) Choose a new folder location by clicking **Browse**  next to the **Folder** box.
- Click **OK** to create the new drawing part.
- (Optional) Use the **Populate Title Block** dialog box to automatically add content to individual unlocked title block cells in the drawing.
- (Optional) Use the **View Creation Wizard** or the **Base** command to add views to the drawing.
- In this example, an E sized template was created and added views.



Note When creating a drawing this way, by default the template will set all of the drafting preferences to those contained in the template. If there is a desire to use a Drafting Standard to set the Drafting preferences, it is required either to select the **Drafting Standard** after the template part is created, or to set the **Use Settings from Standard** customer default on the **Drawing→General** tab of the **File** tab→**Utilities→Customer Defaults→Drafting** dialog box.

- Click the **Assembly Navigator**  tab in the Resource bar.

Note that the working section is now in an assembly file and the original part file has been added as a component.

- Sections
  - block_dwg1
    - block

Although this is an assembly of the original part, all Drafting commands and operations perform in the same way as they do in the original part.

Note See the *Assemblies* Help for additional information about working with assemblies.

1.4.2 Editing information on the drawing frame

1.4.1.1 Add a custom drawing sheet

- In the Drafting application, use one of the following methods to open the **Sheet** dialog box.
  - Choose **Home** tab→**New Sheet** .
  - Choose **Menu→Insert→Sheet**.
  - In the **Part Navigator**, right-click the **Drawing** node and choose **Insert Sheet**.
- In the **Sheet** dialog box, click **Custom Size**.
- Set the **Height** to 8.5 and the **Length** to 14.
- From the **Scale** list, select **Custom Scale**.
- Set the scale to 1:3.
- (Optional) Expand the **Name** group, and type a new:
  - **Drawing Sheet Name**
  - **Sheet Number**

- **Revision**

- (Optional) Expand the **Settings** group and change the drawing units and projection angle.
- Click **OK** to create the custom drawing sheet.
- The drawing sheet displays with a dashed border.

- (Optional) Use the **Borders and Zones** command to add custom borders and zones to the drawing sheet.

1.4.1.2 Creating a new drawing

Drawings can be created using either a stand-alone workflow method or a model-based workflow method.

Stand-alone drawing workflow

The stand-alone drawing workflow is used to place the drawing data in a single part file. This is recommended for a 2D drafting process where only 2D geometry is used to create the drawing. The 2D curves can be placed directly on the drawing sheet, or can be placed in drawing views, and used to generate 3D model geometry.

Model-based drawing workflow

The model-based drawing workflow uses the existing 3D geometry to generate the 2D drafting data. In this workflow, it is possible to:

- Place a drawing directly in the file that contains the 3D model geometry. The drawing data is associated to the 3D geometry and will update when the model geometry updates.
- Use the master model architecture, in which the drawing data is placed in a file which is separate from the file that contains the model geometry. This workflow is recommended for a 3D drafting process. The drawing data is directly associated to the 3D model geometry and updates when the geometry updates, but different users can work concurrently with the same model data.

Command Finder	**New** 
Location in dialog box	**Drawing** tab→**Relationship** list→**Stand-alone Part**→select a template from the list

Model-based drawing workflow – drawing contained in the master part

Application	Drafting
Prerequisite	Choose **File** tab→**Start**→**Drafting** or press Ctrl+Shift+D to start the Drafting application.

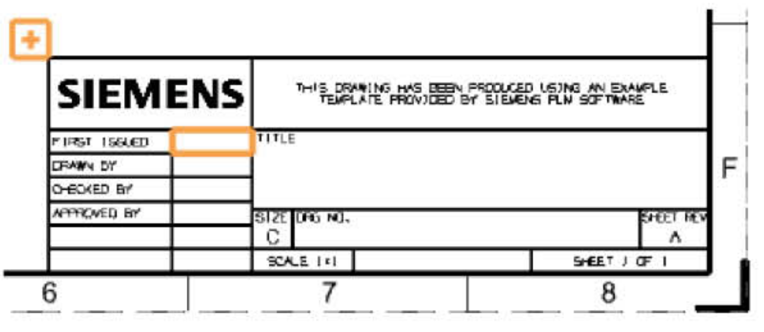
Command Finder	**New Sheet** Note: If the **Always Start Insert Sheet Command** is selected in the **General/Setup** node→**Workflow** node→**Model-Based** group of the **Drawing Preferences** dialog box, then a sheet will automatically be prompted to be added when entering the Drafting application for the first time.

Model-based drawing workflow – drawing contained in a separate, non-master part

Command Finder	**New**
Location in dialog box	**Drawing** tab→**Relationship** list→**Reference Existing Part**→select a template from the list
Prerequisite	It is a must to provide the name of the 3D model to be referenced before the drawing part is created.

1.4.1.3 Populate Title Block

Use the **Populate Title Block** command and the **Populate** command to edit the contents of unlocked title block cells. The **Populate Title Block** command allows globally editing all unlocked cells in all the title blocks in the drawing. The **Populate** command allows editing only the unlocked cells of the currently highlighted title block.



Tip To automatically edit the contents of a title block when creating a drawing from a template, **Display Populate Title Block Dialog on Template Instantiation** check box must be selected in the **Customer Defaults** dialog box.

The **Populate Title Block** command is not available from the shortcut menu when creating a new drawing template or editing an existing drawing template.

Note A drawing template is created by using the **Mark as Template** command.

To use the **Populate Title Block** command to edit the contents of a title block in a drawing template, it is a must to specifically choose **Drafting Tools** tab→**Drawing Format**→**Populate Title Block** Or it is possible to double-click the title block to open the **Edit Definition** dialog box, and then click the **Edit Table** button to edit the contents of the title block.

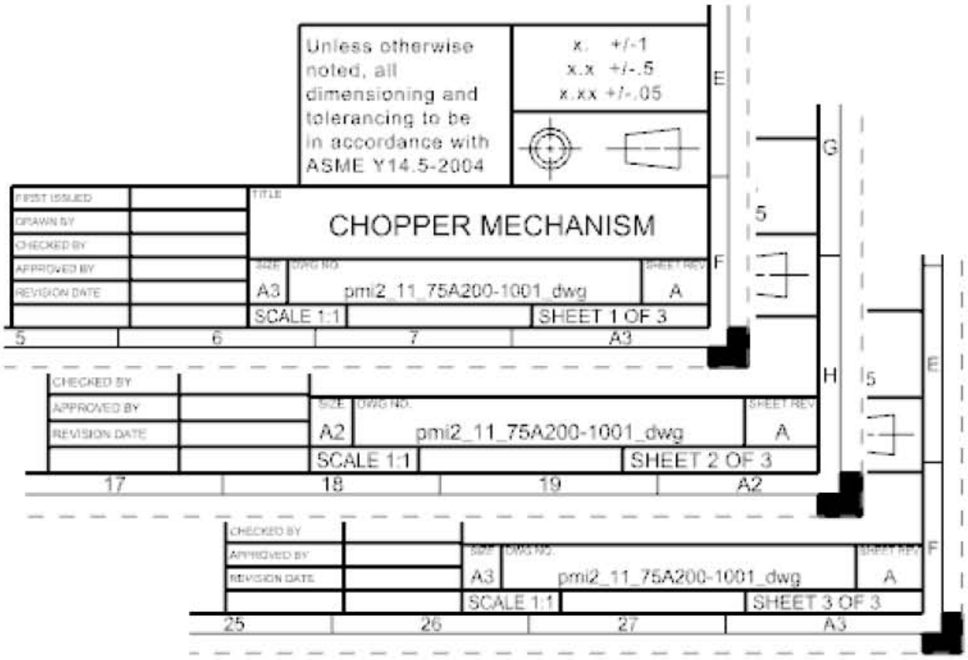
Application	Drafting
Command Finder	**Populate Title Block**

Edit the unlocked cells of a specific title block

Application	Drafting
Graphics window	Right-click the title block→**Populate**

Edit all title blocks in the drawing

Use this procedure to make changes to all of the title blocks in your drawing at the same time.



1. Choose **Drafting Tools** tab→**Drawing Format**→**Populate Title Block** .

   In the **Populate Title Block** dialog box, all *unlocked* cells from all title blocks in the drawing appear in the list.

2. Select a row from the list.

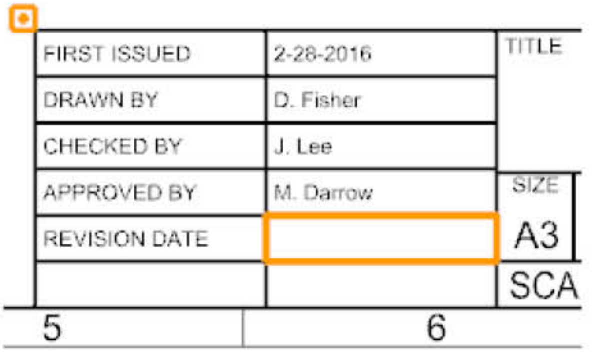
   In this example, the **First Issued** row is selected.

3. In the **First Issued** text box at the top of the dialog box, type **2-28-2016**.

   Note

   For additional characters and symbols, click and use the **Text** dialog box to edit the cell's value.

4. Press Enter to add the value to the cell, and then repeat the step to add information to other empty cells.



5. Click **Close** when finishing adding content to the empty cells, and then review all title blocks in the drawing to validate the changes.

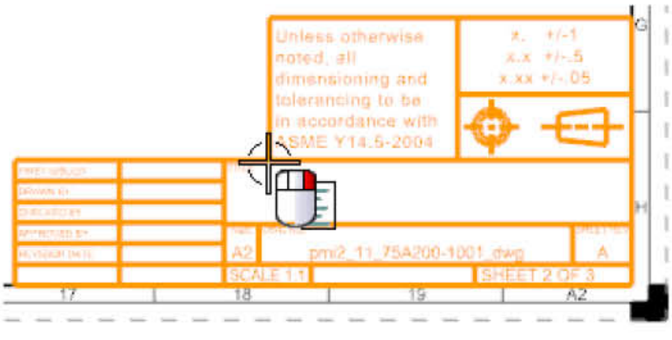
   Note If the same title block is used on other sheets in the drawing, the edits will also appear in those title blocks.

1.4.1.4 Edit a specific title block

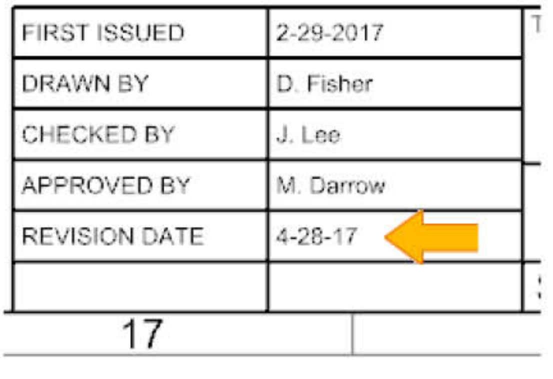
Use this procedure to make changes to a specific title block in the drawing. In this example, a revision date is added to only the title block on the second sheet of a drawing.

1. In the **Part Navigator**, double-click the **"Sheet 2"** node to display the drawing sheet.
2. Right-click the title block and select **Populate**.

   Tip If the title block does not highlight, make sure the layer the title block resides on is selectable.

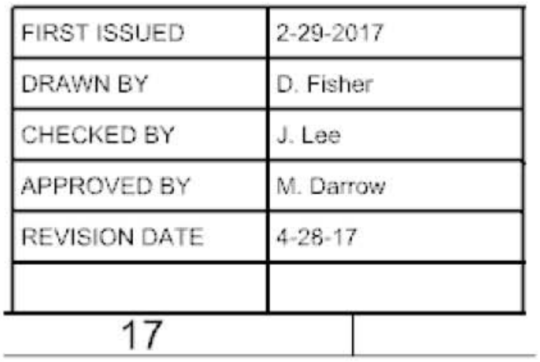
3. 

4. In the **Populate Title Block** dialog box, select the **Revision Date** row from the list.
5. In the **Revision Date** box at the top of the dialog box, type **4-28-2017**, and then hit Enter.



6. Close the **Populate Title Block** dialog box, and then verify that the other title blocks in the drawing do not include the revision date.

FIRST ISSUED	2-29-2017
DRAWN BY	D. Fisher
CHECKED BY	J. Lee
APPROVED BY	M. Darrow
REVISION DATE	
5	6

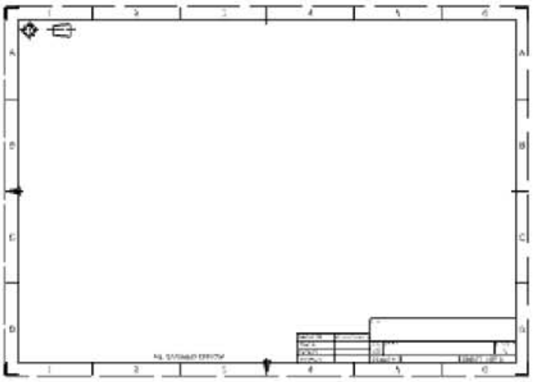


FIRST ISSUED	2-29-2017
DRAWN BY	D. Fisher
CHECKED BY	J. Lee
APPROVED BY	M. Darrow
REVISION DATE	4-26-17
17	

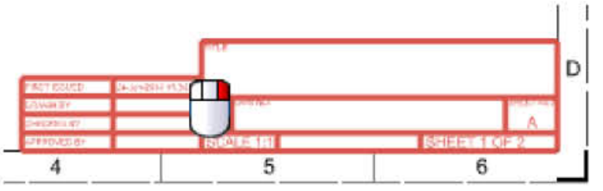
FIRST ISSUED	2-29-2017
DRAWN BY	D. Fisher
CHECKED BY	J. Lee
APPROVED BY	M. Darrow
REVISION DATE	
25	26

1.4.1.5 Reuse a title block

Open the drawing that contains the title block that wanted to reuse.

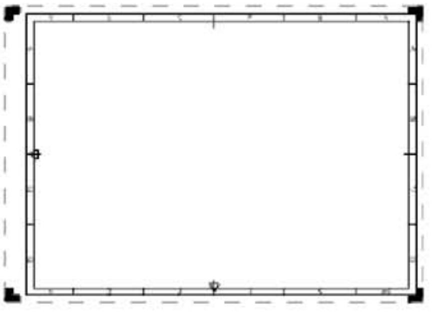


Right-click the title block and choose **Copy**.

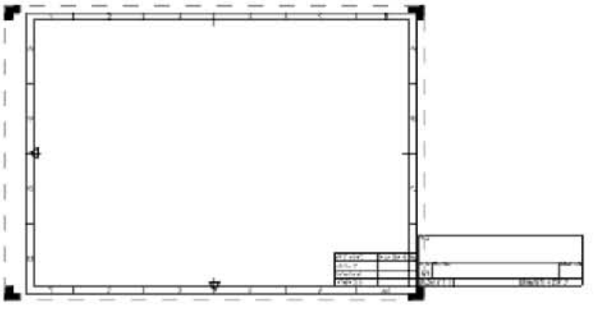


Do one of the following:

- Open another drawing sheet in the current part.
- Open another drawing part and display a drawing sheet.

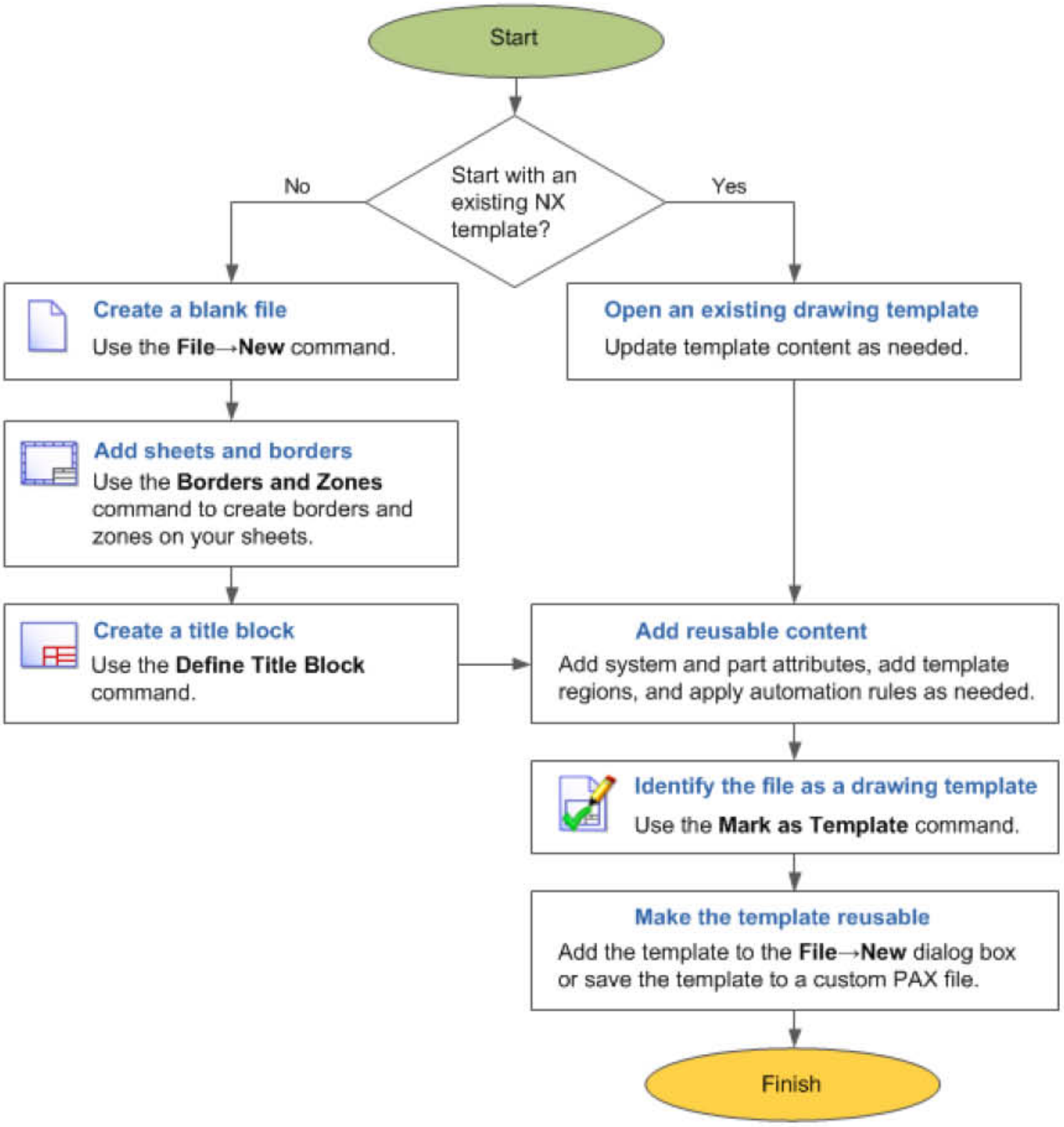


In the graphics window, right-click and select **Paste**.



Reposition the title block if necessary.

1.4.1.6 Workflow for drawing templates



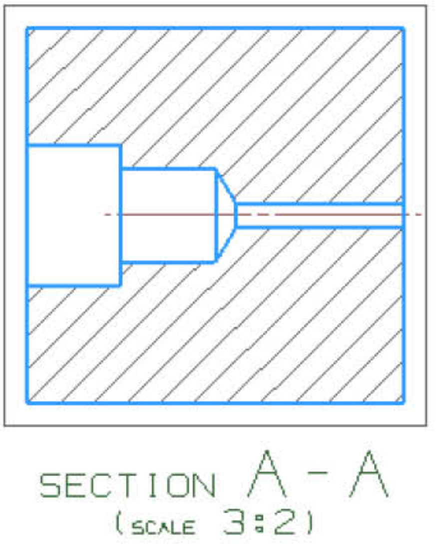
1.4.2 View Labels

It is possible to automatically generate associative labels for any drafting view. When the view is either moved or deleted, the labels are likewise moved or deleted.

A view label consists of a prefix and a label letter. A scale label consists of a prefix and scale value. The scale value is associated with the actual scale of the view and updates when the view scale changes.

Setting view label options helps to:

- Control the visibility of the view label and scale label
- Set the view label's prefix name, letter, letter format, and letter size.
- Set the view scale label's text position, prefix name, prefix text size, letter format, and letter size.



Section View with view label and view scale label

NX uses the same dialog box to both set view label preferences and edit existing view labels. Use the **Drafting Preferences** dialog box to set preferences for new view labels, or use the **Settings** dialog box to edit the settings of existing view labels. Different options are available in the dialog box depending on whether setting preferences or editing an existing view label are on progress.

Application	Drafting
Command Finder	**Drafting Preferences**
Location in dialog box	Set the format of view labels: **View** node→**Common** node→**View Label** node Display and set view label properties for new views: **View** node→**Base/Drawing** node→**Label** node **View** node→**Projected** node→**Label** node **View** node→**Section** node→**Label** node **View** node→**Detail** node→**Label** node

Display and set view label properties for an existing view

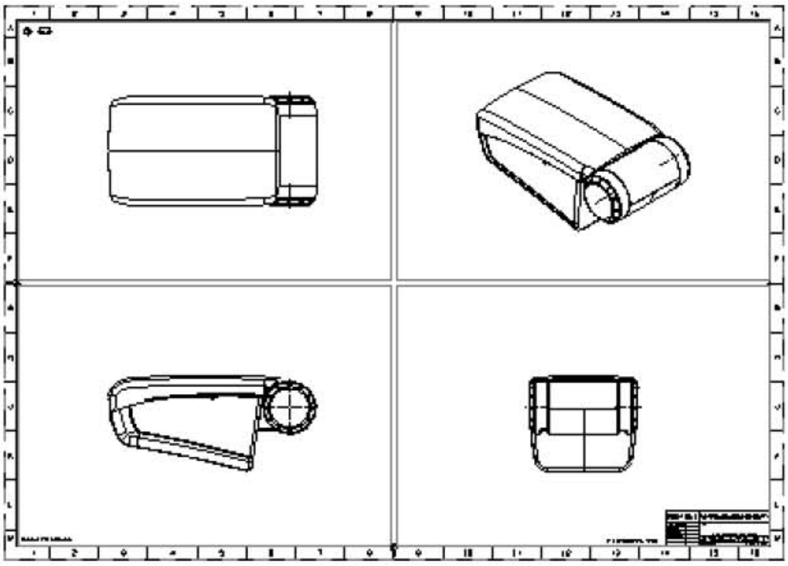
Application	Drafting
Graphics window	Double-click a view boundary Right-click a view node in the **Part Navigator**→**Settings** Double-click an existing view label
Location in dialog box	Set the format of the view label

	Common node→**View Label** node Display and set view label properties for view **Base/Drawing** node→**Label** node **Projected** node→**Label** node **Section** node→**Label** node **Detail** node→**Label** node

1.4.3 Select the projection method for the object

1.4.3.1 View Creation Wizard

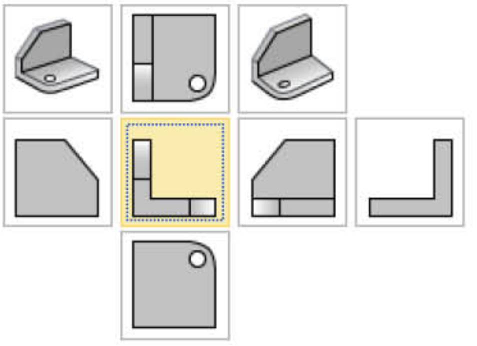
The **View Creation Wizard** streamlines the process of adding one or more drafting views to a drawing sheet.



The wizard guides the process of:

- Selecting a currently loaded, recently loaded, or unloaded part or assembly to be included in the views.
- Specifying the preview style and view settings.
- Optimizing view settings for large assembly views.
- Inheriting PMI data from the model.
- Selecting assembly arrangements.
- Specifying a custom view orientation for the parent view.
- Generating a multi-view layout.
- Placing the view layout on the drawing sheet.

From the parent view, it is possible to select up to five standard orthographic view projections. The projection angle of the views is derived from the orientation of the parent view and the drawing's projection angle.



It is also feasible to include an isometric or trimetric view in the layout.

Application	Drafting
Command Finder	**View Creation Wizard**

1.4.3.2 Base views

Use the **Base** command to add any standard modeling or custom view saved in a part to a drawing sheet. A single drawing sheet may contain one or more base views. From base views, you can create associated child views such as **Projected**, **Section**, and **Detail** views.

The **Base** command provides options that help to:

- Add any view from the master model part, the current part, or another loaded part.
- Specify the position and orientation of a view on the drawing.
- Define the view scale and settings.
- Control the appearance of components in views on assembly drawings.

Note Exploded views can only be added as base views if the exploded view resides in the part that contains the drawing. See Exploded Views in Drafting for additional information.

Application	Drafting
Command Finder	**Base View**
Shortcut menu	Right-click a drawing sheet border→**Add Base View**
Part Navigator	Right-click a sheet node→**Add Base View**

1.4.3.3 Projected views

You can project views from an existing base, drawing, orthographic, or auxiliary view. NX automatically infers orthographic and auxiliary alignment as the cursor is moved in a circular motion about the parent view's center.

NX automatically infers:

- A hinge line.
- A vector direction perpendicular to the hinge line. The vector arrow indicates the projection direction from the parent view.

It is possible to dynamically orient the hinge line and also reverse the projection direction before placing the view.

Application	Drafting
Command Finder	**Projected View**
Shortcut menu	Right-click a view border→**Add Projected View**
Part Navigator	Right-click a view node→**Add Projected View**

1.4.3.4 Create an orthographic view

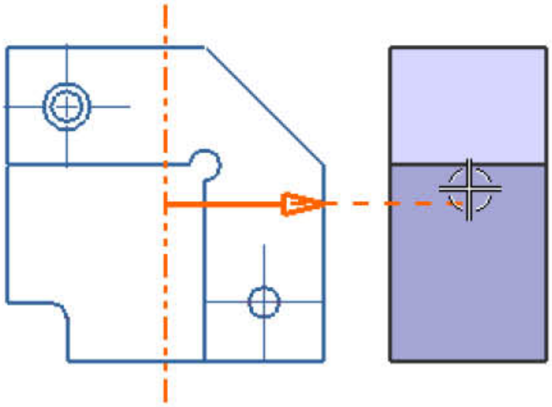
When the **Always Start Projected View Command** option is selected in the **Settings** dialog box, NX places in projected view mode immediately after placing a base view on the drawing sheet. The following example shows how to create an orthographic view from an existing view.

Right-click an existing view's border and choose **Add Projected View**.

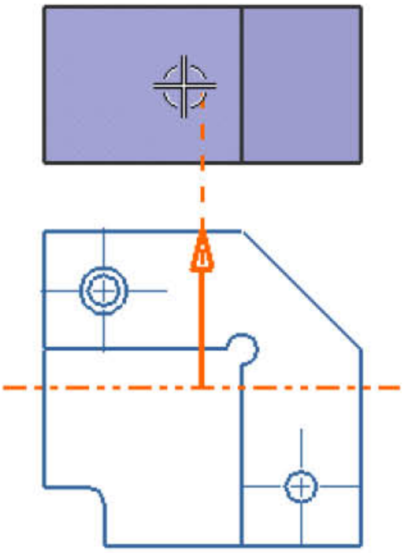
The hinge line and arrow vector symbol appear in the view.

(Optional) Choose any of the Projected view options or right-click and choose shortcut options as desired.

Move the cursor to the right or left of the parent view to let NX snap the view into position.

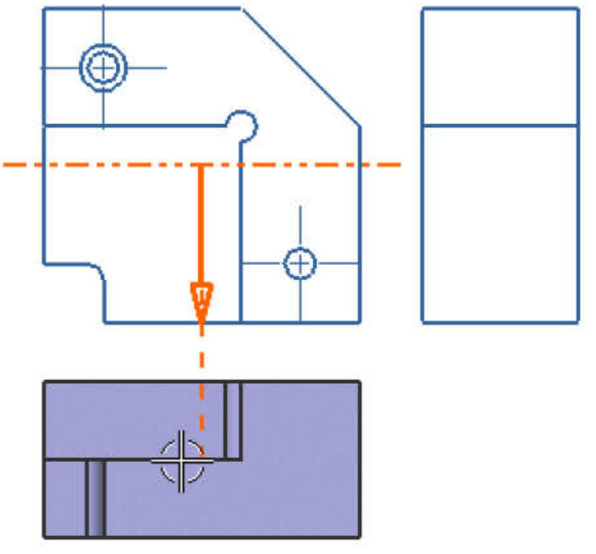


Optional: move the cursor above or below the parent view:

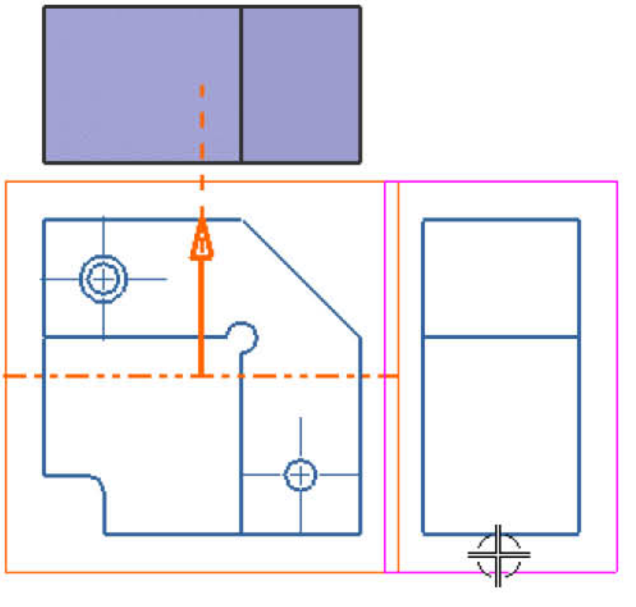


Click to place the view.

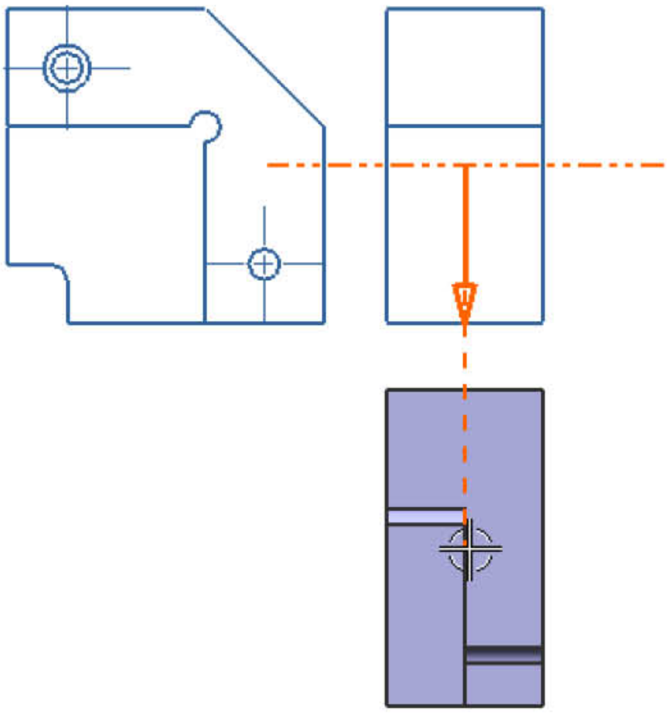
It is feasible to continue to project views from the same parent view by positioning the cursor as needed.



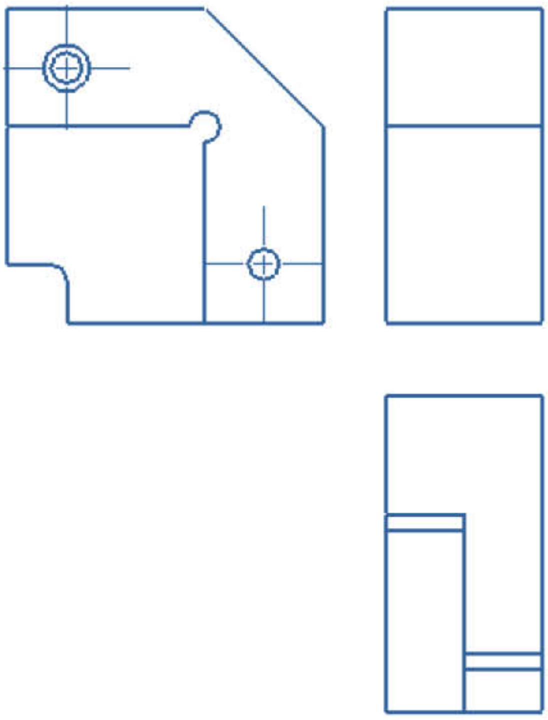
If necessary to project from the previously placed view or from an entirely different view, in the **Parent View** group, click **Select Parent** and then select a different view.



Now the hinge line and arrow vector appear in the new view selected.

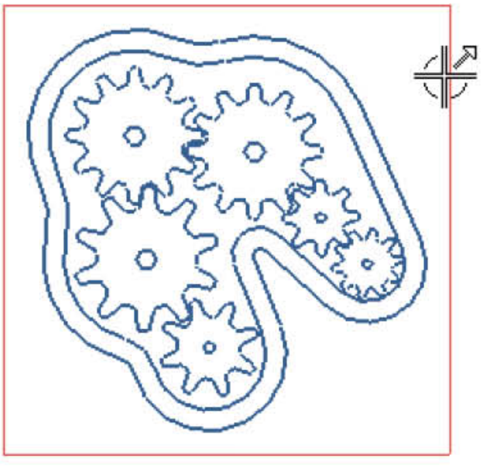


Click to place the view.



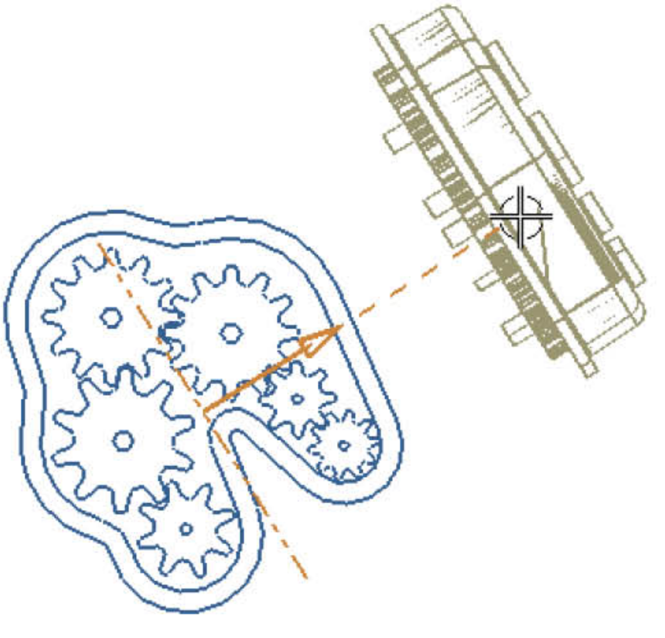
1.4.3.5 Create an automatic auxiliary view
To create an auxiliary view:

Right–click an existing view's border and choose **Add Projected View**.

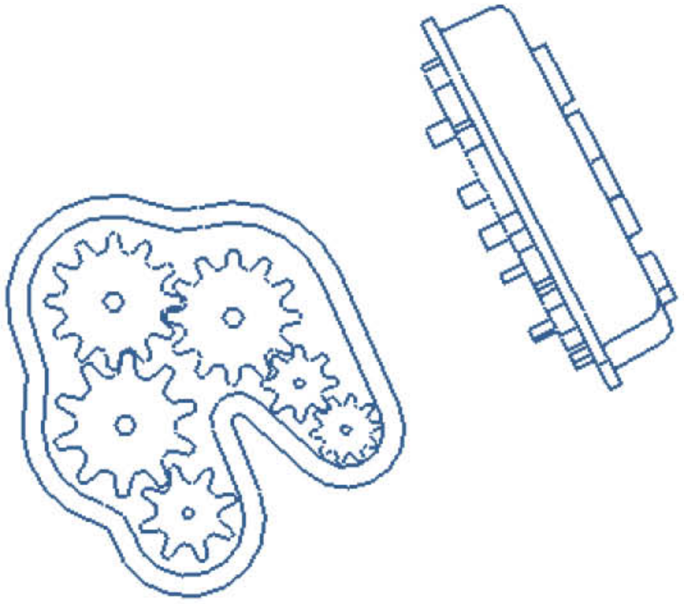


In the **Hinge Line** group, from the **Vector Option** list, make sure **Inferred** is selected.

Rotate the hinge line to an oblique angle about the center of the parent view.



Click to place the view on the drawing sheet.



1.4.3.6 Create an associative auxiliary view

Inferred

You can use the **Inferred** method to automatically specify an associative hinge line for auxiliary views. When rotating the hinge line so that it becomes parallel to a planar face on the part, that face becomes the defining feature for the hinge line.

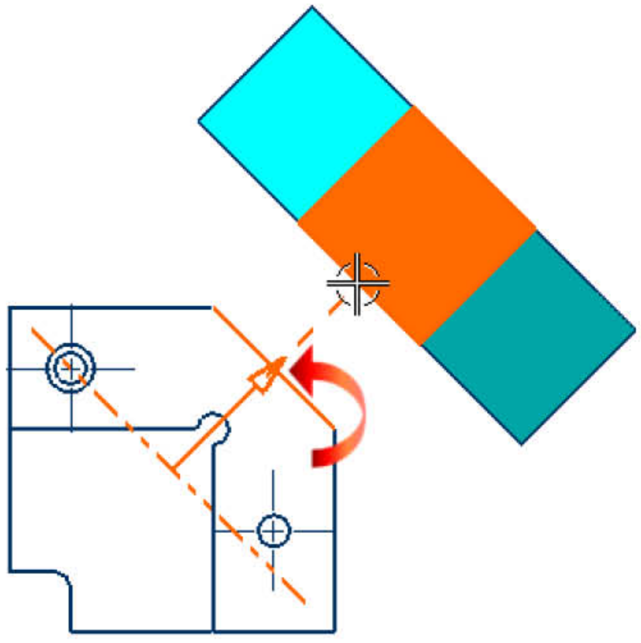
Right-click an existing view's border and choose **Add Projected View**.

In the **Hinge Line** group, from the **Vector Option** list, select **Inferred**.

Select the **Associative** check box.

Rotate the hinge line about the center of the parent view.

When the hinge line becomes parallel to a planar face on the part, the face highlights.



Click to place the view on the drawing sheet.

The face becomes the defining geometry for the hinge line. Were the angle of this face to change, the hinge line and auxiliary view would update accordingly.

Defined

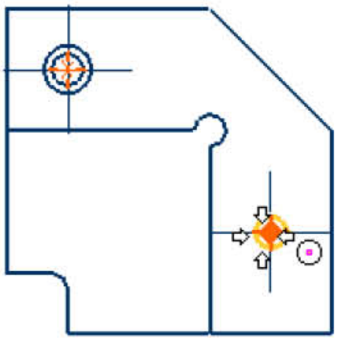
Use the **Defined** method to manually select planar faces, linear edges or other types of model geometry to define an associative hinge line for the auxiliary view.

Right-click an existing view's border and choose **Add Projected View**.

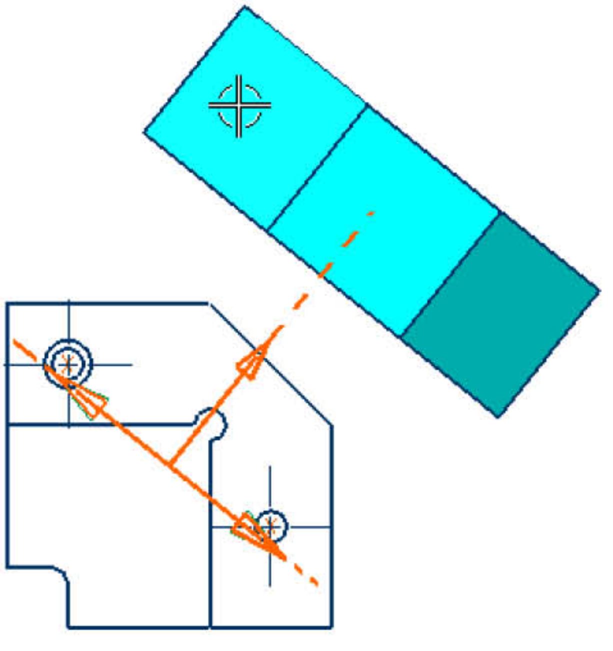
In the **Hinge Line** group, from the **Vector Option** list, select **Defined**.

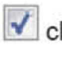
From the **Vector Constructor** list, select **Two Points** .

Select the two holes in the part.



From the **View Origin** group, under **Placement**, select **Perpendicular to Line** from the **Method** list.



If necessary, select the **Reverse Projected Direction**  check box.

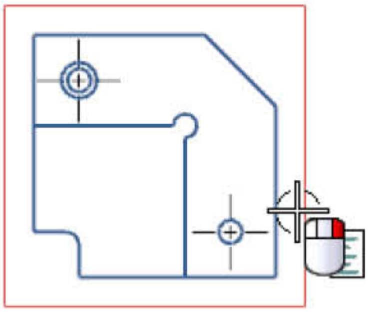
Click to place the view on the drawing sheet.

The hinge line becomes associated with the two hole centers. Were either hole to move in the part, the hinge line and auxiliary view would update accordingly.

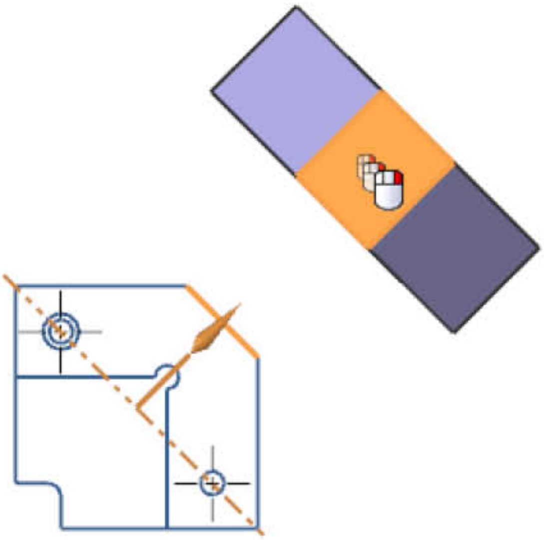
1.4.3.7 Display projected views with viewing direction reference arrows

By default, viewing direction reference arrows are only displayed with parent views for non-orthographic projections, such as auxiliary views. To create an auxiliary view with a viewing direction reference arrow:

Right-click the border of the parent view and choose **Add Projected View**.

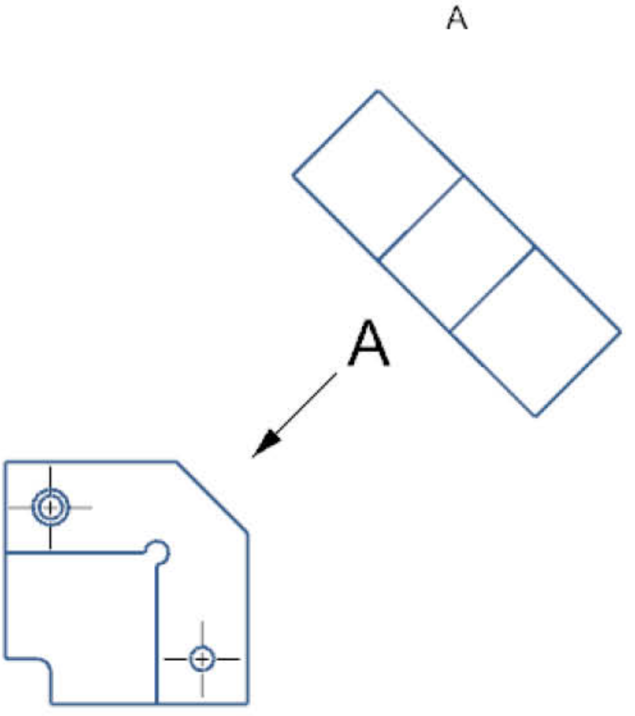


Drag the projected view to the desired alignment position with its parent view.



Click to place the view on the drawing sheet.

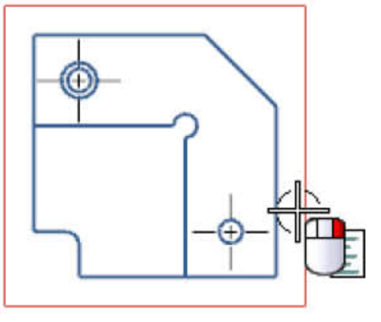
Click **Close**.



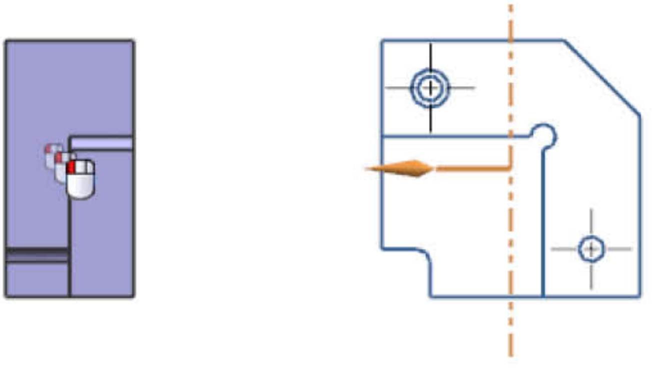
The viewing direction reference arrow and corresponding label automatically appear on the drawing sheet.

It is also possible to apply viewing direction reference arrows to any projected view as desired by using the shortcut menu command **Display Viewing Direction Arrow**. To create an orthogonal view with a viewing direction reference arrow and label:

Right-click the border of the parent view and choose **Add Projected View**.



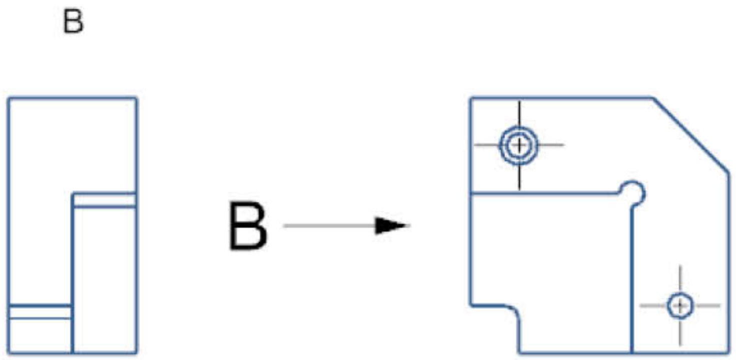
Drag the projected view to the desired orthographic position with its parent view.



Right-click and choose **Display Viewing Direction Arrow**

Click to place the view on the drawing sheet.

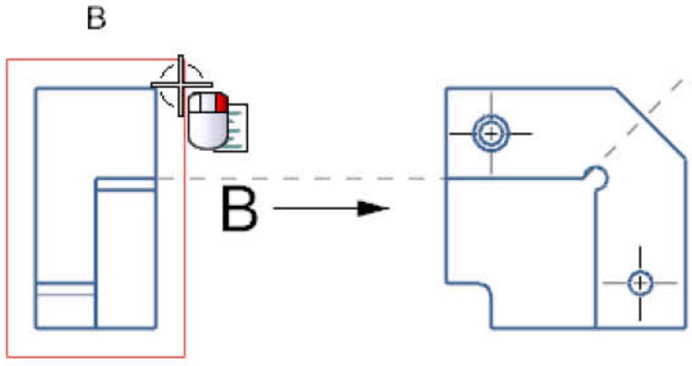
Click **Close**.



The direction arrow and label appear in the views.

To change the viewing direction reference arrow letter:

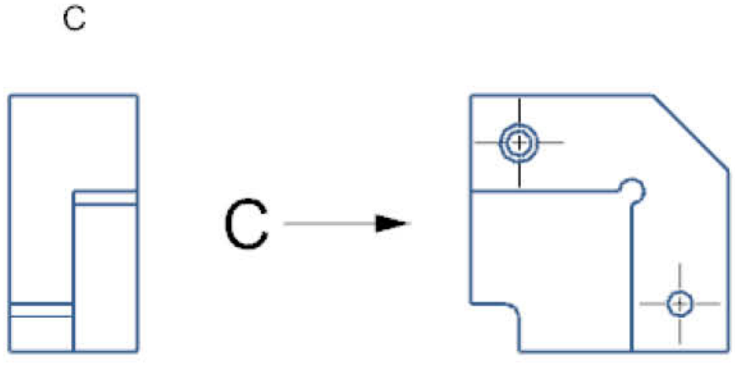
Right-click the border of the projected view and choose **Settings**.



Under the **Common** node, select the **View Label** node.

In the **Letter** box, type a valid character.

Click **OK**.



To remove the arrow letter and view label:

Right-click the border of the projected view and choose **Settings**.

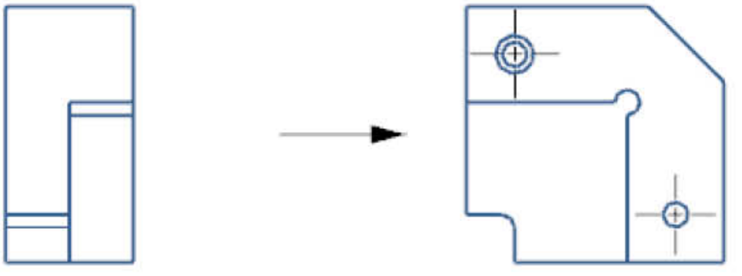
Under the **Projected** node, select the **Label** node.

Clear the **Show View Label** ☐ check box.

Under the **Projected** node, select the **Arrow** node.

Clear the **Display with Reference Arrow** check box.

Click **OK**.



To edit the viewing direction reference arrow style:

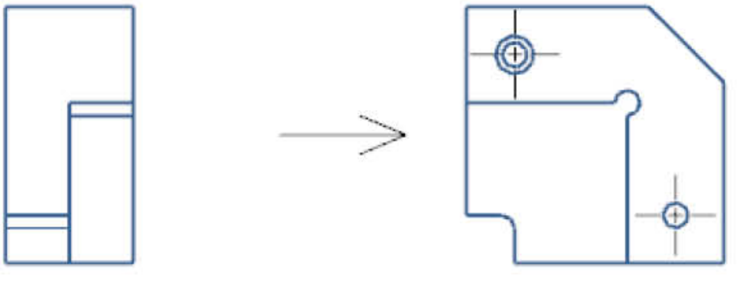
Right-click the border of the projected view and choose **Settings**.

Under the **Projected** node, select the **Arrow** node.

In the **Arrowhead** group, set the following:

Style	=	**Open**
Length	=	**10**
Angle	=	**45**

Click **OK**.



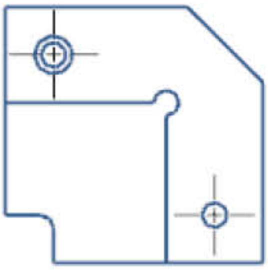
To omit the arrow altogether:

Right-click the border of the projected view and choose **Settings**.

Under the **Projected** node, select the **Settings** node.

From the **Display Arrow on Parent View** list, select **No**.

Click **OK**.



The reference arrow is removed.

1.4.3.8 Locate a projected view and its parent view

You can locate a parent view from a projected view in the **Part Navigator**. You can also locate a projected view from its parent view, and vice versa, in the graphics window.

Locate a project view's parent view in the Part Navigator

In the **Part Navigator**, expand the **Sheet** node that contains the projected view.

Right-click the **Projected** node and choose **Navigate to Parent**.

The parent view is maximized in the graphics window.

1.4.3.9 Locate a projected view or its parent view in the graphics window

To locate a projected view's parent view:

On the drawing sheet, right-click the boundary of the projected view and choose **Navigate to Parent**.

To locate a projected view from its parent view:

On the drawing sheet, right-click the viewing direction reference arrow and choose **Navigate to Projected View**.

For more information on viewing direction arrows, see Display projected views with viewing direction reference arrows.

1.4.3.10 Drawing View

Use the **Drawing View** command to create a drawing view, which is an empty drafting view that does not reference any model geometry. Although a drawing view is not associated to model geometry, it can be identified as a standard view orientation (**TOP**, **LEFT**, and so on) or defined its orientation relative to the model's absolute CSYS.

A drawing can be placed view using one of two options:

Center

Allows creating and adjusting the position of the drawing view on the sheet by specifying a center point. The resulting boundary is an **Automatic Rectangle**.

Corners

Allows controlling the size of the drawing view and its position on the sheet by specifying two diagonal points. The resulting view boundary is a **Manual Rectangle**.

To create view-dependent geometry in the drawing view:

- Make the drawing view the **Active Sketch View** and create sketch curves in it.
- Expand the drawing view and use the curve tools to construct 2D geometry inside the view.

Application	Drafting
Command Finder	**Drawing View** 
Shortcut menu	Right-click a drawing sheet boundary→**Add Drawing View**
Part Navigator	Right-click a sheet node→**Add Drawing View**

1.4.4 Measure the part to output the required size

1.4.4.1 Interactive on-screen controls for dimensions

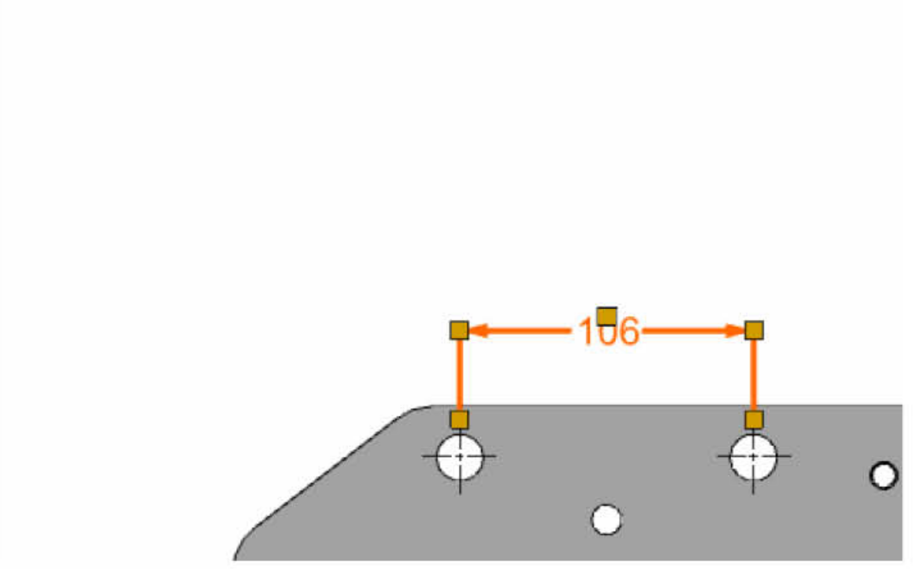
It is possible to set and change the associativity and appearance of dimensions, dimension lines, and extension lines using interactive, on-screen controls in NX These controls include:

- The ability to let NX automatically position and place a dimension. This allows creating a dimension with the fewest number of mouse button clicks.
- Interactive, on-screen controls for setting and changing the associativity and appearance of dimensions, dimension lines, and extension lines.

  These controls include:

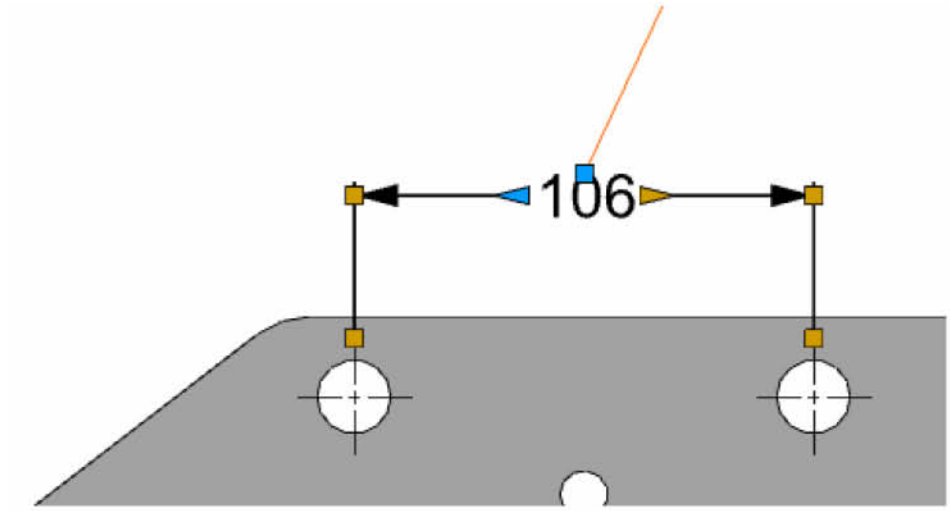
  - Access handles that, when selected, display a scene dialog for modifying frequently used options and settings for the annotation object.

    Note Only one access handle can be active at a time.

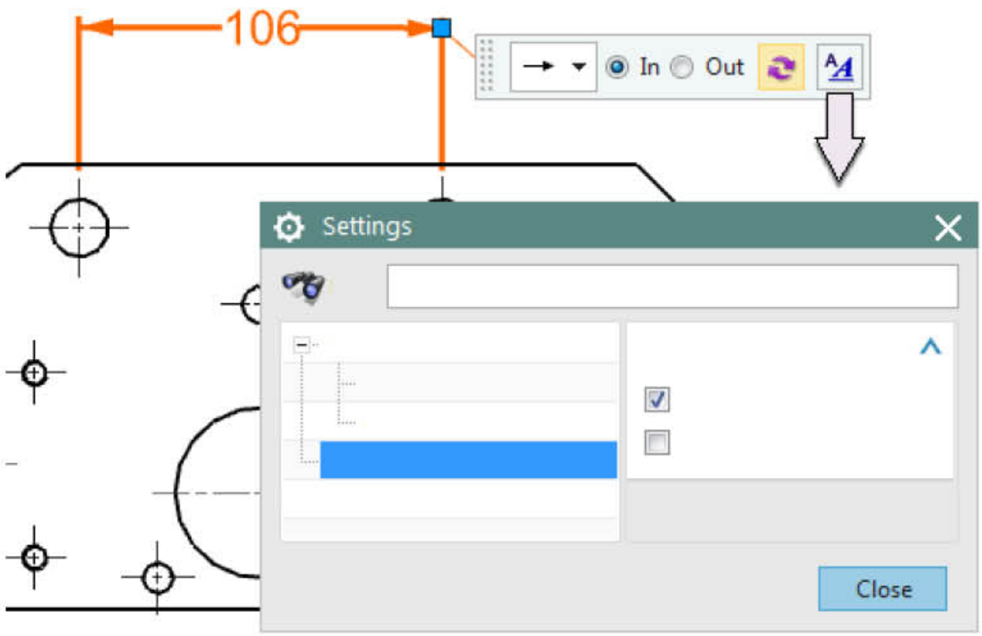
  - 

    Tip
    - The **Pause delay for on-screen dialog** Drafting preference and customer default allows adjusting the amount of time, in seconds, that are required to pause the cursor before scene dialogs appear.
    - Press F3 to suppress the display of all scene dialogs.

  - Drag handles for repositioning dimension and extension lines.



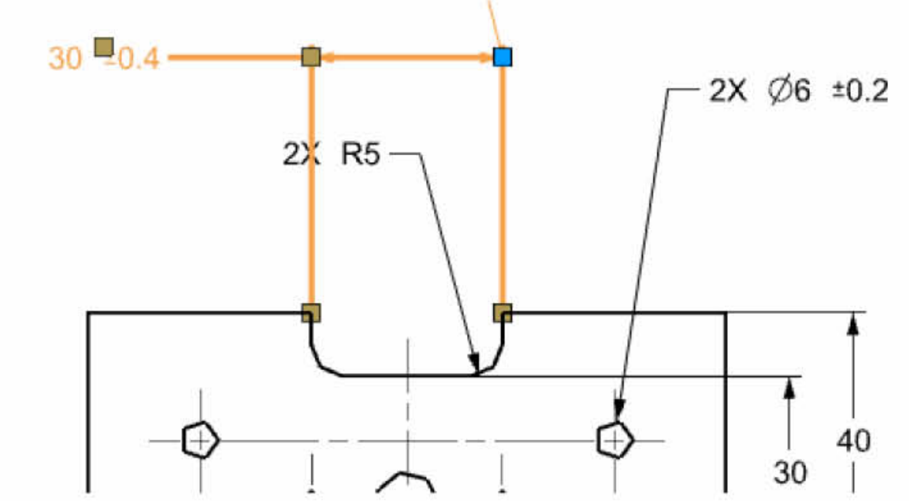
- **Settings** buttons for access to additional options associated with the annotation object.



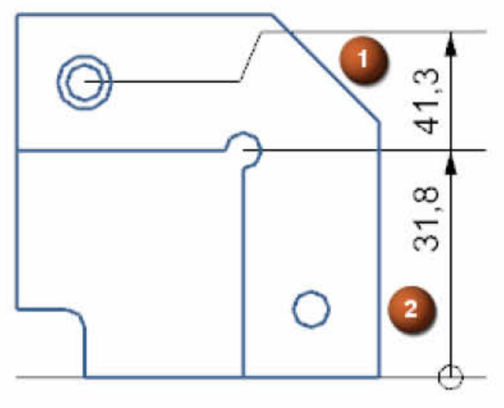
Graphics window	**When creating a dimension:** • Pause the cursor until the scene dialog appears. Use the scene dialog to immediately start editing the textual content of a dimension. Note Click the **Edit** button to enter the edit mode and access additional scene dialogs. **To edit an existing dimension:** • Double-click the dimension to enter the edit mode, or right-click the dimension and choose **Edit**.

1.4.4.2 Jogs on extension lines

Extension lines connect the dimension to the object being dimensioned and can be straight or have a jog. It is possible to add or remove jogs while creating or editing linear and ordinate dimensions. Interactive on-screen handles and controls help to add, place, and size the jog for one extension line or for both extension lines.



For ordinate dimensions, jogs are inferred by default. NX determines when a jog is added to the extension line. For linear dimensions, jogs are not inferred and it is a must to manually add them to the extensions lines of a linear dimension either while creating it or after it is created.



1. Ordinate extension line with a jog
2. Ordinate extension line without a jog

Interactively add or remove jogs

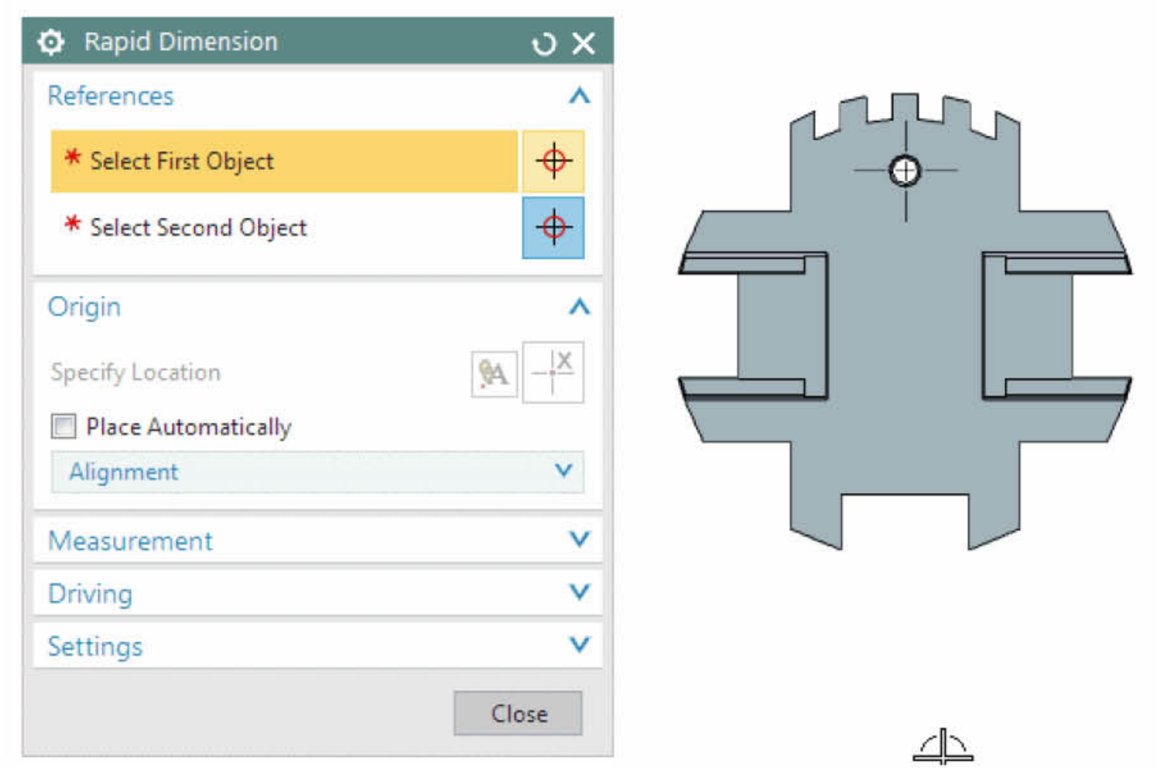
Application	Drafting and PMI
Graphics window	Double-click an existing dimension, or while creating a dimension, click the **Edit** on-screen button to enter the dimension edit mode, and then click the Extension Line access handle to display the on-screen controls.

Set the jog parameters

1.4.4.3 Inferring rapid and linear dimensions from a selection

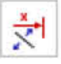
The selection process for creating dimensions using the **Rapid** or **Linear** commands is unique in that, for some measurement types supported by these commands, NX can infer the dimension from a single selection.

For example, if the vertical edge of a part is selected (and not a point on the edge), NX will infer a vertical dimension. In this instance, **Specify Location** in the **Origin** group becomes the current option, and it is feasible to immediately click to place the dimension.



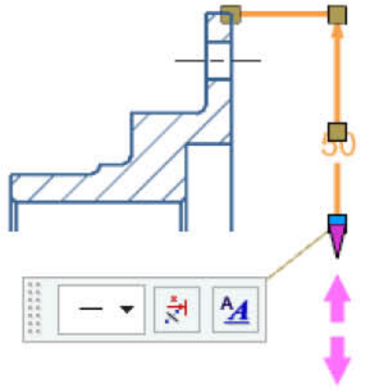
Sometimes NX may infer a dimension from a single mouse click unintentionally. In this case, the creation of a different dimension can be forced by moving the cursor over another valid drafting object. When placing the cursor over another object, **Select Second Object** in the **References** group becomes the current option, and the second object for the dimension can be selected. Once the second object is selected, the **Specify Location** option in the **Origin** group becomes active again, and then it is possible to click to place the dimension.

1.4.4.4 Single-sided dimensions

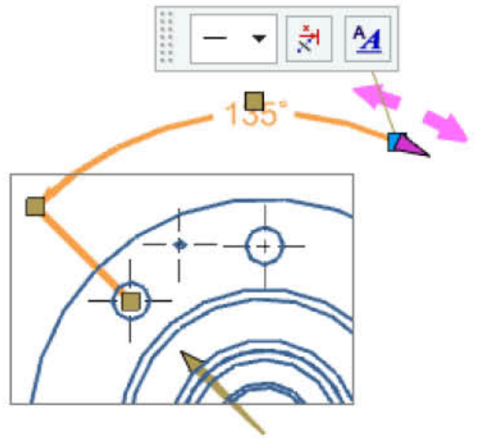
The **Display as Single Sided Dimension** option helps to create and edit linear and angular dimensions with only a single leader line. Use single-sided linear or angular dimensions for geometry that is larger than the size of the drawing view or sheet, or where the measurement origin is associated to geometry that is not visible. While creating or editing a single-sided dimension, you can use the **Flip Dimension Side**  button to switch between dimension lines.

Note Chain dimension sets and narrow dimensions are not supported. Also, it is impossible to create single-sided dimensions using the **Rapid** dimension command.

When in edit mode, access handles are available to indicate which extension line and arrow to retain. The access handle can also be used to set the length of the remaining, stunted arrow line. Alternately, it is also possible to set the length of the remaining line using the **Length** option in the **Settings** dialog box.



Example of a linear single–sided dimension



Example of an angular single–sided dimension

Set the single sided option

Application	Drafting and PMI
Graphics window	Right-click while creating a dimension→**Settings** Double-click an existing dimension→**Settings**
Location in dialog box	**Single Sided** node→**Display as Single Sided Dimension**

1.4.4.5 Flip Dimension Side button

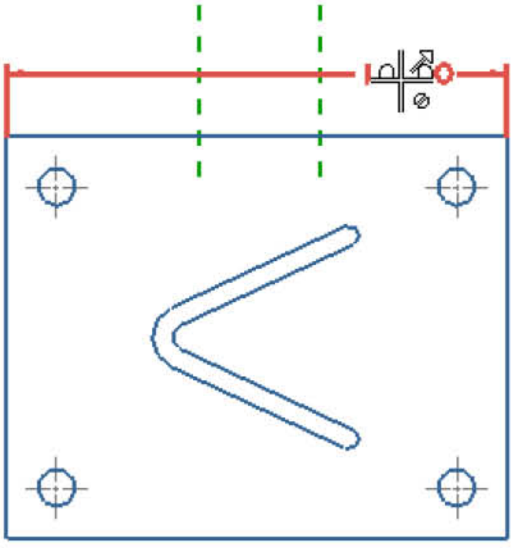
Application	Drafting, PMI
Graphics window	While in edit mode, click the Arrow Line access handle, and then turn the **Flip Dimension Side** on-screen button on or off. Right-click while creating a dimension→**Settings** Double-click an existing dimension→**Settings**
Location in dialog box	**Single Sided** node→**Flip Dimension Side**

Set the length of the remaining arrow line

Application	Drafting, PMI
Graphics window	While in the edit mode, click and drag the Arrow Line access handle Right-click while creating a dimension→**Settings** Double-click an existing dimension→**Settings**
Location in dialog box	**Single Sided** node→**Length**

1.4.4.6 Auto-centering dimensions

When using **Manual Placement** options, dimension text automatically snaps to the center of the dimension line when it crosses within the range of an invisible snap corridor.

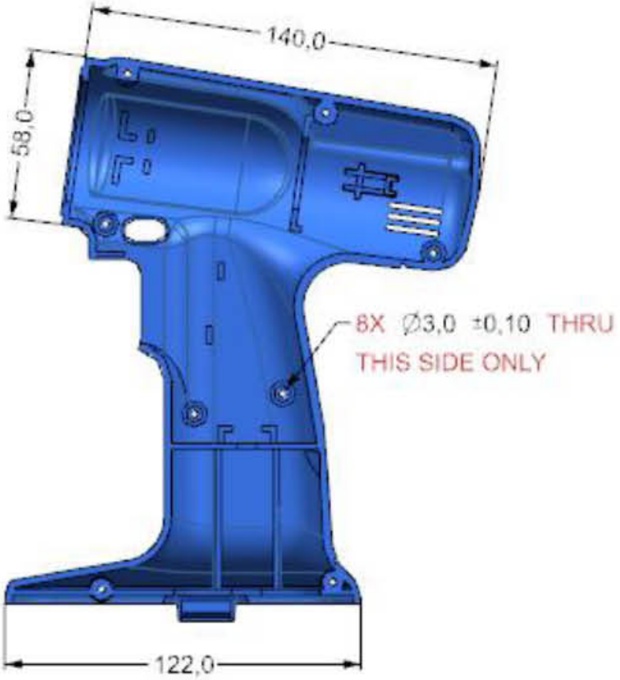


Invisible snap corridor shown here in green dashed lines

This behavior is useful when you want to center your dimension but do not want to use the **Automatic Placement** option.

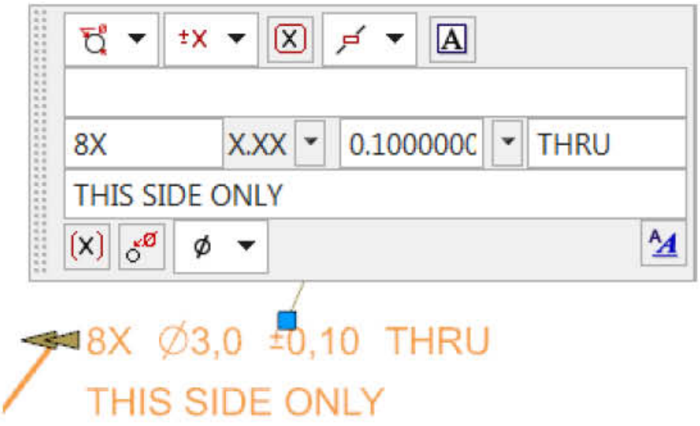
1.4.4.7 Appending text to dimensions

It is possible to add text before, after, above, or below the dimension while creating a dimension, or whenever editing a dimension.



1.4.4.8 Scene dialog

When placing or editing a dimension, a scene dialog appears in the graphics window. It can be used to interactively add, modify, or delete appended text.



A scene dialog can contain different options depending on the dimension type, and is available in Drafting and PMI, and in the Sketch task environment.

While creating a dimension

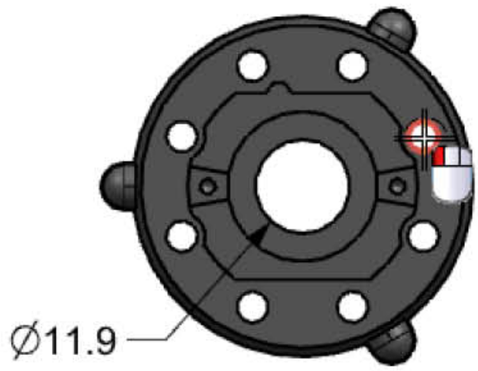
Application	Drafting
Graphics window	Pause before placing a new dimension, and then click inside the scene dialog wherever you need to apply appended text. Right-click and choose **Edit Appended Text** to open the **Appended Text** dialog box.

For an existing dimension

Application	Drafting
Graphics window	Right-click the dimension and choose **Edit Appended Text**. Double-click the dimension and, in the scene dialog, type the appended text in the appropriate box.

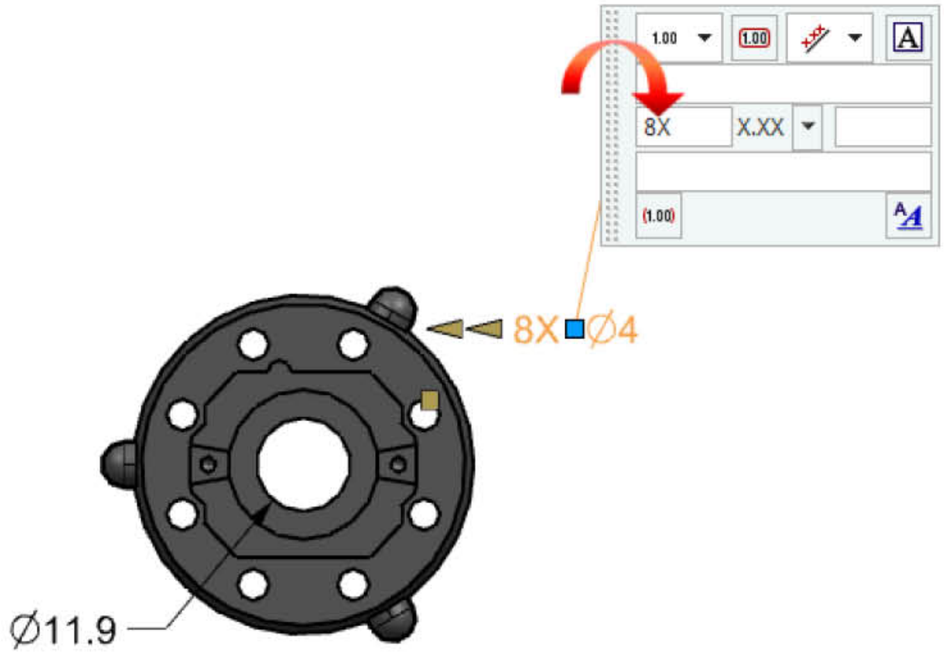
Add appended text to a new dimension

1. Choose **Home** tab→**Dimension** group→**Radial**.
2. In the **Measurement** group, from the **Method** list, select **Diametral**.
3. In the graphics window, select the hole.



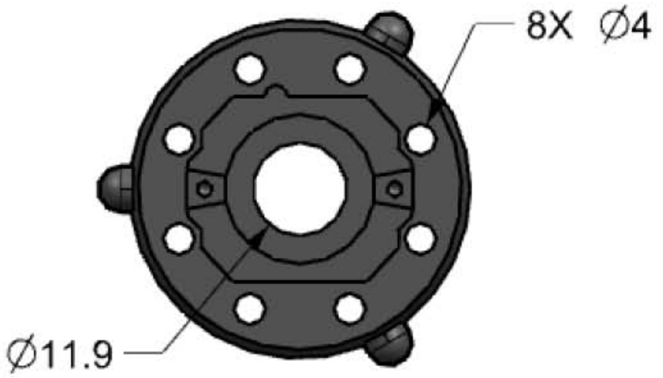
4. Pause the cursor in a temporary location until the on-screen **Text** dialog appears.

5. In the **Before** appended text box, type **8X**.



Tip Use Alt + any arrow key to quickly move to the associated appended text box.

6. Click to place the dimension.



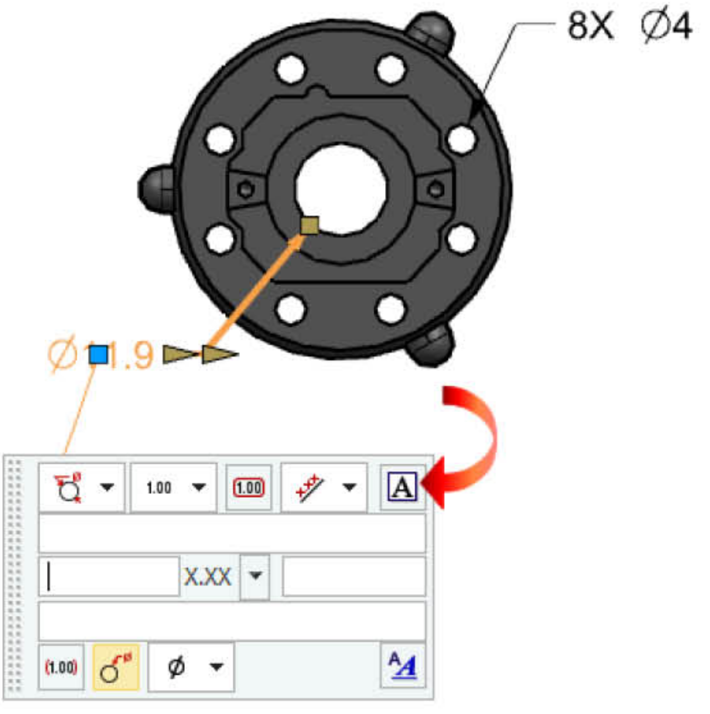
Add appended text to an existing dimension

1. Double-click an existing hole dimension to place it in edit mode.

   If there is a simple text to append to the dimension, add the text directly in the appended text boxes. This procedure demonstrates how to add more complicated text to a dimension.

2. If necessary, click the **Text** access handle.

3. In the scene dialog, click **Edit Appended Text** .

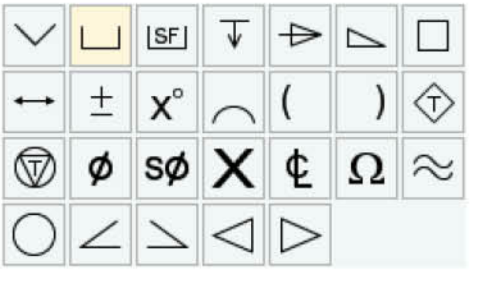


Tip To access more quickly the **Appended Text** dialog box, right-click any dimension and choose **Edit Appended Text**. It is not required to first place the dimension in an edit mode.

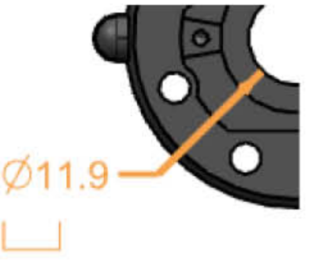
4. in the **Controls** group, from the **Text Location** list, select **Below**.
5. In the **Text Input** group, expand **Symbols**.

   Tip When not seeing the **Symbols** group, right-click the top border of the dialog box and select **Appended Text (More)**.

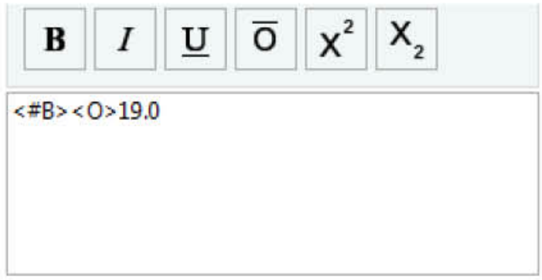
6. In the **Symbols** group, from the **Category** list, select **Drafting**.
7. Click **Insert Counterbore**.

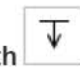


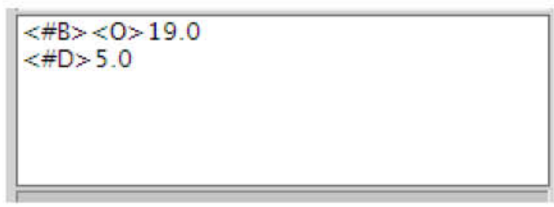
Note that a counterbore symbol appears below the hole dimension.



8. In the **Symbols** group, click **Insert Diameter** .
9. In the **Text Input** window, after the diameter characters, type **19.0**.



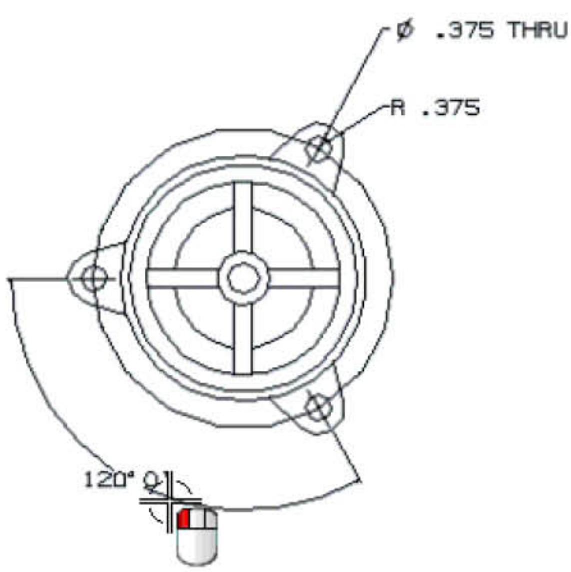
10. Press Enter and then click **Insert Depth** .
11. After the depth characters, type **5.0**.



12. Click **Close** twice to create the appended text.

Add appended text to multiple dimensions
This example shows how to simultaneously add appended text to three different dimensions

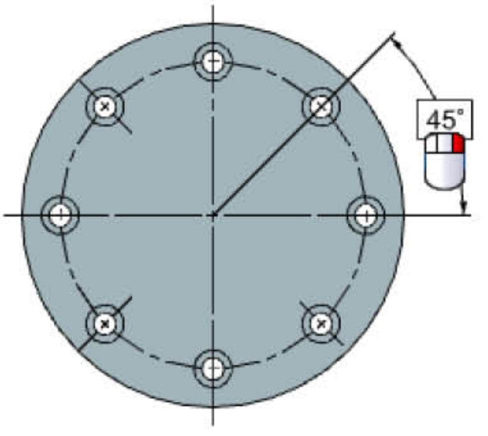
1. Select the "120° 0'" dimension.



2. Right-click and choose **Edit Appended Text**.
3. Select the diameter and radius dimensions.
4. In the **Appended Text** dialog box, in the **Controls** group, from the **Text Location** list, select **Before**.
5. In the **Text Input** window, type **3X**.
6. Click **Close** to save the changes.

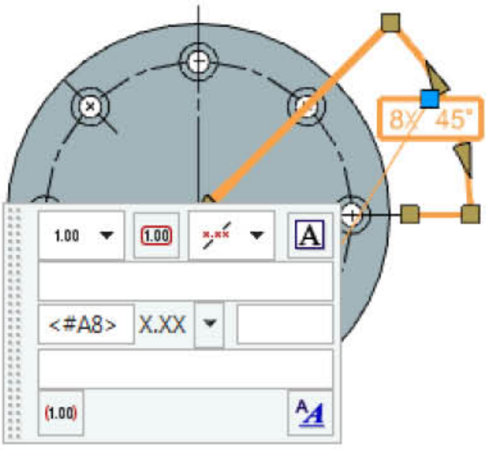
Add appended text to a basic dimension
This procedure demonstrates how to add appended text inside the box of a basic dimension.

1. Right-click the basic dimension and select **Edit**.

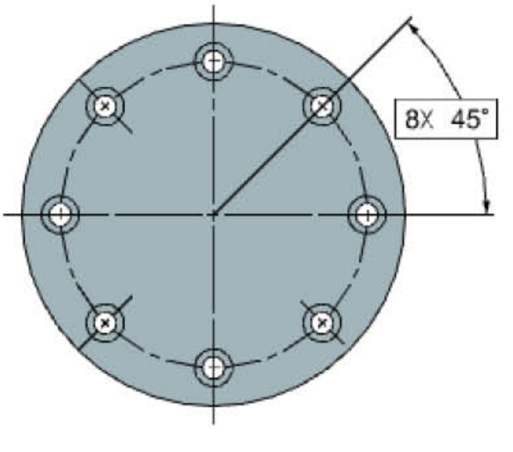


2. In the scene dialog, in the **Before** box, type **<#Anum>**, where **num** is equal to the number of additional objects.

   In this example, **<#A8>** is used.



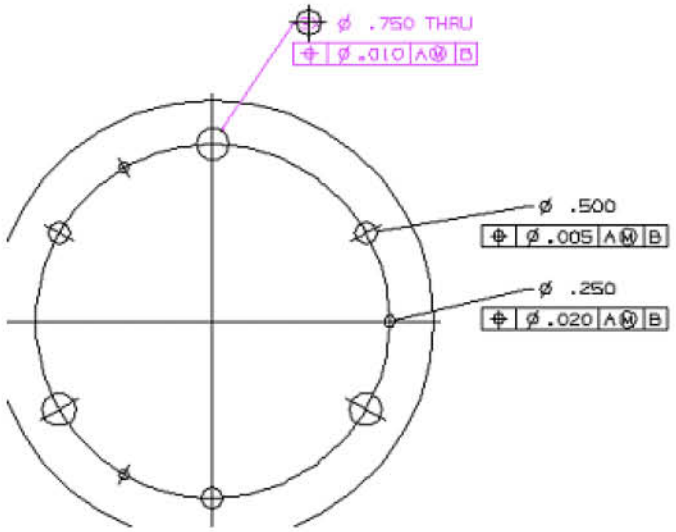
3. Click **Close** in the **Angular Dimension** dialog box.

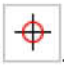


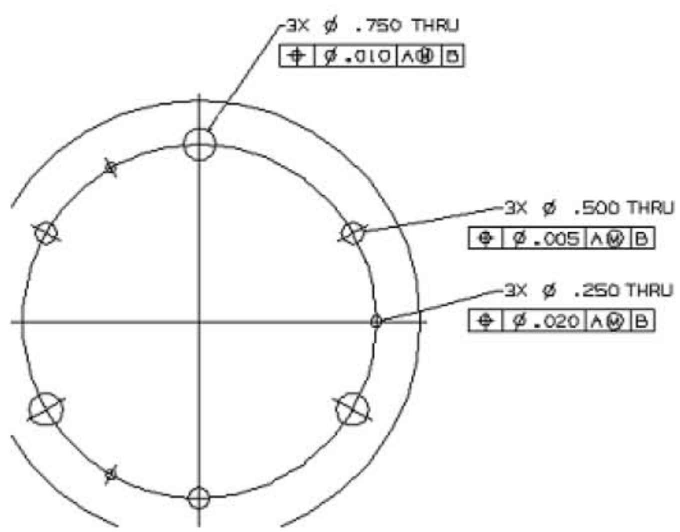
Add existing appended text to other dimensions

This example shows how to copy appended text from one dimension and apply it to other dimensions.

1. Place the cursor over the "3X" appended text of the .750 diameter dimension.



2. Right-click and choose **Edit Appended Text**.
3. In the **Appended Text** dialog box, in the **Controls** group, from the **Text Location** list, select **Before**.
4. In the **Dimension** group, click **Select Dimension** .
5. Select the .500 diameter dimension and the .250 dimension.
6. Click **Close**.

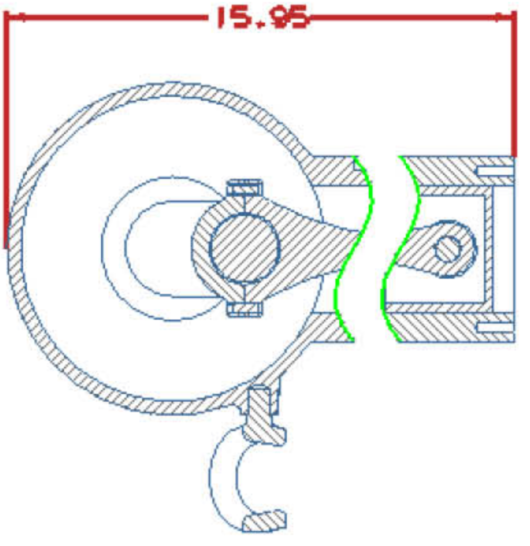


1.4.4.9 Replace a dimension's value with manual text

Note Sometimes, it is a must to replace associative dimension text with manual text, such as when you illustrate a tabular note with a dimension view. In such cases, manual dimension text does not update when the model objects associated with it change.

To create manual dimension text:

1. Right-click an existing dimension and choose **Settings**.



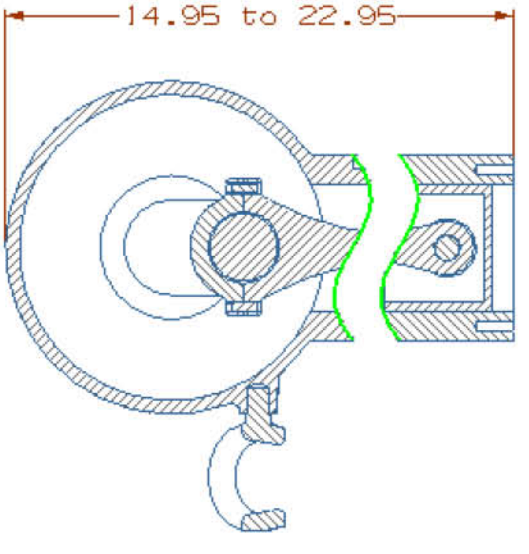
2. Expand the **Text→Format** node.

3. Select the **Override Dimension Text** check box.

4. For simple text, type a new value or text string in the text box, or click **Edit Text** to access all of the text editor options.

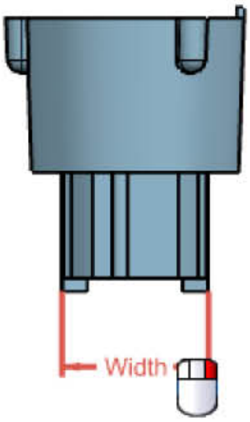
   Note A message appears whenever you manually edit the value of an associated dimension. Click **OK** to proceed.

5. In the **Settings** dialog box, type a new value and click **Close** to save it.

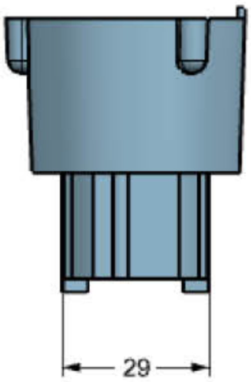


1.4.4.10 Revert manual dimension text to an associative value

1. In the graphics window, right-click the manual dimension and choose **Settings**.



2. In the **Settings** dialog box, expand the **Text** node, and then select the **Format** node.
3. In the **Format** group, clear the **Override Dimension Text**  check box.
4. Click **Close**.



The manual text is replaced with an associative value.

Tip To quickly identify any dimensions on the current drawing sheet with manual text in them, choose **Menu→Information→Other→Object-specific→Dimensions with Manual Text**. The dimensions will highlight on the drawing.

1.4.4.12 Rapid dimension

Rapid Dimension  helps quickly create different dimensions from a group of general, well-used dimension types using a single command and a basic set of selection options.

The following dimension types are supported for creation.

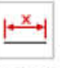
-  **Inferred** – determines the type of dimension to create based on the selected objects and the cursor location.
-  **Horizontal**
-  **Vertical**
-  **Point–to–Point**
-  **Perpendicular**

-  **Cylindrical**
-  **Angular**
-  **Radial**
-  **Diametral**

You can use the **Rapid Dimension** command to create the dimension from one of the supported dimension types. In edit mode, the selected dimension will invoke the dialog box associated with its dimension type.

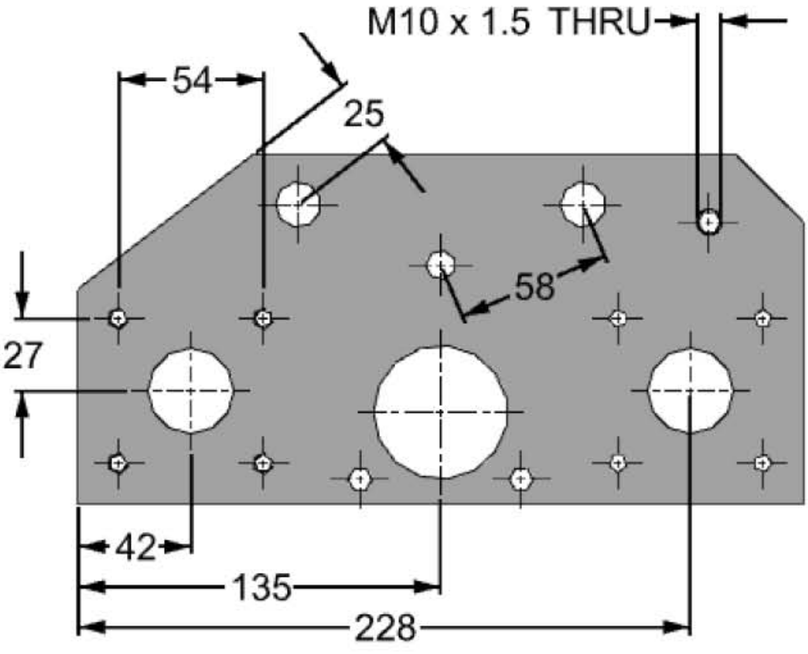
Application	Drafting and Layout
Command Finder	**Rapid Dimension** 

Linear Dimension

The **Linear Dimension**  command help to create one of six different types of linear dimensions as standalone dimensions or as a set of chain or baseline dimensions. It is possible to create the following dimension types:

- **Horizontal**
- **Vertical**
- **Point–to–Point**
- **Perpendicular**
- **Cylindrical**
- **Hole Callout** (in linear format, with or without secondary depth dimensions).

Note It is possible to create a set of chain or baseline dimensions using horizontal, vertical, point-to-point, or perpendicular dimension types.



Using the **Linear Dimension** command, to:

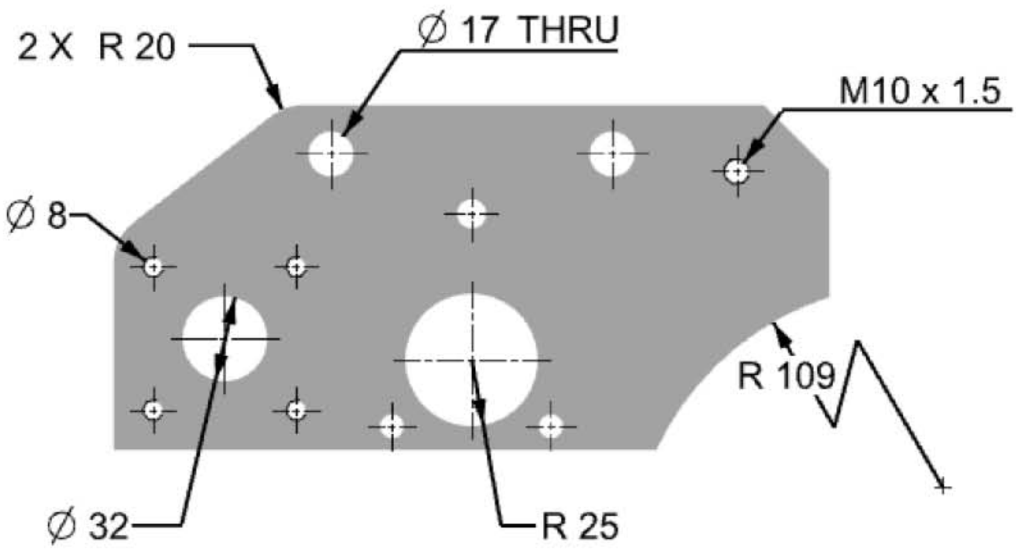
- Let NX infer the type of linear dimension to create based on the selected objects.
- Dynamically change the type of linear dimension while creating or editing the dimension.
- When dimensioning sketch curves on the drawing, determine whether or not the dimension will affect the sketch curves if it is changed. This is referred to as a driving dimension. This option is only visible for appropriate dimension types.
- Manually position the dimension, or let NX automatically place the dimension.
- Convert a baseline dimension set to a chain dimension set, and a chain dimension set to a baseline dimension set.
- Manually override the computed value of a dimension while creating or editing the dimension using the **Override Dimension Text** option in the **Settings** dialog box. This override option is available for all dimensions except driving dimensions and PMI dimensions.
- Set the measurement type to **Directed** or **Feature of Size**, which enables capture of additional design information. This information is especially useful in downstream applications such as tolerance analysis and inspection applications. The option to set the measurement type is available if the **Enable Directed Dimensions Drafting** preference is enabled. You cannot apply a measurement type to cylindrical, hole callout, chain, or baseline dimensions.

Application	Drafting and Layout
Command Finder	**Linear Dimension** 

Radial Dimension

The **Radial Dimension**  command lets you create one of five types of radial dimensions.

- **Radial**, with **Radius to Center** and **Folded Radius** variations
- **Diametral**
- **Hole Callout** (in radial format, with or without secondary depth dimensions).



Using the **Radial Dimension** command to:

- Let NX infer the type of radial dimension to create based on the selected objects.

- Dynamically change the type of radial dimension while creating or editing the dimension.
- When dimensioning sketch curves on the drawing, determine whether or not the dimension value will affect the sketch curves if it is changed. This is referred to as a driving dimension. This option is only visible for appropriate dimension types.
- Manually position the dimension, or let NX automatically place the dimension.
- Manually override the computed value of a dimension while creating or editing the dimension using the **Override Dimension Text** option in the **Settings** dialog box. This override option is available for all dimensions except driving dimensions and PMI dimensions.

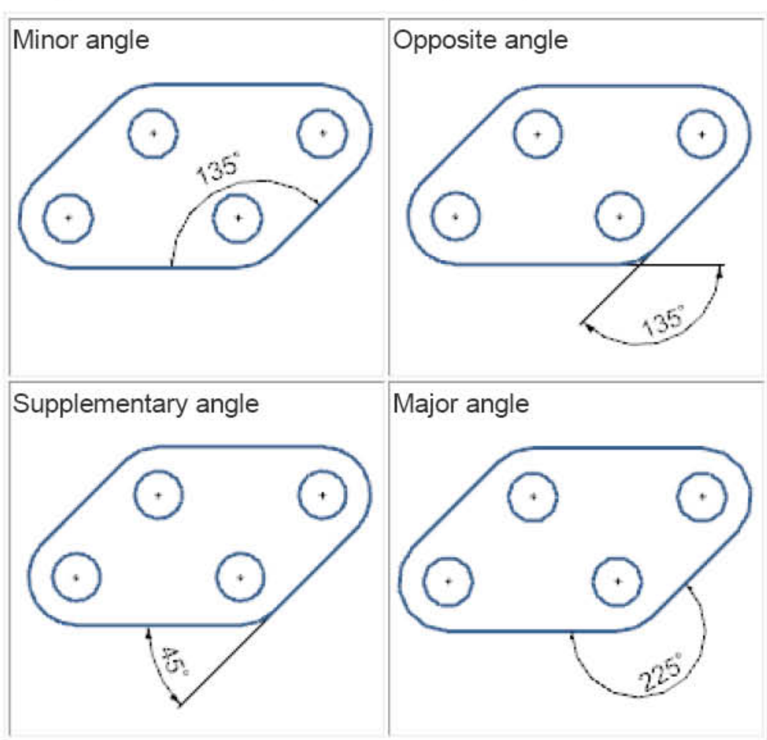
Application	Drafting and Layout
Command Finder	**Radial Dimension**

Angular Dimension

Use the **Angular Dimension** command to measure the angle between two objects in a view. The objects selected can be any combination of:

- Lines
- Model edges
- Cylindrical and planar faces
- Dimension extension lines
- Centerline symbol components
- User-defined vectors

When selecting two objects in a view, any one of the angles depicted below can be dimesioned:

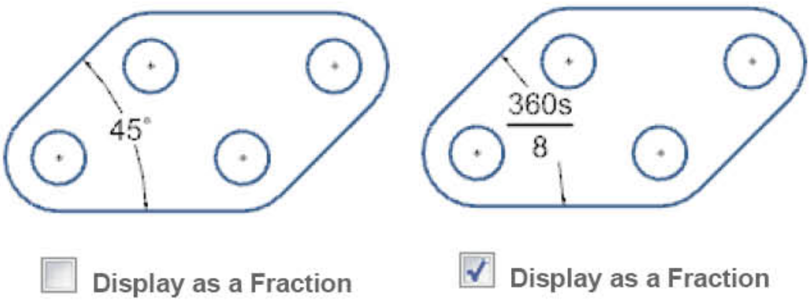


Angular dimension units

Angular dimension units can be displayed in either:

- Whole degrees
- Fractional degrees (in decimal or fractional format)
- Degrees and minutes
- Degrees, minutes, and seconds

It is possible to specify an angular dimension as a fractional division of an arc by setting the **Display as a Fraction** preference.



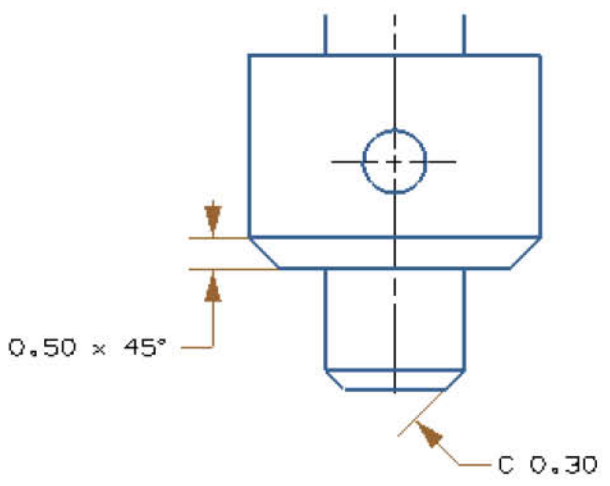
Application	Drafting and Layout
Command Finder	**Angular Dimension**

Display as a Fraction

Menu	**Preferences**→**Drafting**→**Dimension** node→**Text** node→**Units** page→**Display as a Fraction**

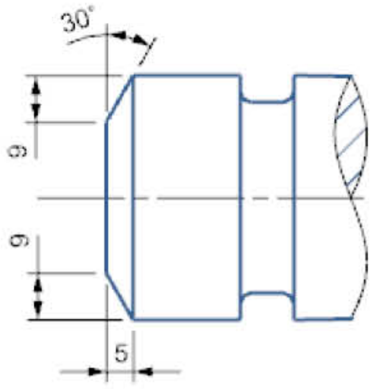
Chamfer Dimension

It is possible to apply the **Chamfer Dimension** to edges chamfered at 45 degrees.



When dimensioning a chamfered edge that is angled 45 degrees from an adjacent linear edge, no further selections are required. If an adjacent linear edge cannot be detected, then it is a must to either select an adjacent linear edge or manually create the chamfer dimension.

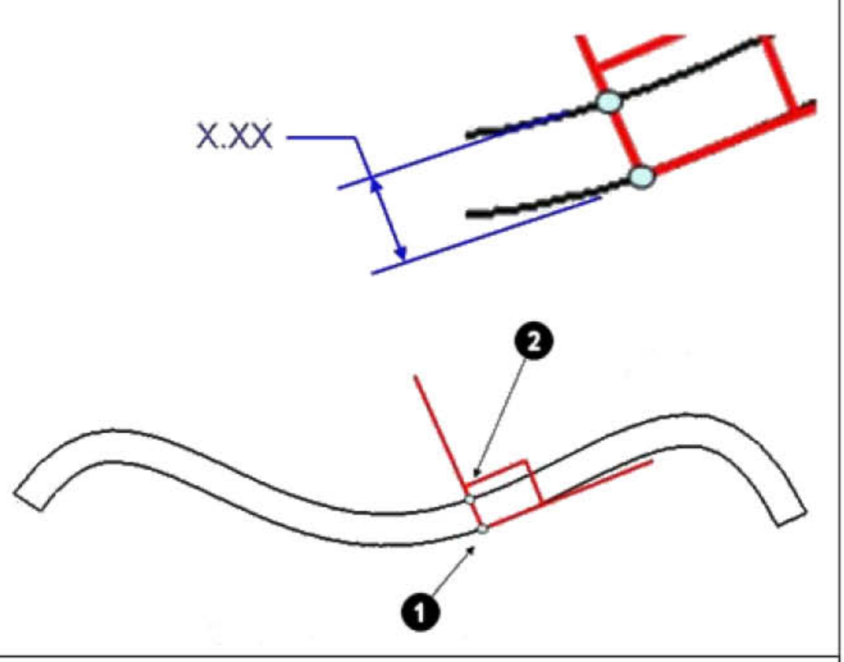
Note For any chamfer that varies from 45 degrees, it is required to use other dimensioning techniques to dimension it.



Application	Drafting and Layout
Command Finder	**Chamfer Dimension**

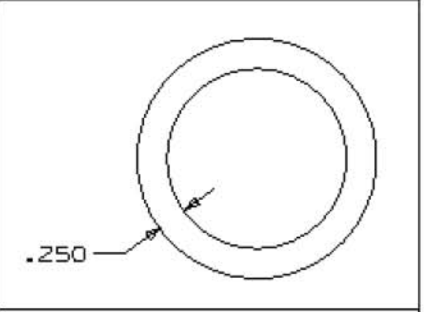
Thickness Dimension

The **Thickness Dimension** command helps to create a dimension between two curves (including splines). The thickness dimension measures the distance between a point on the first curve and the intersection point on the second curve. It measures in the normal direction from the point specified on the first curve.



1	**Object selection (point on curve used for dimension)**
2	**Object selection**

If the two curves are arc-like objects (arc, splines shaped like arcs, cylindrical faces, or circular centerlines), NX creates a concentric circle dimension, which measures the difference in radii between two concentric arcs. It then computes the numerical difference between the two radii.



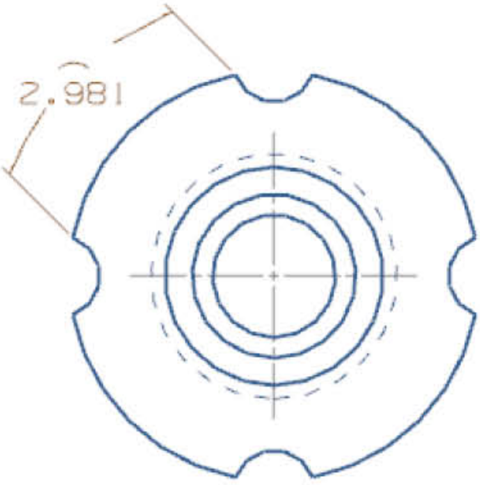
Concentric circle dimension

Note Non-concentric curves, while selectable, are not supported by the thickness command and erroneous results will be displayed. Use a different dimension command if you need a dimension between two non-concentric curves.

Application	Drafting and Layout
Command Finder	**Thickness Dimension**

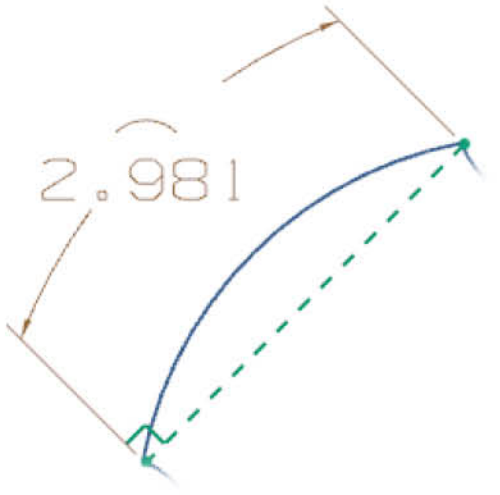
Arc Length Dimension

Use the **Arc Length Dimension** command to dimension the distance along the perimeter of a circular segment.



Arc length dimension

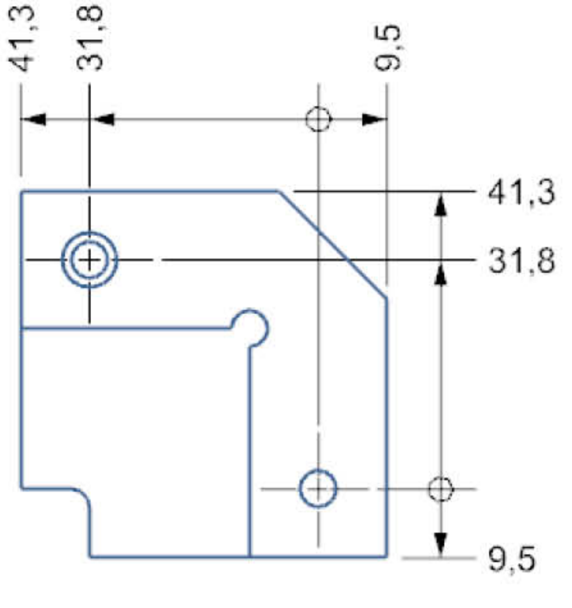
Typically, an arc length dimension is used to measure the travel distance around a camera or to measure the length of a tube or cable. It is recognizable by its two radial arrow segments and by the arc symbol that appears above the dimension text. The dimension's linear extension lines are oriented perpendicular to the chord of the arc segment being dimensioned.



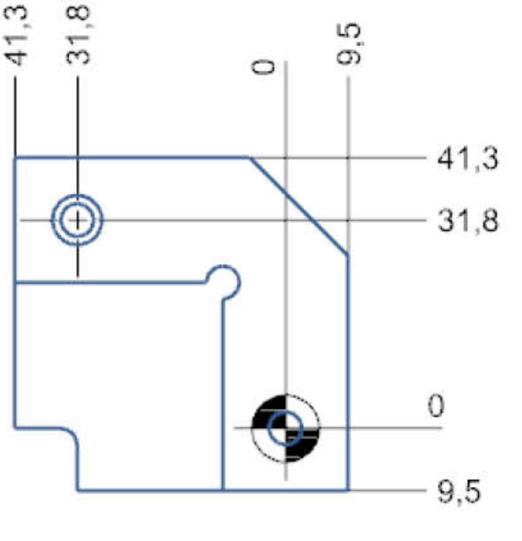
Application	Drafting and Layout
Command Finder	**Arc Length Dimension**

Ordinate dimensions

Use the **Ordinate Dimension** command to measure the linear distance from a common origin point to an object in the view. An ordinate dimension generally consists of dimension text and a single extension line, and can be displayed either with or without a dimension line.



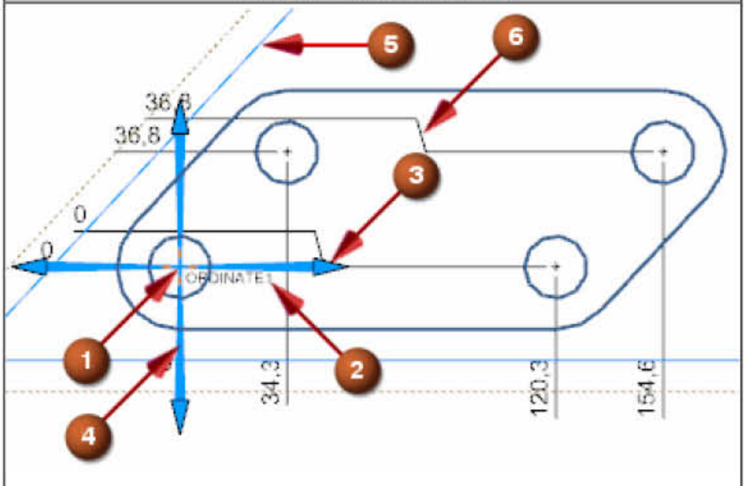
With dimension lines



Without dimension lines

In the figure below, all of the elements that typically comprise an array of ordinate dimensions are shown. The ordinate origin, name, baselines, margins, and associated dimensions are collectively referred to as an *ordinate dimension set*.

Ordinate dimension set



1. Ordinate origin

The common start point for all of the ordinate dimensions.

2. Ordinate set name

Identifies the ordinate set on the drawing sheet.

3. Baseline

Points in the X direction of measurement from the origin.

4. Perpendicular Baseline

Points in the Y direction of measurement from the origin.

5. Margins

Align the dimensions with each other.

6. Ordinate Dimension (with jog)

Measures the distance from the ordinate origin to objects in the view.

Multiple ordinate dimension sets can be placed on a single drawing sheet or even within a single view. It is also possible to select an existing ordinate origin in one view to dimension an object in another view. The ordinate origin can be in any view on any drawing sheet.

Note When an ordinate dimension is associated with an origin in a different view, it is not copied when the view it appears in is copied

When you create an ordinate dimension set, it is recommended to confine all of its defining elements to a single view. Placing an ordinate origin in one view and its margins in another view can lead to undesirable results.

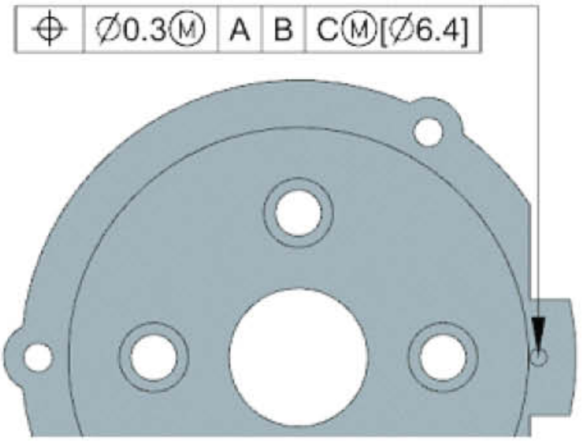
As a rule, you create ordinate dimension sets on the drawing sheet just like any other dimension. You can, however, create them in an expanded drawing view. If you do that, you must also perform any subsequent edits to the ordinate set within the expanded view.

Application	Drafting
Command Finder	**Ordinate Dimension**

1.4.5 Advanced tools in 2D drawing export

1.4.5 1 Feature Control Frame

Use the **Feature Control Frame** command to specify the geometric tolerances applied to the features of a model.



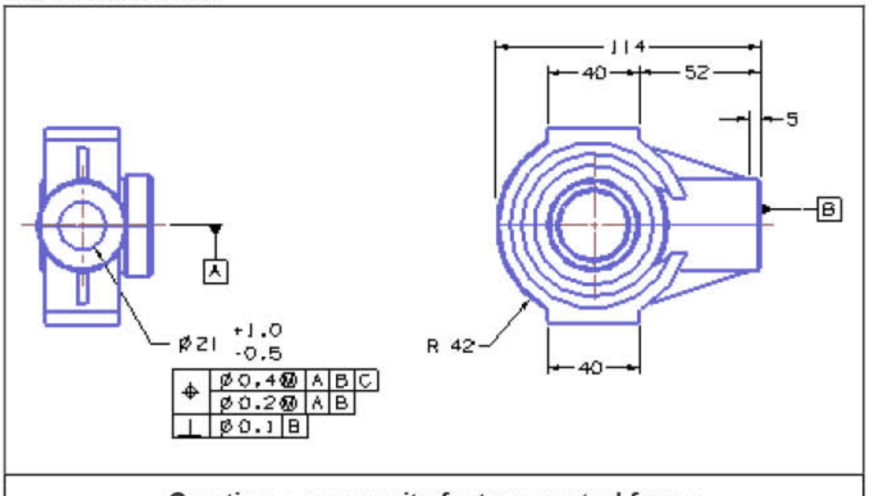
You can create and edit the following, with or without leaders:

- Single line feature control frames.
- Multi-line tolerance frames.
- Composite feature control frames.
- Composite feature control frames with one or more additional tolerance frames underneath.

You can also attach any of the above feature control frames to an existing dimension.

Application	Drafting and Layout
Command Finder	**Feature Control Frame**

Create a feature control frame



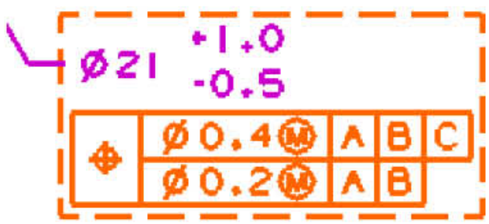
Creating a composite feature control frame

Note The following example creates a feature control frame from the Drafting application. You can also create the same feature control frame from PMI.

Create a composite Feature Control Frame (FCF)

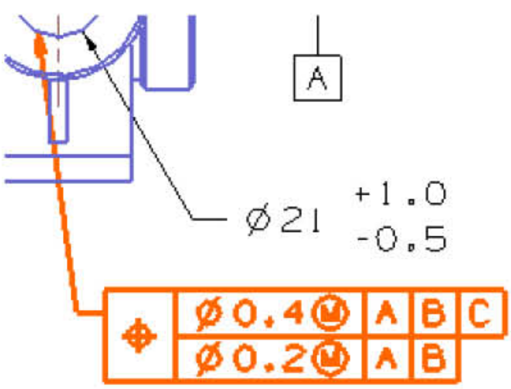
1. Choose **Home** tab→**Annotation** group→**Feature Control Frame** .
2. In the **Alignment** group, select **Stack Annotation** and **Align Horizontal or Vertical**.
3. In the **Frame** group, set **Characteristic** to **Position** , and **Frame Style** to **Composite Frame** .
4. Make sure **Frame 1** is highlighted in the **List** box.
5. Set the **Tolerance** options:
   a. Set the tolerance shape to **Diameter** Ø.
   b. Type **0.4** in the Tolerance Value box.
   c. Set the Tolerance Material Modifier to **Maximum Material Condition** Ⓜ.
6. Set the **Primary Datum Reference** to **A**.
7. Set the **Secondary Datum Reference** to **B**.
8. Set the **Tertiary Datum Reference** to **C**.
9. Select **Frame 2** from the **List** box.
10. Set the tolerance and datum references as desired.

    In this example, the tolerance is set to 0.2, the **Primary Datum Reference** is A, and the **Secondary Datum Reference** is B.
11. To position the composite FCF, do one of the following:
    - Position the FCF so that it is stacked below the hole dimension.

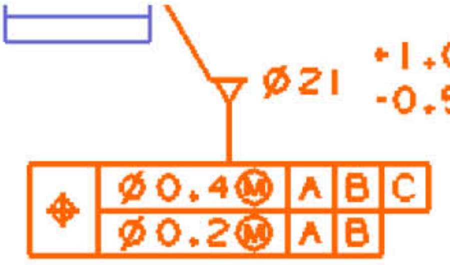


- In the **Leader** group, click **Select Terminating Object**, then click the hole edge to attach the leader.

  Note Make sure the **Point on Curve** option in the Selection bar is available before selecting the hole edge.



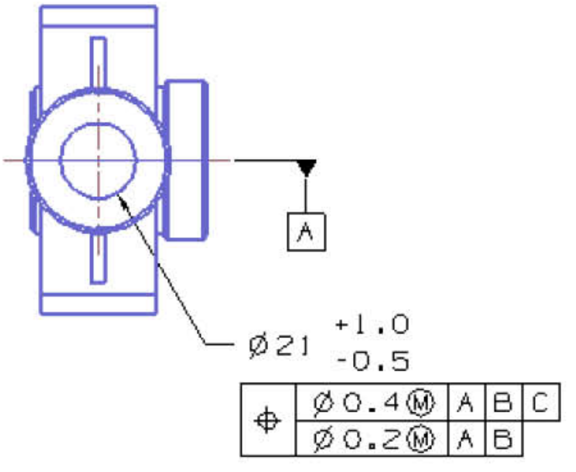
- In the **Leader** group, set the leader **Type** to **Datum**, then click the hole dimension stub and drag to position the FCF.



Note

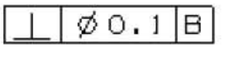
To add a third row to the composite frame, you would click **Add New Frame** .

12. Click to place the FCF.

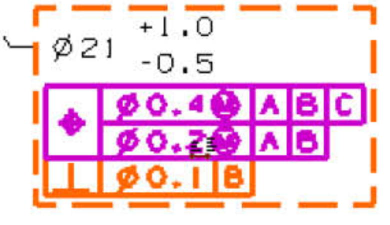


1.4.5.2 Add a single FCF

1. With the **Feature Control Frame** dialog box still open, in the **Frame** group set the **Characteristic** to **Perpendicularity** ⊥ and the **Frame Style** to **Single Frame** .
2. Set the **Tolerance** and **Primary Datum** options to produce an FCF as shown below.

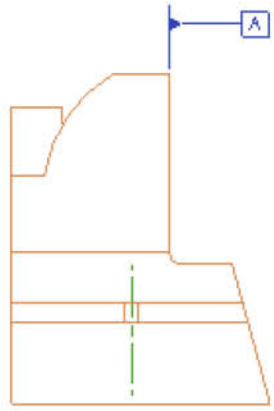


3. Drag the new FCF until it stacks below the composite FCF.



4. Click to place the symbol, then click **Close** to close the dialog box.

5. **Datum Feature Symbol**
6. Use the **Datum Feature Symbol** command to create a GD&T datum feature symbol (with or without a leader) to indicate a datum feature on your drawing.



Datum Feature Symbol (A)

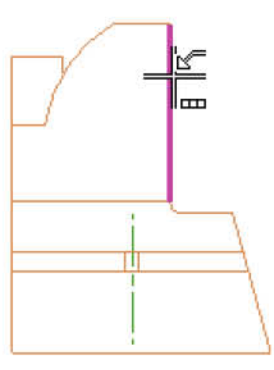
Application	Drafting and Layout
Command Finder	**Datum Feature Symbol**

1.4.5.3 Create or Edit a Datum Feature Symbol

Create a Datum Feature Symbol

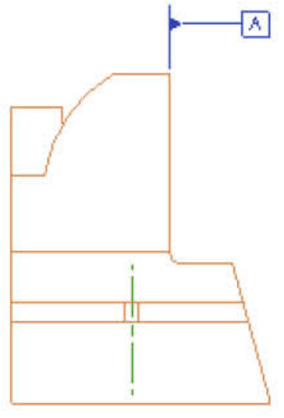
1. Choose **Home** tab→**Datum Feature Symbol** .
2. [Optional] In the **Leader** group, select the leader **Type** you want.
3. [Optional) Set the **Style** options for the leader type.

   For example, to create a filled datum triangle feature, set the leader **Type** to **Datum**. then set the **Arrowhead** style to **Filled Datum**.
4. In the **Datum Identifier** group, verify the correct datum letter is displayed in the **Letter** box.
5. Highlight an edge with your cursor.



Highlight an edge.

6. Click and drag to create the datum feature symbol and extension line.
7. Click once to place the symbol.



Place the symbol.

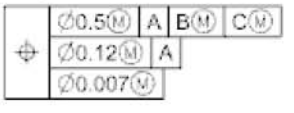
Note Use the Shift key to position both the datum leader line and the datum extension line while placing a datum with a **Datum** leader type.

1.4.5.4 Edit a Datum Feature Symbol

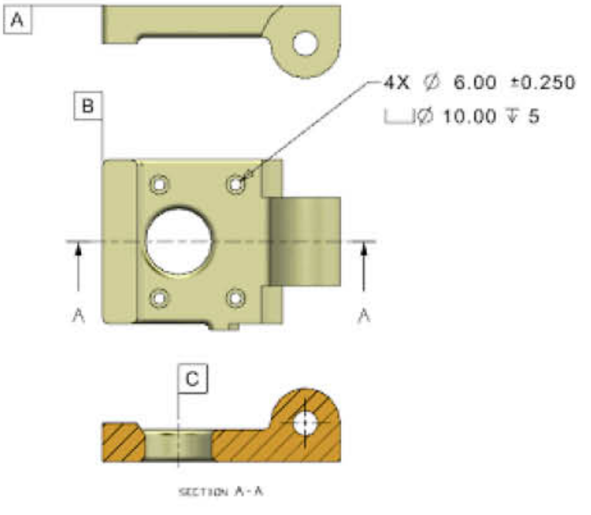
1. Double-click a datum feature symbol.
2. On the **Datum Feature Symbol** dialog box, from the **Settings** group, click **Setting** .
3. In the **Settings** dialog box, modify your settings.
4. Click **OK**.

1.4.5.5 Create a composite Feature Control Frame

This example shows how to create a three-row composite feature control frame.



1. Open a part and start the Drafting application.



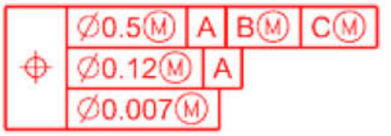
2. Choose **Home** tab→**Annotation** group→**Feature Control Frame** .

3. In the **Frame** group, set the following:
    - **Characteristic = Position**
    - **Frame Style = Composite Frame**
4. In the **Tolerance** subgroup, set the following:
    - Shape = **Diameter**
    - Value = **0.5**
    - Modifier = **Maximum Material Condition**

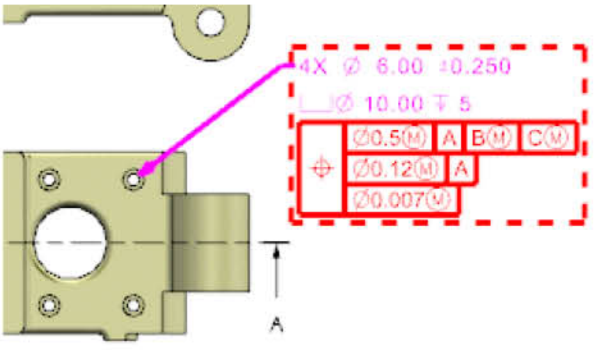
5. In the **Primary Datum Reference** subgroup, set the reference letter to **A**.
6. In the **Secondary Datum Reference** subgroup, set the following:
    - Reference letter = **B**
    - Modifier = **Maximum Material Condition**
7. In the **Tertiary Datum Reference** subgroup, set the following:
    - Reference letter = **C**
    - Modifier = **Maximum Material Condition**

8. In the **List** subgroup, select **Frame 2**.
9. In the **Tolerance** subgroup, set the following:
    - Shape = **Diameter**
    - Value = **0.12**
    - Modifier = **Maximum Material Condition**
10. In the **Primary Datum Reference** subgroup, set the reference letter to **A**.

11. Click **Add New Frame**.
12. In the **Tolerance** group, set the following:
    - Shape = **Diameter**
    - Value = **0.007**
    - Modifier = **Maximum Material Condition**



13. In the **Alignment** group, select **Stack Annotation**.
14. Drag the feature control frame so that it stacks beneath the counterbore hole annotation, and click to place it.



1.4.5.6 Add text to your feature control frame

You can add text before, after, above, and below a Feature Control Frame (FCF) while you are creating it.

1. Create the feature control frame but do not place it on the drawing.

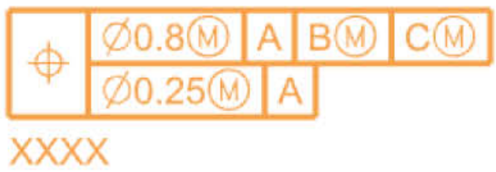
2. Expand the **Text** group of the **Feature Control Frame** dialog box.

   Note that control code for the FCF, <My Feature Control Frame>, already exists in the input box
3. In the **Text** group, use one of the following methods to add text to the feature control frame:
   - Place your cursor in front of the<My Feature Control Frame> control code and then type your text to add it to the beginning of the FCF.

   - Place your cursor at the end of the <My Feature Control Frame> control code and type your text to add it to the end of the FCF.

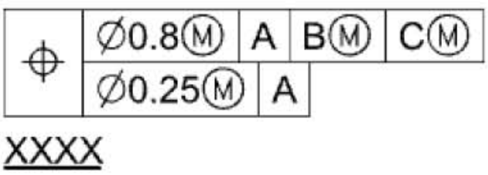
   - Place your cursor before the <My Feature Control Frame> control code, hit Enter to add a blank line, then place your cursor on the blank line and type your text to add it to the top of the FCF

- Place your cursor at the end of the <My Feature Control Frame> control code, hit Enter to add a blank line, and then type your text to add it to the bottom of the FCF



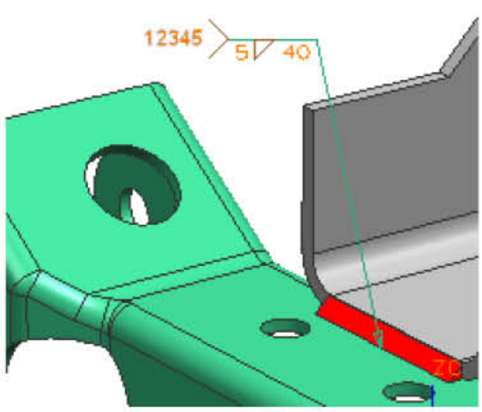
4. (Optional) You can underline the text by doing the following:
   a. Right-click in the graphics window and choose **Settings**.
   b. Select the **GDT** node.
   c. In the **Feature Control Frame** group, select an option from the **Underline Additional Text** list.

      In this example, **All** is selected.
   d. Click **Close** to exit the **Settings** dialog box.
5. Click to place the FCF, or select a point and then click and drag to place the FCF with a leader line.



1.4.5.7 Weld Symbol

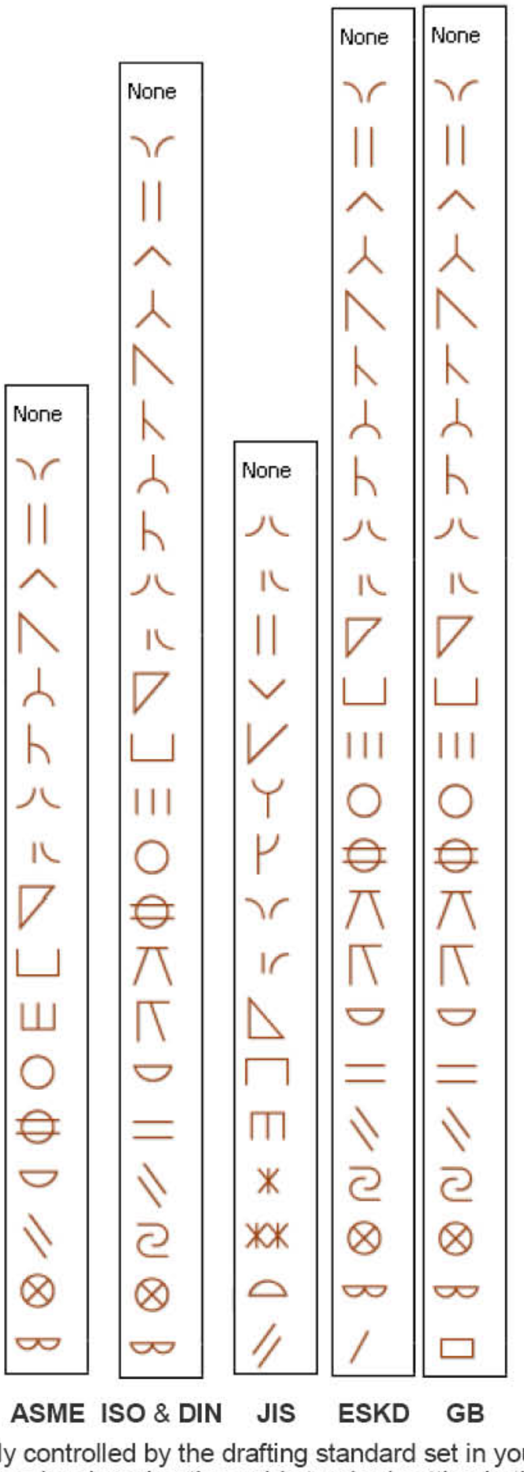
Use the **Weld Symbol**  command to create various weld symbols in both Metric and English parts and drawings. Weld symbols are associative and relocate when the geometry they are associated to changes or is flagged as out of date. You can edit weld symbol properties such as text size, font, scale and arrow dimensions.



Weld Symbol Standards

You can create ASME, ISO, DIN, JIS, ESKD and GB standards compliant weld symbols.

Following are examples of symbols available above and below the continuous reference line.



Weld symbol types are initially controlled by the drafting standard set in your part. You can change the available weld symbol types by changing the weld standard setting in the **Drafting Preferences** dialog box.

Application	Drafting and Layout
Command Finder	**Weld Symbol**

Set or change the weld standard setting

Application	Drafting
Command Finder	**Drafting Preferences**
Location in dialog box	**General/Setup**→**General** node→**Standard** group→**Weld**

Crosshatch

Use the **Crosshatch** command to fill a specified area with a pattern. A crosshatch object includes the hatch pattern and defining boundary entities.

You can define a crosshatch boundary area with a single selection inside a face or a closed loop of supported curve types such as solid edges, section edges, silhouettes, and basic curves.

Note You can not add crosshatch objects to perspective views. Perspective views are views which have **Use Perspective** set in the **Settings** dialog box for the view.

You can also define the boundary by selecting objects. To define a boundary, you can select curves, solid silhouettes, solid edges, and section edges. Text islands are created when you also select notes, PMI annotations, and GD&T symbols. Make sure that you select the boundary objects in consecutive (connected) order to keep the shape of the crosshatch predictable.

When adding crosshatches to a non-master drawing part, or to a drawing contained within an assembly, you cannot select a mix of part objects and component objects to define the boundary. However, you can define a boundary made up entirely of part objects (such as silhouette curves) or component objects (such as component edges). If you want to define a crosshatch boundary using a mixture of part objects and component objects, you must first display the view with extracted edges.

Note If you change your view **Representation** setting from **Exact** to **Smart Lightweight**, or if you change the **Resolution** setting of your **Smart Lightweight** view, existing crosshatch patterns may become retained.

Where do I find it?

Application	Drafting and Layout
Command Finder	**Crosshatch**
Graphics Window	Double-click a crosshatch pattern Right-click a crosshatch pattern →**Edit**

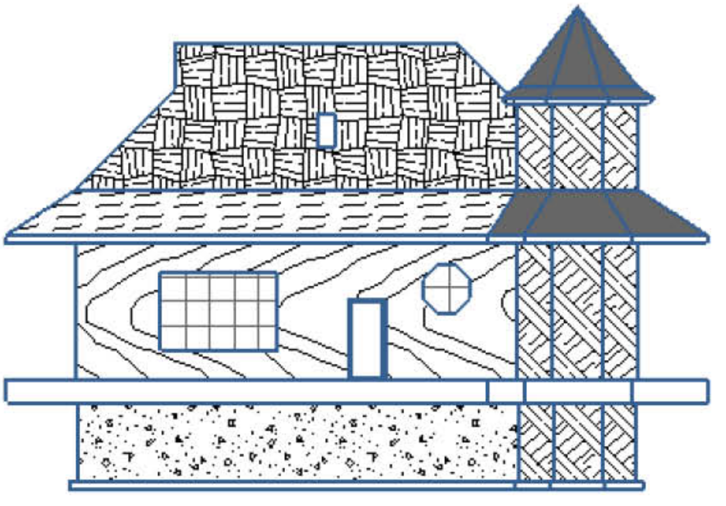
Area Fill

Use the **Area Fill**  command to create an area fill object which consists of a specified pattern of complex lines enclosed by a set of boundary curves. **Area Fill** also includes solid fill which fills the inside of the boundary with color or gray scale.

Note You can not add area fill objects to perspective views. Perspective views are views which have **Use Perspective** set in the **Settings** dialog box for the view.

You use the **Area Fill** dialog box to both create and edit attributes of an area fill object, including boundary curves, pattern type, and color, width, angle and tolerance of the pattern lines.

Ten ANSI Y14.2M fill pattern styles are available representing ten common patterns.



Note If you change your view **Representation** setting from **Exact** to **Smart Lightweight**, or if you change the **Resolution** setting of your **Smart Lightweight** view, existing area fill patterns may become retained.

Where do I find it?

Application	Drafting and Layout
Command Finder	**Area Fill** 
Graphics Window	Double-click an area fill pattern Right-click an area fill pattern →**Edit**

Assembly drawings

Like single part files, you can create your drawing directly in your assembly file, or add the assembly as a master part to a non-master drawing file. Once the drawing is created, you can use the drafting tools to create and annotate views on separate drawing sheets.

Parts lists are derived directly from the components in the assembly, regardless of how the drawing is created (in the master part or in a non-master part). Dimensions, labels, symbols, and other drafting aids are fully associative with the geometry in the component to which they are attached. Therefore, the assembly drawing updates whenever the referenced components are modified. Similarly, view-dependent modifications made at the drawing level are retained when the component geometry is edited.

Note All view dependent edits are made in the top-level work part. This is the non-master drawing file if you are using the master model strategy. Likewise, any 2D geometry added to a drawing sheet or drawing view is created in the top-level work part.

Dimensions, drafting notes, and drafting labels that are present in a component part are not visible in the assembly where the component part is used. If you wish to display component-level annotation in the assembly, consider using PMI.

Effects of assembly load options on drawings

When you open an assembly part, NX uses assembly load options to determine how to find and load component parts that are referenced by that assembly. The amount of data loaded into your computer's working memory is determined by your **Assembly Load Options**.

When you create a drawing view, only the currently loaded geometry is displayed in the view. However, if you fully load the part, the additional geometry is added to the drawing view once you update it.

Loading structure only

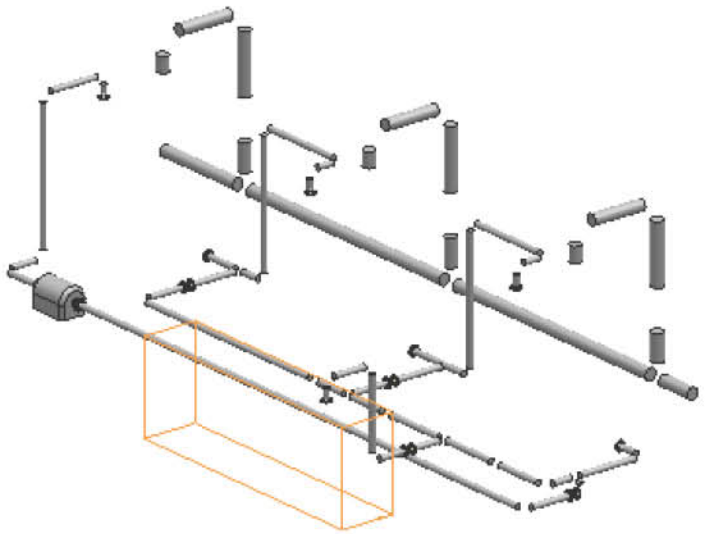
When you select **Structure Only**, only the assembly structure is opened and no components are loaded. Any existing silhouettes, extracted edges, or faceted representations remain in your drafting views until you update the drawing sheet.

When you are ready to work in a view, you can load only the components or subassemblies required for your work. If you hover over the node of an unloaded component in the **Assembly Navigator**, a geometry box appears in the view indicating the component's location. To load a component, right-click the node of an unloaded component node and choose one of the following:

Note Annotations and dimensions associated to unloaded data, such as attributes, will appear retained until you load the data, and a question mark ? is displayed next to drawing view nodes in the **di**.

- **Open→Component**
- **Open→Assembly**

You may find it easier to load components in a model view instead of a drafting view because a geometry box appears in the location of the component whose node you hover over in the **Assembly Navigator**.



Note If you set the **Load Structure Only** check box in the **Open** dialog box, the **Load** option in the **Assembly Load Options** dialog box is automatically set to **Structure Only** in the same NX session.

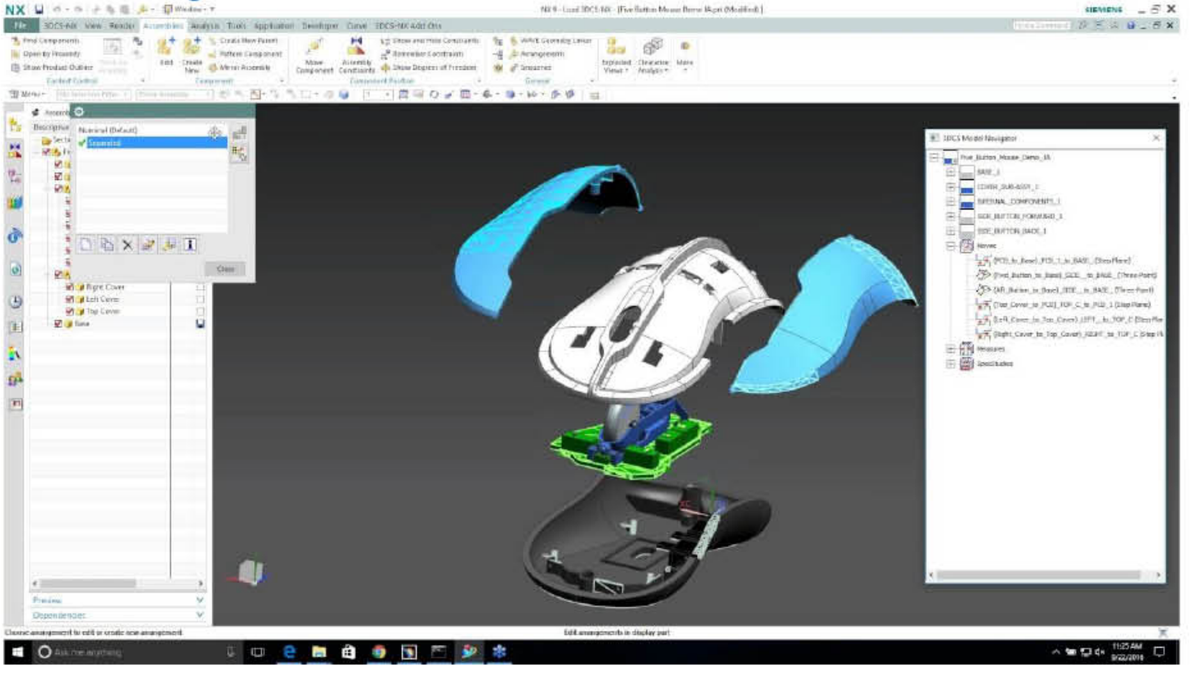
Partially loaded

When you partially load a part or an assembly, only the geometry from the active reference set is loaded. You may want to use partial loading when you need to display certain components to complete your task, but your assembly is large enough to reduce your computer's performance when all the data is loaded in the memory. You can save 20 to 30 percent of your computer's memory when you partially load components.

When you partially load a part or an assembly, you can still annotate geometry in the drafting view, but you must fully load the assembly before you can completely update the view.

1.4.6 Checking drawings and printing drawings

1.5. Assembling sub-assemblies



1.5.1 Add Component

Use the **Add Component** command to add one or more component parts to the work part.

If you add a part family template part, the **Select Family Member** dialog box appears.

You can:

- Add one or more instances of selected components in a single operation.

  Note When you add multiple components at the same time, you may want to use the **Scatter Components** option to prevent the components from being positioned in the same location.

- Select multiple parts to add in a single operation.
- Add the same component with different settings by selecting **Keep Selected** from the **Part To Place** group to keep selected parts selected.

You can also add a component by dragging it to the XC-YC plane. To do this, enable the **Position on XC-YC Plane** customer default.

Tip To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default**.

Application	Assemblies
Command Finder	**Add Component**

1.5.1.1 Add components to an assembly

1. Choose **Assemblies** tab→**Component** group→**Add**.
2. Select one or more parts to add to the assembly from any of the following:
   - The graphics window
   - The **Assembly Navigator**
   - In the **Part To Place** group, from the **Loaded Parts** list box or the **Recent Parts** list box
   - The **Part Name** dialog box that opens when you click **Open**

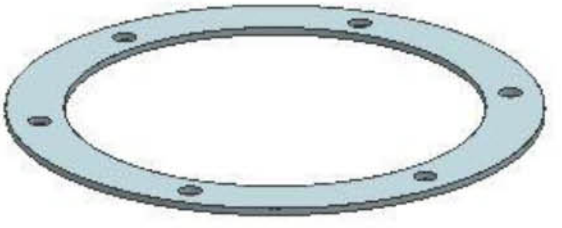
     Note If you type the name in the **File Name** box, instead of selecting it from the list box, you must include the part name and its extension, such as .prt.
3. The components to be added are displayed in the graphics window.
4. In the **Part To Place** group, do the following:
   - (Optional) To keep the selected parts selected after selecting **Apply**, select the **Keep Selected** check box.
   - In the **Count** box, type the number of instances to be added.
5. In the **Location** group, from the **Assembly Location** list, select a positioning option.

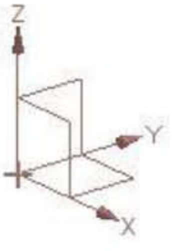
   If you selected **Snap**, specify a location.
6. From the **Placement** group, click one of the following:
   - **Move** to place the component using movement functionalty
   - **Constrain** to place the component with assembly constraints
7. In the **Settings** group, do the following:
   - (Optional) select the **Enable Preview Window** check box.
   - (Optional) In the **Settings** group, from the **Reference Set** list, select the reference set to use for the added components.
   - From the **Layer Option** list, select an option to define where components should be located.
8. Click **OK** or **Apply**.

1.5.1.2 Create a custom anchor point

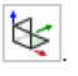
This example shows how to create a custom anchor point for a component, so you can more easily place the component in situations where the absolute origin of the component is not an acceptable anchor point.

1. Open the component.



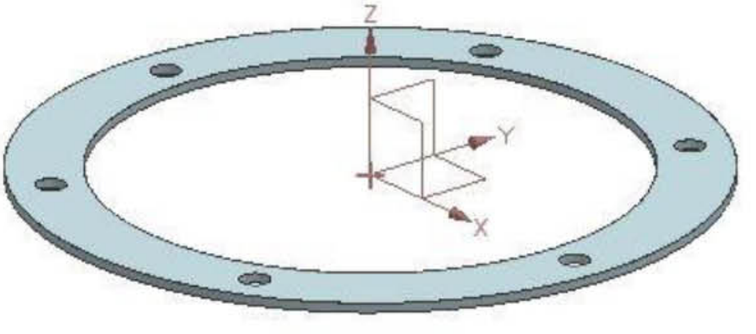


In this example, the part was created in an assembly context by WAVE-linking a face and extruding it. As a result, the absolute origin of the part is at the absolute origin of the assembly it was created in, which is not an acceptable anchor point for placing the part in a different assembly. We need to create a new anchor point.

2. Choose **Home** tab→**Feature** group→**Datum Plane** Drop-down list→**Datum CSYS** .

   You will use a datum CSYS and use it to define the anchor location and orientation.

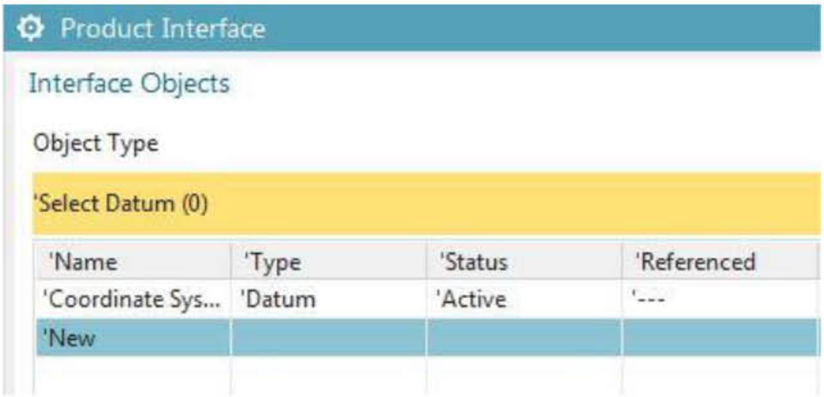
3. Create a datum CSYS at the new anchor location.



4. To define the CSYS as an anchor, select **Assemblies** tab→**General** group→**Product Interface** .

5. From the **Interface Objects** group, from the **Object Type** list, choose  **Datum**.

6. Select the datum CSYS you created.

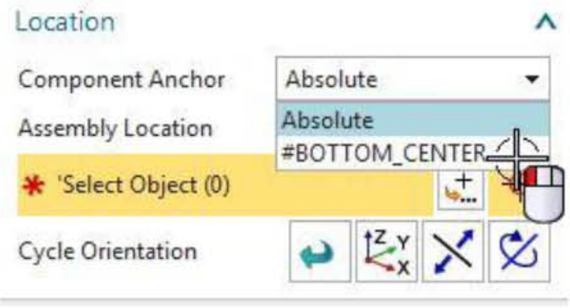


The datum CSYS is added to the list in the dialog box. However, in the **Add Component** dialog box, the name of this anchor is **Coordinate System**, which does not provide useful information about the anchor point. You need to rename the anchor point.

7. Right-click the datum CSYS and choose **Properties**.
8. On the **General** tab, in the **Name** box, type a new name for the coordinate system.
9. Click **OK** twice.

   The new anchor point is defined. You will add the component into an assembly to finish the process.

10. Save the component.
11. Open the assembly.
12. Choose **Home** tab→**Assemblies** group→ **Add** .
13. In the **Part To Place** group, from the **Loaded Parts** list box, select the component you want to add to the assembly.
14. In the **Location** group, from the **Component Anchor** list, select the anchor you created earlier.



15. Place the component in your assembly.

The new anchor point is saved with the component, so you and downstream users can easily locate this component to move or apply constraints to.

1.5.1.3 Add a part family member to an assembly

This example shows how to add a single instance of a part family member to an assembly, and position the component at a selected origin.

1. Choose **Assemblies** tab→**Component** group→**Add** .
2. In the **Part To Place** group, click **Open**  and select a part family template part.
3. In the **Choose Family Member** dialog box, from the **Matching Members** list, select a member part.

   Tip Even if you know the part name, you may want to set **Criteria** so you can use the reporting and update mechanisms at a later time. This can help capture your design intent.

4. If you want to set **Criteria** to filter for and select the member of the part family, do the following:
   a. Select an attribute in the **Family Attributes** list box.

      The attribute name appears in the **Criteria** box, and the possible values appear in the **Valid Values** list box.

   b. (Optional) Type an expression or modify the expression in the **Criteria** box.

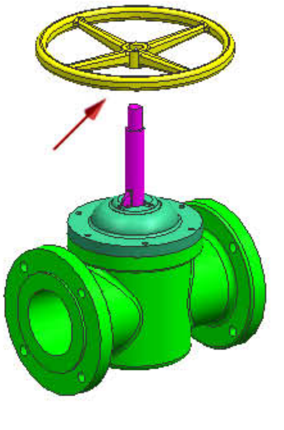
      Tip You can click **Clear All Criteria** any time you want to remove and redefine all the criteria.

   c. From the **Valid Values** list box, select a value that meets the requirements for the assembly.

      The **Valid Values** list includes all values for the selected family attribute.

      Example If “length” has valid values of **8** and **12**, you can select **8**. Now, **12** moves to the **Not Valid Values** list box. The **Matching Members** now lists only family members whose length attribute is 8.

   d. Repeat this step until you define all necessary selection criteria.
5. In the **Matching Members** list box, select one of the remaining parts, and then click **OK**.
6. In the **Placement** group, click **Move** or **Constrain**.
7. In the **Settings** group, do the following:
   - From the **Reference Set** list, select a reference set type.
   - From the **Layer Option** list, select **Work**.
8. Click **OK** to place the part family member.



1.5.1.4 Create New Component

Use the **Create New Component** command to create a new component in the assembly by selecting geometry and saving it as a component.

You can use **Create New Component** to create an assembly using a top-down design method.

With the top-down method, you can:

- Copy or move existing geometry into a new component.
- Create an empty component file and add geometry to it later.

NX creates a new component file that contains the selected geometry and also creates a component in the assembly file.

Note You must save the new component. It is not saved automatically.

Application	Assemblies
Command Finder	**Create New Component** 

1.5.1.5 Reflect Component

When you want both left-hand and right-hand versions of a component, you can add the component to the assembly and then reflect the component to create the other version. The reflected component is a new instance of the original component; a new part file is not created. You can also reflect subassemblies or multiple components.

Editing a reflected component

The reflected and non-reflected component instances are referencing the same part file, so the geometry in both is always the same. Because the reflected instance has a reflection transform applied to it, all the geometry in it appears reflected. Editing the geometry in the original component results in the edits appearing in both the reflected and non-reflected component instances.

Because you cannot make a reflected component instance the work part, you cannot directly edit the geometry in the reflected component instance.

Note Use the **Mirror Assembly** command instead of the **Reflect Component** command if you need left-hand and right-hand versions that don't match. Mirroring a component produces a new part file that you can edit during or after creation.

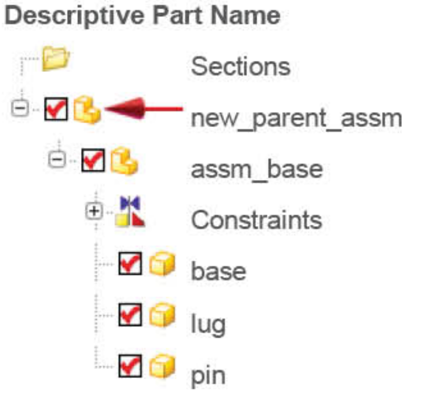
Preparing reflected components for downstream processes

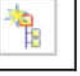
Because a reflected component is an instance of an existing component, you must ensure that downstream processes, including manufacturing, recognize which instances are reflected and which instances are not reflected. For example, you can assign a unique component name to a reflected component in the **Assembly Navigator** properties, if your company's processes are capable of distinguishing between reflected and nonreflected instances based on component names.

Application	Assemblies
Command Finder	**Reflect Component** 

1.5.1.6 Create New Parent

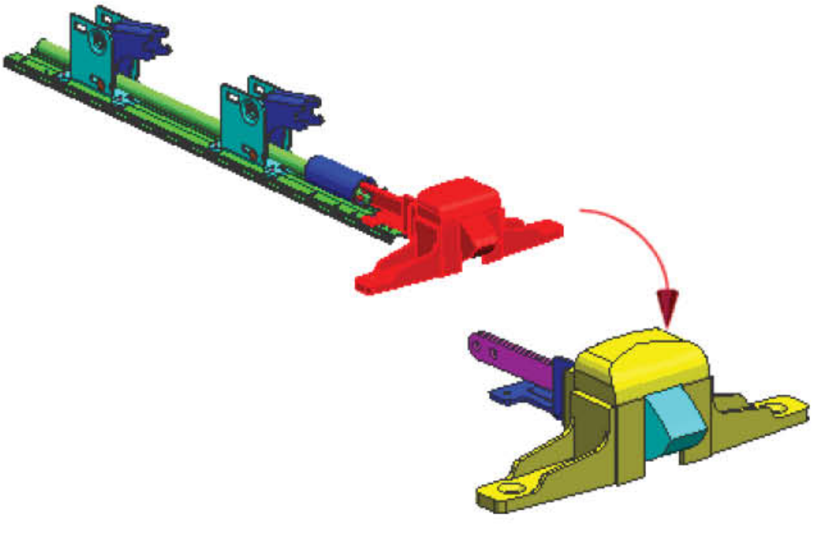
Use the **Create New Parent** command to create a new parent part for your current active part. During the operation, an empty assembly is created, and the current active part is added to it as a child. When the operation is complete, the new parent assembly becomes the active and work part.



Application	Assemblies
Command Finder	**Create New Parent** 

1.5.1.7 Create a new parent assembly

This example shows how to create a new parent part file for a selected component.



1. (Optional) In the **Assembly Navigator**, ensure that the current active part is the part for which you want to create a new parent part file.

2. Choose **Assemblies** tab→**Component** group→**Create New Parent** .

3. In the **Create New Parent** dialog box, from the **Templates** list, select a template.
4. In the **New File Name** group, in the **Name** box, specify a unique name.

Tip
Click  to select a part that has a similar name and modify the name for a new parent part.

5. In the **Folder** box, specify the directory for the parent part file.

Tip
Click  to navigate to the directory for the new parent part file.

6. Click **OK**.

The **Assembly Navigator** lists the parent part file. The new parent part file becomes the work part.

Component

Use the **Replace Component** command to remove an existing component and replace it with another component that is a *.prt file type.

Note You will be alerted that the component to be replaced is not a revision of the replacement part, if the new component is:

- Not created from the same template part as the original.
- Not a descendent of the same original blank part if no template was used.

Application	Assemblies
Command Finder	**Replace Component** 

1.5.1.8 Make Unique

Use the **Make Unique** command to create a new part file for one or more selected occurrences of the same part.

For example, the following figure shows four selected occurrences of a part named **GKballjoint**. You can use the **Make Unique** command to convert the two occurrences near the tires to reference a new unique part file named **GKballjoint_tire**. The other two occurrences, which are attached to the steering column subassembly, continue to reference the **GKballjoint** part file.

When you create a unique part file, you can modify the occurrences that reference the new part without affecting occurrences that still reference the original part. For example, you can modify the **GKballjoint_tire** part file so its occurrences fit larger tire subassemblies without affecting the **GKballjoint** ball joints that are attached to the steering column subassembly.

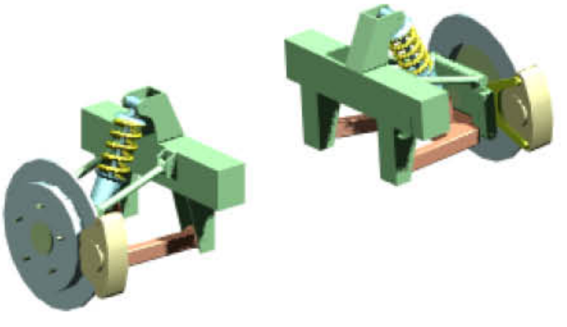
Application	Assemblies
Command Finder	**Make Unique**

1.5.1.9 Mirror Assembly

Use the **Mirror Assembly** command to:

- Create associative or nonassociative mirrored components in an assembly.
- Position new instances of the same parts at mirror locations.
- Create new parts that contain linked mirror geometry.

Many assemblies represent one side of a fairly symmetric larger assembly. You can create one side of your assembly and create a mirrored version to form the other side of your assembly.



You can mirror an entire assembly, or you can select individual components to be mirrored. You can also specify components to be excluded from the mirrored assembly. Each mirrored body is added to the reference set of its source body.

You can create a mirrored component in one of the following ways:

- Create mirror geometry. NX creates a new part file that contains the linked mirror body features for all the solid geometry in the original component. It also adds the new mirror part to the assembly as a component.
- Create a new instance of the selected part or subassembly, and reposition it across the mirror plane.

You can specify the plane of symmetry that is used to mirror the component, using one of six **Reuse and Reposition** options to define the plane of symmetry. Each positioning option is based on one of the following:

- The inertial axes of the geometry in the part or subassembly.
- The WCS of the part or subassembly.
- A user defined plane of symmetry, which is useful when the part is not truly symmetric. For more information, see *User defined plane of symmetry for mirrored assemblies*.

Before you create the mirrored assembly, you can preview it and make corrections using the **Mirror Review** step.

Application	**Assemblies**
Prerequisite	An assembly must be the work part.

Command Finder	Mirror Assembly 

Mirror an Assembly

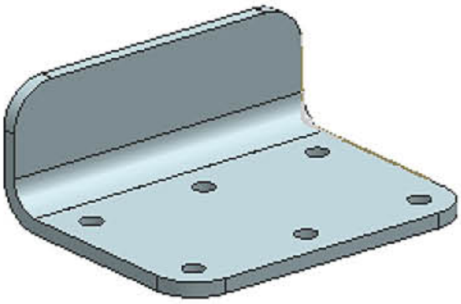
1. Choose **Assemblies** tab→**Component** group→**Mirror Assembly** .

   The **Mirror Assemblies Wizard** welcome page opens.

2. Click **Next**.

   The **Select Components** page is displayed.

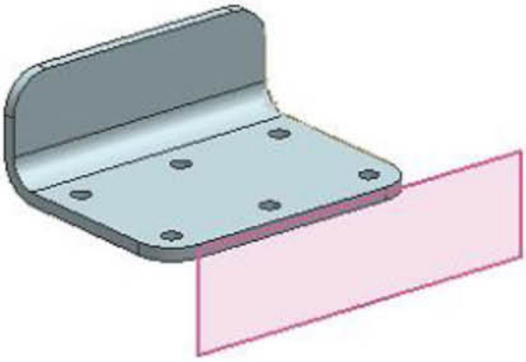
3. Select the components to mirror and click **Next**.



   The components must be children of the work assembly.

   The **Select Plane** page is displayed.

4. Select an existing plane or click **Create Datum Plane** , and create a datum plane.



5. Click **Next**.

   The **Naming Policy** page is displayed. If no component uses the **Mirror Geometry** type, you can click **Finish**. Otherwise, proceed to the next step.

6. Specify the **Naming Rule** that you want to use for these opposite-side parts, and specify a **Directory Rule** for the parts.

   You can add a prefix or suffix to the source part names, or replace the string in the original name.

7. Click **Next**.

   The **Mirror Setup** page is displayed. Selected components are listed in the right-hand panel.

8. (Optional) Select a different mirror type for each component. Select a component from the list of components in the right panel and click any of the following:

   - Click **Associative Mirror**  to create an associative opposite-side version of a component and create new parts.

- Click **Non Associative Mirror** to create an nonassociative opposite-side version of a component and create new parts.
- Click **Exclude** to exclude the selected component.

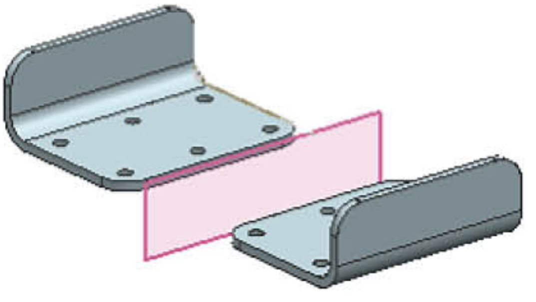
Reuse and Reposition is the default mirror type.

9. Click **Next**.

   The mirrored components appear in the graphics window. The **Mirror Review** page is displayed.

10. (Optional) Make corrections before finishing the operation. You can do any of the following:

    - Click **Cycle Reposition Solutions** to cycle through the options for reuse and reposition.
    - Change a component's mirror type from **Reuse and Reposition** to **Associative Mirror** or **Non Associative Mirror**.

    - Select components and click **Exclude**.

11. Click **Next** or **Finish**.

    The **Name New Part Files** page is displayed.

    To change the name, folder, or attributes of the new mirrored parts, do the following:

    - Click **Use the button to name the mirrored parts and set attributes** to open the **Name Parts** and review the names that the wizard applied to your new opposite-side parts in the **Parts to Rename** list.
    - Select a name in the **Name** column to change it.
    - In the **Name and Location** group, in the **Name** box, specify a new unique name.

      Note In a Teamcenter environment, double-click the object in the **Object Name** column, and change the name in the **Edit** dialog box.

    - In the **Folder** box, specify a new folder.
    - Click **OK**.

    Note In a Teamcenter environment, you can also assign the following values:

    Secondary Attributes

    Alternate Ids

    Projects

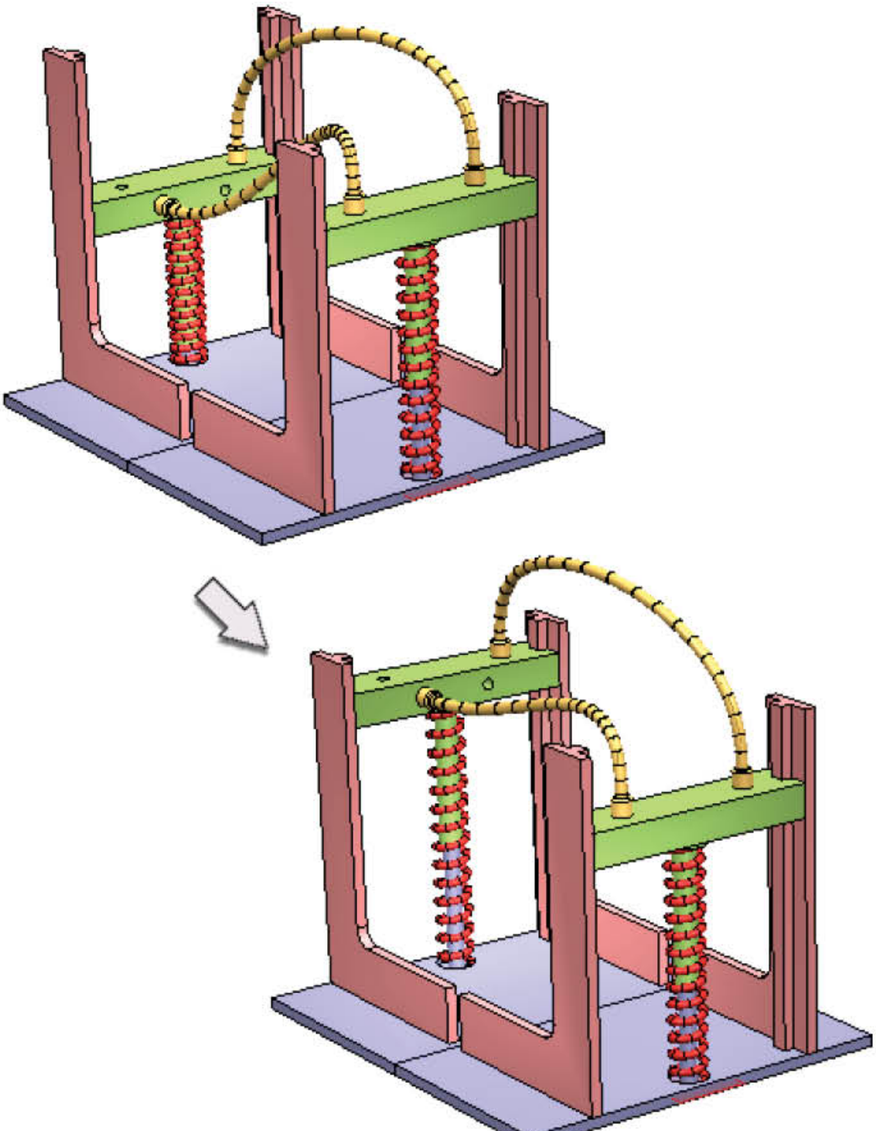
 **Folder**

12. Click **Finish**.

Note In a Teamcenter environment, the attributes must be fully specified, as indicated in the **Fully Specified** column ✔.

1.5.1.10 Deformable parts

You can define a part as capable of assuming more than one shape when it is added to an assembly. This is especially useful for parts such as springs or hoses, which often have different shapes in the same assembly. Deformable components are also known as flexible components.

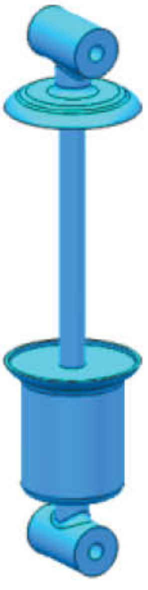


To use a deformable component in an assembly, you must:

- Define the deformable component. You can do this either before or after you add it to the assembly.
- Add the deformable component to the assembly.
- Edit the deformable component in the assembly to the desired shape.

Add a deformable part to an assembly

1. Make sure your assembly is the work part.



2. Choose **Assemblies** tab→**Component** group→**Add** .
3. Select a deformable part and position it somewhere close to the assembly.

   In this example, the spring is a deformable component.

   The deformation dialog box appears. The name of this dialog box is the name of the deformation.

4. In the deformation dialog box, modify the deformation input parameters, and click **OK**.

   If you do not want to deform the part at this time, click **Cancel**.

5. Constrain the deformable component to the assembly as shown.

6. Save the assembly.

Create an Assembly Arrangement
Use this procedure to create assembly arrangements.

1. In the **Assembly Navigator**, select an assembly or subassembly.

2. Choose **Assemblies** tab→**General** group→**Arrangements** .
3. From the list box, select an arrangement and click **Copy** .
4. Rename the copied arrangement.

   You can accept the new default name, rename it immediately, or rename it at a later time with the **Rename** option .
5. Move the components to the new positions within the assembly and save your parts.

   Check for arrangement-specific options in either the **Move** or **Suppression** dialog boxes.

   If you have two or more arrangements, you can switch arrangements by selecting the one that you want from the **Assembly Navigator**→**Arrangements** shortcut menu.

If you want to change your arrangement to the one you created, right-click the assembly or subassembly where you created it and choose the new arrangement.

Note The arrangement is saved when you save your parts.

If you want to make the new **Arrangement** the default, click **Set as Default** .

1.5.1.11 Move Component

Use the **Move Component** command to move and optionally copy one or more components in an assembly. You can move components dynamically, or you can create temporary constraints to move the components into position.

When you move components, the **Modified** column in the **Assembly Navigator** displays a **Modified** icon in the rows of the parts that are affected.

Note Your assembly may have some constraints that were created by NX as needed to complete earlier operations. This can cause parts that you thought were not affected by the move to be marked as modified. For example, NX may create hidden constraints when you add routing geometry to the assembly. These constraints do not appear in the graphics window, the **Assembly Navigator**, or the **Constraint Navigator**.

Moving components that are not immediate children

By default, you can only move components that are immediate children of the work part.

However, you can set a customer default to let you move components anywhere in your assembly when you are in the **Move Component** dialog box, regardless of what your work part is. In the **Customer Defaults** dialog box, under **Move Component**, set **Scope** to **Anywhere in Assembly**.

Tip To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .

This is useful, for example, if you want to simultaneously move a collection of components that do not share the same assembly parent.

Caution The use of the **Anywhere in Assembly** customer default can significantly reduce the performance of the **Move Component** command when the assembly structure of the displayed part has a large number of assembly constraints.

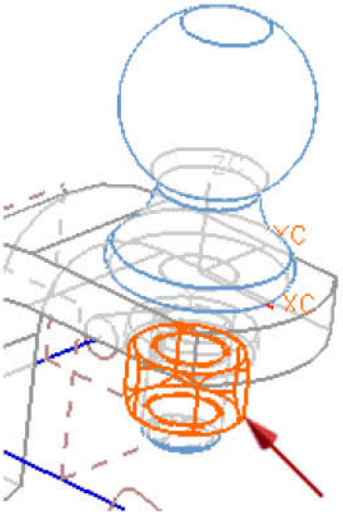
Application	Assemblies
Command Finder	**Move Component**

1.5.1.12 Move a component

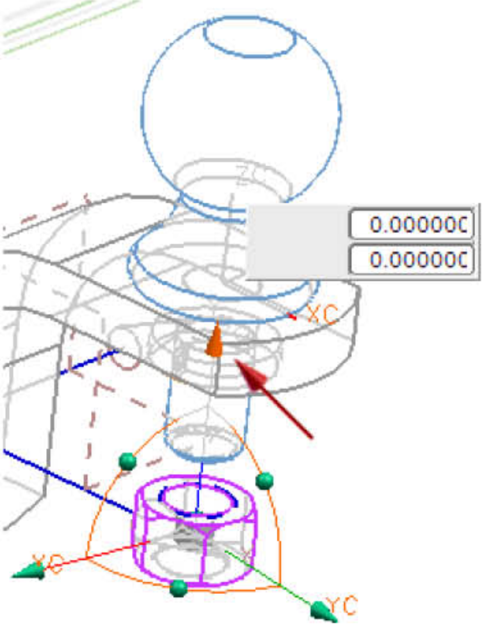
Move a component dynamically

This example shows how to move a component dynamically using handles and without collision detection.

1. Choose **Assemblies** tab→**Component Position** group→**Move Component** .
2. In the **Move Component** dialog box, in the **Components to Move** group, ensure that **Select Components**  is active.
3. Select one or more components to move.



4. In the **Transform** group, from the **Motion** list, select **Dynamic**.
5. In the **Copy** group, from the **Mode** list, ensure that **No Copy** is selected.
6. In the **Collision Detection** group, from the **Collision Action** list, select **None**.
7. In the **Transform** group, ensure **Specify Orientation** is active.
8. Drag the handles to move the components to a new location.



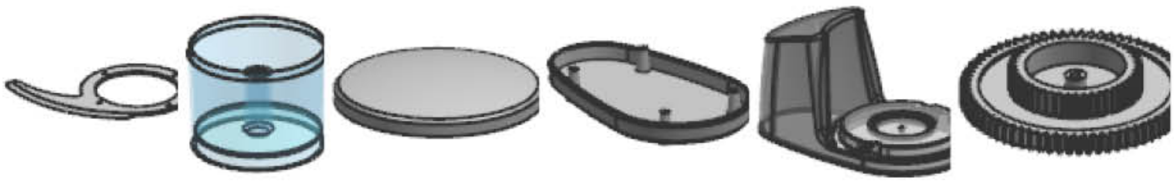
1.5.2 Constrain an assembly

1.5.1.1 Assembly constraint types

Assembly constraint type	Description
Align/Lock	Aligns two axes in different objects, while simultaneously preventing any rotation about the common axis. It is commonly used as one of the constraints when you want to fully constrain a bolt in a hole.
Angle	Specifies an angle between two objects, optionally around a specified axis.
Bond	Constrains objects together so they move as a rigid body. Note Bond constraints can only be applied to components, or to components and assembly-level geometry. Other objects are not selectable.
Center	Centers one or two objects between a pair of objects, or centers a pair of objects along another object.
Concentric	Constrains two circular or elliptical edges so the centers are coincident and the planes of the edges are coplanar.
Distance	Specifies the 3D distance between two objects. Note If a distance constraint is created between two edges, or two points, or an edge and a point, positive and negative values are treated as identical. NX does not recognize negative signs for these cases because it can move these geometries continuously from one side to the other while solving the constraint. For these cases, values are sign-independent in limits on driven distance constraints as well as in distance constraint values. For other types of geometry such as faces, negative values are recognized.
Fit	Constrains two objects with equal radii, such as circular or elliptical edges, or cylindrical or spherical faces. The linear tolerance is 0.1 mm for cylindrical faces, and the angular tolerance is 1 degree for conical faces. If the radii later become non-equal, the constraint is invalid. A fit constraint is useful for locating pins or bolts in holes. Note Fit constraints created during the releases between NX 5.0 and NX 6.0.1 are listed as **Legacy Fit** constraints. Fit constraints created in NX 6.0.2 or later releases are listed as **Fit** constraints. Legacy fit and fit constraints differ only in their behavior with cones or tori. A legacy fit constraint makes the axes of two cones or tori coincident. A fit constraint makes two cones or tori fully coincident.

Fix	Fixes an object at its current position. Note A fix constraint is useful when you need an implied stationary object. With no fixed node, the entire assembly has freedom to move.
Parallel	Defines the direction vectors of two objects as parallel to each other.
Perpendicular	Defines the direction vectors of two objects as perpendicular to each other.
Touch Align	Constrains two components so they touch or align with each other. Note **Touch Align** is the most commonly-used constraint. You can specify whether you want a touch constraint or an align constraint, or you can use the **Prefer Touch** orientation to let NX calculate the constraint type.

This example shows how to create and constrain an assembly by using the following components to assemble a food chopper.



chopper_blade	chopper_bowl	chopper_bowl_cover	chopper_case_base	chopper_case_top	chopper_gear1

chopper_gear2	chopper_motor	chopper_rotor	chopper_spindle	chopper_support_plate

As you add each component to an assembly, consider which constraints it needs in order to be positioned correctly in the assembly. Not all components should be fully constrained. For example, some components such as gears need to rotate, and other components may need to slide in one or more directions.

Note In the example, the assembly is created by adding existing components and constraining each component when it is added. Another common workflow is to create each component in context of the assembly and to constrain it at that time.

1. Create an assembly part file for the chopper.
   a. Choose **File** tab→**New**.
   b. In the **New** dialog box, on the **Model** tab, in the **Templates** group, select the **Assembly** row.

c. In the **New File Name** group, define the following.

- **Name** = a name for your assembly. In this example, **Name** = **chopper_assy**.
- **Folder** = the directory where your parts for the assembly are located.

d. Click **OK**.

The **Add Component** dialog box is displayed, and the assembly is displayed as the top-level node of the assembly structure in the **Assembly Navigator**.

For more information, see *Add Component*.

2. Add the first component.

a. In the **Part** group, select or open the first component.

Note You can assemble components in any order that is convenient during the design phase, and create alternate orders later. For more information, see *Ordering components in the Assembly Navigator*.

b. The first component is displayed in the **Component Preview** window if your **Preview** check box is selected.

c.

d. In the **Assembly Navigator**, the first component is displayed as a component node under the assembly node, and the current order is displayed at the end of the assembly node.

chopper_assy (Order: Chronological)
chopper_case_base

e. In the **Placement** group, from the **Positioning** list, select **Absolute Origin**.

f. Click **OK**.

3. Constrain the first component in one of the following ways.

- Constrain the component with a fix constraint.
- Fully constrain the component to assembly-level geometry, such as a datum CSYS.

In this example, the first component is constrained by a fix constraint.

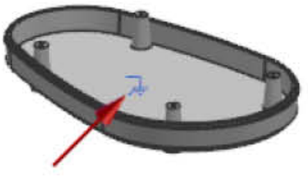
c. Choose **Assemblies** tab→**Component Position** group→**Assembly Constraints** .

d. In the **Assembly Constraints** dialog box, from the **Type** list, select **Fix**.

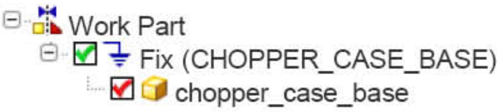
e. In the **Settings** group, ensure that the **Associative** check box is selected.

f. In the graphics window, select the first component.

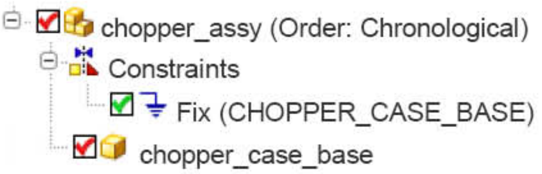
The fix constraint is displayed in the graphics window.



You can use the **Constraint Navigator** to help you analyze, organize, and work on assembly constraints. You can specify a **Constraint Navigator** grouping mode to specify how the information is organized in the navigator. This example uses the **Group by Constraints** mode, which lists constraints under the top-level **Work Part** node. You can expand a constraint to get more information about the components that are involved in the constraint.



You can also see some information about constraints in the **Assembly Navigator**, although the **Assembly Navigator** is designed primarily to help you explore components and the assembly structure. In the **Assembly Navigator**, a **Constraints** node is displayed under the assembly node. You can expand the **Constraints** node to see the fix constraint.



g. Click **OK**.

4. Add and constrain a second component.

. Choose **Assemblies** tab→**Component** group→**Add** .

a. In the **Add Component** dialog box, in the **Part** group, select or open a component.

b. In the **Placement** group, from the **Positioning** list, select **By Constraints**.

c. Click **Apply**.

The **Assembly Constraints** dialog box is displayed.

Tip In the **Preview** group, you can select the **Preview Component in Main Window** check box to preview how your current constraints position the component in your assembly.

d. Constrain the second component according to your design intent.

In the example, the chopper_case_top is constrained to the chopper_case_base by two touch constraints. For both constraints:

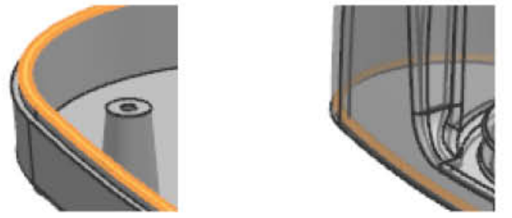
- **Type = Touch Align**
- **Orientation = Prefer Touch**

Each constraint is created after you select one object in the base and one object in the top. You do not need to click **OK** or **Apply** until you finish creating both constraints.

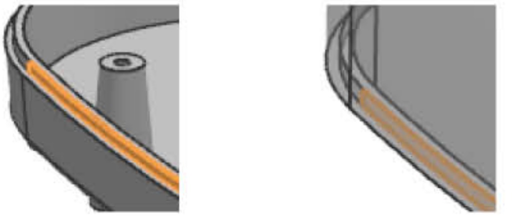
Note A lightning bolt symbol may be temporarily displayed beside the constraint in the graphics window while NX calculates whether the constraint type is touch or align. See *Assembly constraint notes* for more information.

Select geometry in the base Select geometry in the top

First touch constraint

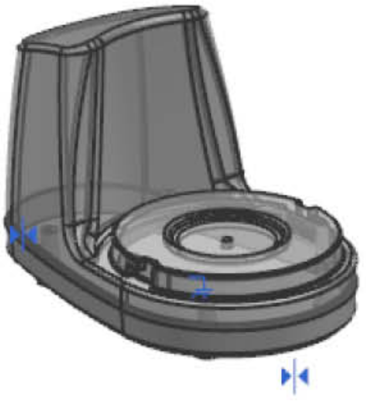


Second touch constraint



e. Click **OK**.

NX adds the second component to the assembly and positions the components according to the constraints.



In the **Constraint Navigator**, the new constraints are displayed under the **Work Part** node.

- Work Part
  - Fix (CHOPPER_CASE_BASE)
    - chopper_case_base
  - Touch (CHOPPER_CASE_BASE, CHOPPER_CASE_TOP)
  - Touch (CHOPPER_CASE_BASE, CHOPPER_CASE_TOP)

In the **Assembly Navigator**, the new constraints are displayed under the **Constraints** node.

- chopper_assy (Order: Chronological)
  - Constraints
    - Fix (CHOPPER_CASE_BASE)
    - Touch (CHOPPER_CASE_BASE, CHOPPER_CASE_TOP)
    - Touch (CHOPPER_CASE_BASE, CHOPPER_CASE_TOP)
  - chopper_case_base
  - chopper_case_top

5. Continue adding and constraining components until the assembly is complete.

   . In the example, the chopper_support_plate component is constrained with one touch constraint and one align/lock constraint.

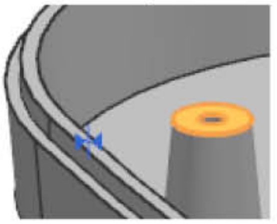
   For the touch constraint:

   - **Type = Touch Align**

- **Orientation = Prefer Touch**

Select geometry in the base Select geometry in the support plate

Touch constraint

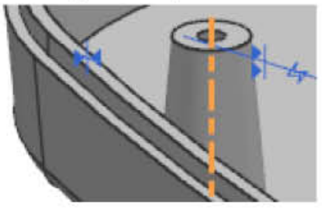


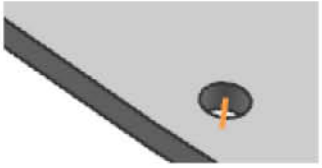
For the align/lock constraint:

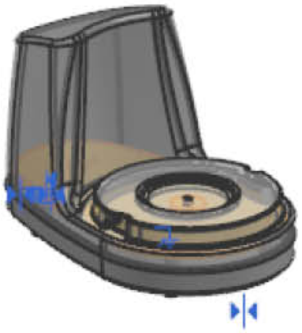
- **Type = Align/Lock**

Select geometry in the base Select geometry in the support plate

Align/lock constraint







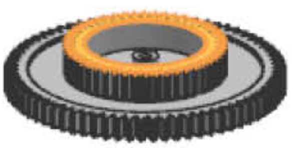
a. In the assembly, the chopper_gear1 component is constrained to the chopper_support_plate component using a touch constraint and an align constraint.

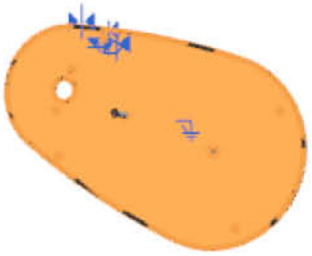
For the touch constraint:

- **Type = Touch Align**
- **Orientation = Prefer Touch**

Select geometry in gear1 Select geometry in the support plate

Touch constraint

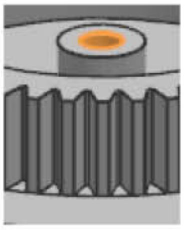
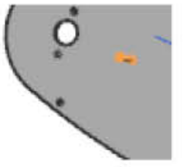




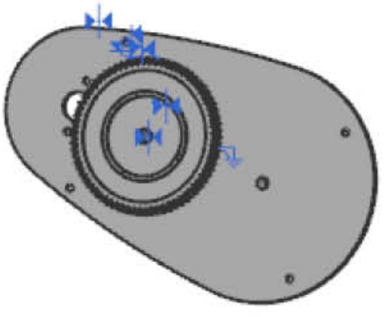
For the align constraint:

- **Type = Touch Align**
- **Orientation = Infer Center/Axis**

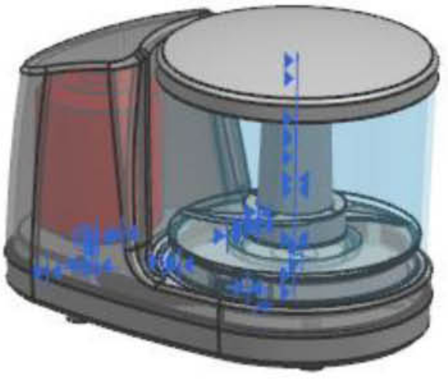
Note When you use the **Infer Center/Axis** orientation, you can select the cylindrical face or edge whose axis you want to align.

	Select geometry in gear1	Select geometry in the support plate
Align constraint		

Tip You can show and hide components as necessary to help you add and constrain new components.



The following figure shows the completed food chopper.



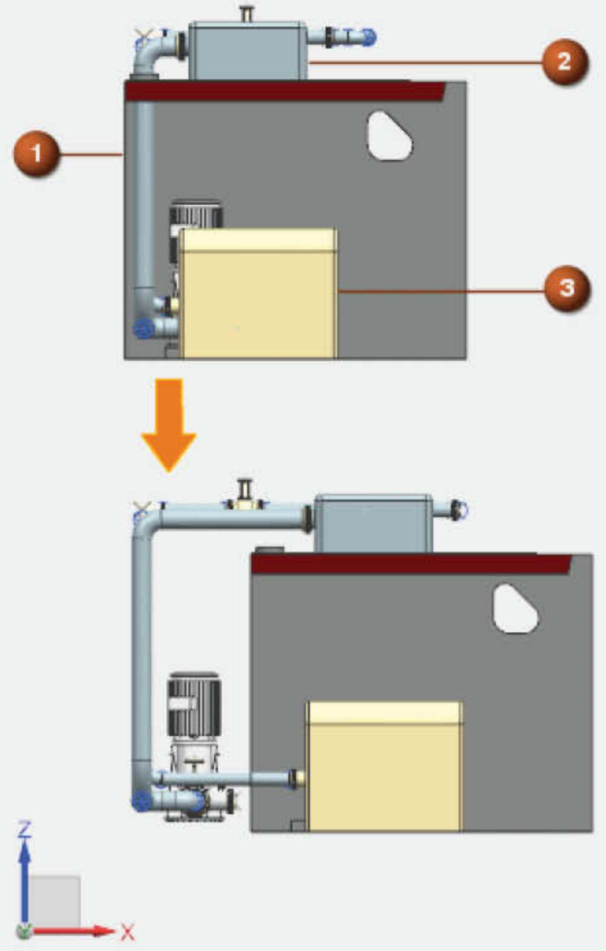
1.5.1.2 Assembly Constraints dialog box

Constraint Type	
These options also appear in the **Transform** group of the **Move Component** dialog box when **Motion** is set to **By Constraints**.	
List	Lets you specify the type of assembly constraint by selecting its icon. For more information, see *Assembly constraint types*.
Geometry to Constrain	
These options also appear in the **Constraints** section of the **Move Component** dialog box.	
Orientation	Appears only when **Type** is **Touch Align**. Lets you influence possible solutions for a **Touch Align** constraint as follows:

	• **Prefer Touch** — Presents a touch constraint when touch and align solutions are both possible. (Touch constraints are more common than align constraints in most models.) **Prefer Touch** presents an align constraint when a touch constraint would over-constrain the assembly. • **Touch** — Constrains objects so their surface normals are in opposite directions. • **Align** — Constrains objects so their surface normals are in the same direction. • **Infer Center/Axis** — Specifies that when you select a cylindrical, conical, or spherical face, or a circular edge, NX uses the object's center or axis itself for the constraint. Note When more than one solution is possible, you can flip through the solutions.
Subtype	Appears only when **Type** is **Angle** or **Center**. (**Angle** only) Specifies whether the angle constraint is: • **3D Angle** — Measured between two objects without requiring a defined axis of rotation. Note The value of a 3D angle constraint is 180 degrees or less. Because a 3D angle constraint defines the angle in space between two objects without using a defined axis, the minimum angle between the objects is used to solve the constraint. • **Orient Angle** — Measures the angle constraint between two objects, using a selected axis of rotation. It can support up to 360 degrees of rotation. (**Center** only) Specifies whether the center constraint is: • **1 to 2** — Centers one object between a pair of objects. • **2 to 1** — Centers a pair of objects along another object. • **2 to 2** — Centers two objects between a pair of objects.
Port	Appears when you open a Routing assembly and when the **Type** list is set to **Angle**. This option specifies the constraints with respect to the connection ports. •  **Align** specifies a parallel constraint between the alignment vectors of the two ports. •  **Rotate** specifies a parallel constraint between the rotation vectors of the two ports.
Axial Geometry	Appears only when **Type** is **Center** and **Subtype** is **1 to 2** or **2 to 1**. Specifies what NX uses for the center constraint when you select a face (cylindrical, conical, or spherical), or a circular edge:

	• **Use Geometry** — Uses the face (cylindrical, conical, or spherical) or edge for the constraint. • **Infer Center/Axis** — Uses the center or axis of the object.
Select Object	Lets you select the objects for the constraint. This option's name may vary slightly to tell you how many objects to select. For example, this option is sometimes named **Select Two Objects**, depending on the settings of other options in the **Assembly Constraints** dialog box.
Point Constructor	Appears only when **Type** is **Center**, **Fit**, **Touch Align**, or **Distance**. Opens the **Point Constructor** to let you define a point for the constraint.
Create Constraint	Appears only when **Type** is **Bond**. Constrains selected objects together so they must move as a rigid body.
Reverse Last Constraint	Available only when there are two solutions to a constraint. Shows you the other solution to the constraint.
Cycle Last Constraint	Appears only for distance constraints when there are more than two solutions. Lets you cycle through the possible solutions for the distance constraint.
Angle	
Appears after you select objects when **Constraint Type** is set to **Angle**.	
Angle	Displays the angle value between the selected objects and lets you specify the behavior of the angle when the components are moved. If you want the angle you specify to: • Remain constant - Select the **Angle** check box to define the constraint as driving. Other constraints will adjust as necessary. • Clear the **Angle** check box to define the constraint as a driven constraint.
Angle Limits	
Appears after you select objects when **Constraint Type** is set to **Angle**.	
Angle Limits	Lets you specify a range of motion for the components by defining an upper limit, a lower limit, or both for the angle constraint.
Distance	
Appears after you select objects when **Constraint Type** is set to **Distance**.	
Distance	Displays the distance value between the selected objects and lets you specify the behavior of the distance when the components are moved.. If you want the distance value you specify to: • Remain constant - Select the **Distance** check box to define the constraint as driving. Other constraints will adjust as necessary.

	• Clear the **Distance** check box to define the constraint as a driven constraint.
Distance Limits	
Appears after you select objects when **Constraint Type** is set to **Distance**.	
Distance Limits	Lets you specify a range of motion for the components by defining an upper limit, a lower limit, or both for the distance constraint.
Settings	
Arrangements	Specifies how the constraints affect component positioning in other arrangements: • **Use Component Properties** — Specifies that the setting of **Arrangements** on the **Parameters** tab of the **Component Properties** dialog box determines the positions. The **Arrangements** setting can be either **Individually Positioned** or **Same Position in All**. • **Apply to Used** — Specifies that the constraint is applied to currently used arrangement.
Dynamic Positioning	Specifies that NX solves constraints and moves components as you create the constraints. If the **Dynamic Positioning** check box is not selected, NX does not solve constraints or move objects until you click **OK** or **Apply** in the **Assembly Constraints** dialog box.
Associative	Specifies that constraints are added to the assembly when you close the **Assembly Constraints** dialog box. The constraints are saved when you save the component. Constraints created when the **Associative** check box is cleared are transient. They are deleted when you click **OK** to exit the dialog box or when you click **Apply**. Note When you define an assembly constraint, the state of the **Associative** check box determines whether or not the constraint is associative. You can define multiple associative and nonassociative assembly constraints before clicking **OK** or **Apply**.
Move Curves and Routing Objects	Moves routing objects and related curves when they are used in a constraint.
Dynamic Update of Routing Solids	Dynamically updates the routing objects when you move connected components. Example In this assembly, when you move the compartment (1) and substations (2) and (3) along the X-axis, NX automatically updates the routing stock to accommodate the change.

	
Enabled Constraints	Lets you simplify the **Constraint Type** group to include only the types you use. Select or clear the check box beside a constraint type to display or remove it from the group.

Limits on distance and angle constraints

If you want to limit the range of motion for a component positioned by a distance or angle constraint, you can define an upper or lower limit, or both, for the constraint value. Limiting the range of motion is useful, for example, when you want to prevent collisions between components.

You can define limits in any of the following situations:

- When you create a distance or angle constraint using the **Assembly Constraints** command.
- When you edit a distance or angle constraint, for example, when you right-click the constraint in the **Assembly Navigator** or **Constraint Navigator** and choose **Edit**.
- When you move a component by constraints, for example, when you open the **Move Component** dialog box and choose **By Constraints** from the **Motion** list.
- When you edit the constraint in arrangements, for example, by right-clicking the constraint and choosing **Edit in Arrangements**.

Driving constraints and driven constraints

Distance constraints and angle constraints are either driving constraints or driven constraints.

- For a driving constraint, you define a value for the angle or distance. If the constraint has defined limits, you should make sure that the value you specify is between the limits. Otherwise, as designed, the constraint fails.

Note Distance and angle constraints created in previous versions are driving constraints.

- For a driven constraint, the angle or distance varies within defined limits for movement that is driven by other constraints or directly by moving the component using the **Move Component** command.

You can define a distance or angle constraint as driving or driven using either of the following methods:

- In the **Assembly Constraints** dialog box, in the **Distance** group or **Angle** group, you can select or clear the checkbox. The constraint is a driving constraint if the check box is selected . If the checkbox is not selected , the constraint is a driven constraint.

  Note The options also appear in the **Move Component** dialog box when you move components by constraints.

- In the **Assembly Navigator**, you can right-click a distance or angle constraint. If the constraint is a driving constraint, the **Make Driven** command is available. If the constraint is a driven constraint, the **Make Driving** command is available.

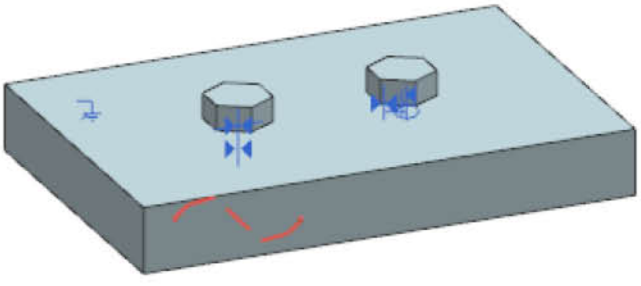
If you want to see which constraints in your assembly are driven constraints, you can open the **Constraint Navigator** and scroll to the **Driven** column. A driven distance constraint has an icon, and a driven angle constraint has an icon. Distance and angle constraints that do not have an icon are driving constraints.

Differences between Align and Align/Lock constraints
Align/Lock constraints

Use an **Align/Lock** constraint type to:

- Align two axes in different components while preventing any rotation about the common axis. The **Align/Lock** constraint behaves like an **Align** constraint except for this lack of rotation.
- Align two circular arcs in different components while preventing the components from rotating relative to each other. The **Align/Lock** constraint behaves like a **Concentric** constraint except for this lack of rotation.

In the following figure, the bolts have the same constraints, except the bolt on the left has an **Align** constraint between the centerlines of the bolt and hole. The bolt on the right has an **Align/Lock** constraint between the same centerlines. The bolt on the left has one rotation degree of freedom, which means that the bolt can spin around its centerline. The bolt on the right is locked in place, and therefore has no degrees of freedom.



Note If you create an **Align/Lock** constraint when the geometry of both components are already aligned, as in the previous figure where the centerlines of the bolt and the hole are already aligned, neither component is rotated when the constraint is applied.

An **Align/Lock** constraint is useful in situations such as the following:

- You want to position and fully constrain rotationally-symmetric components.

- You have a component that you want to constrain using an axis, and you want to simultaneously prevent rotation about that axis. For example, you often use an **Align/Lock** constraint to position and lock a bolt in a hole.

Align constraints

An **Align** constraint is more versatile than an **Align/Lock** constraint because you can align other objects besides axes.

An **Align** constraint may be more useful in situations such as the following:

- When you need components to have the ability to rotate. For example, the blade subassemblies in this toy helicopter need to be constrained to the propeller shaft so their axes are aligned, but the blade subassemblies must rotate around the common axis.

- In simple situations, such as when you want to constrain two planar faces.
- In situations where some other constraint determines the orientation. For example, if you have a triangle formed by three struts with holes near their ends and bolts going through the holes. You would use **Align** constraints instead of **Align/Lock** constraints because the length of the struts determines the angle between them.

1.5.1.3 Assembly constraint notes

Tips for selecting geometry objects

While creating constraints, you can select geometry in a component and drag the component to another location. If you hold down the Alt key while dragging, the component rotates. This is a fast way to reposition components, for example, to help you select the geometry you want for the constraint.

Note Because Fix and Bond constraints are designed to hold components in place, the ability to drag components is disabled when you create or edit Fix or Bond constraints in order to prevent accidental movement of the components.

Geometry that is not available for assembly constraints includes the following:

- Faceted geometry
- Geometry that is not a product interface object when the component has a product interface that restricts geometry selection for assembly constraints. For more information, see *Product Interface dialog box*.
- Convergent modeling geometry is often not available because the geometry may be faceted or have other characteristics that make it not selectable for assembly constraints.

When you constrain to cylinders, cones, or tori, you can select either the axis or face of the object for the constraint.

Variable positioning behavior with assembly constraints

You can override a component position so it has a different position in higher-level assemblies than in its parent subassembly. For more information, see *Position override*.

Note When you use variable positioning on a component that includes Fix assembly constraints, the inherited version of each Fix constraint is a Bond constraint. NX converts inherited Fix constraints to Bond constraints because higher-level Fix constraints could cause undesirable behavior, such as preventing movement of the parents of the fixed component in higher-level assemblies. Inheriting to a Bond constraint allows the parent and fixed component to move together, but restricts independent movement of the fixed component.

Bond constraints that are converted from inherited Fix constraints are identified in the **Info** column of the **Assembly Navigator** and in reports run from the **Information** menu.

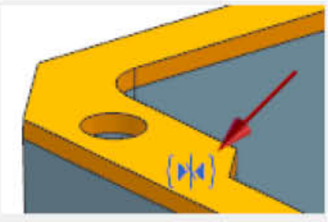
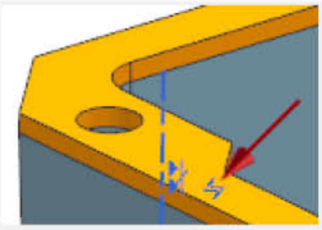
When you open an assembly that includes Fix constraints inherited by variable positioning applied in an earlier release of NX, the higher-level inherited Fix constraints are converted to Bond constraints.

1.5.1.4 Import a part file with assembly constraints

When you import a part file that has assembly constraints, the constraints are included.

However, when you import a part file that has a constraint with arrangement-specific suppression states, the imported part uses the suppression state of that constraint in its active arrangement. Arrangements are not imported with part files, because the assembly that contains the original component may have different arrangements than the assembly to where the component is imported.

Constraint-related symbols in the graphics window

Image	Description
	When an assembly constraint is surrounded by { and } brackets in the graphics window, the constraint is non-associative and will be deleted when you exit the dialog box where you created the constraint. Situations where the brackets may appear include the following: • When the assembly constraint is created in the **Move Components** dialog box. • When the assembly constraint is created in the **Assembly Constraints** dialog box while the **Associative** check box is not selected. The brackets only appear in the graphics window. Because the constraint is temporary, the **Assembly Navigator** and **Constraint Navigator** have no corresponding symbol.
	When you select the **Prefer Touch** option while creating a Touch Align constraint, a lightning bolt symbol may be temporarily displayed. The symbol indicates that NX has not yet determined whether the constraint is a Touch constraint or an Align constraint. NX switches between the types as needed to make the constraint solve. The lightning bolt symbol disappears when NX makes its decision, or when you exit the dialog box, whichever is sooner.

1.5.1.5 Constraints in arrangements

When you make a change that requires NX to solve constraints, in order to speed up the solving time, NX makes the updates only in arrangements that are in use. Arrangements that are in use include the following:

- Any arrangement that is the active arrangement of the displayed part.
- Any arrangement that is used in a component of the displayed part, including the work part in design-in-context cases.

- Any arrangement that is used for the definition of a WAVE-linked feature or geometric expression.
- Any arrangement used in an exploded view of the displayed part or a sequence of the displayed part.
- Any arrangement used in a drafting view that belongs to the displayed part.

For each unused arrangement, NX postpones solving the constraints until one of the following occurs.

- The unused arrangement is put into use.
- The assembly is saved.

  Constraints in all arrangements must be solved when an assembly is saved to prevent potential problems with Teamcenter Integration with NX or with programs such as NX/Open.

Use the **Show and Hide Constraints** command to control the visibility of:

- Selected constraints
- All the constraints associated with selected components
- Only the constraints between selected components

You can also control the visibility of components whose constraints are hidden after a show and hide constraints operation.

Application	Assemblies
Command Finder	**Show and Hide Constraints**

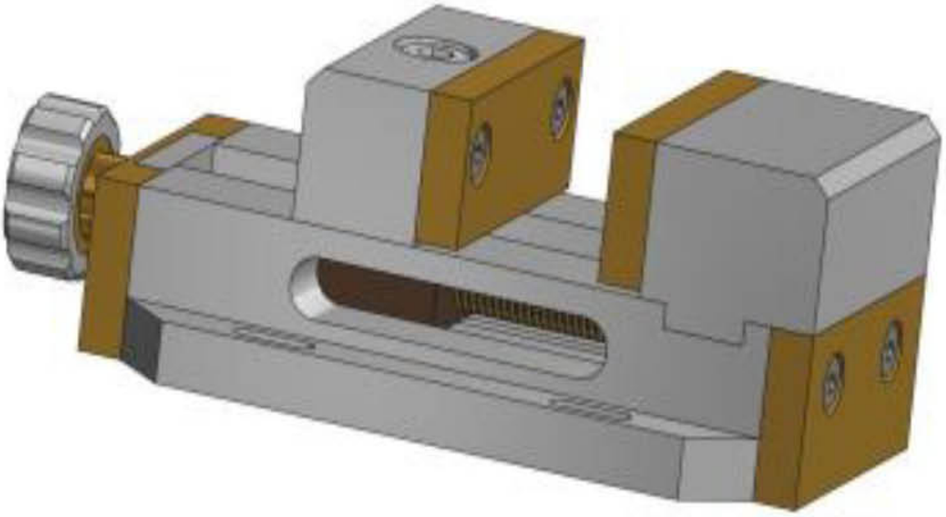
Show only selected assembly constraints

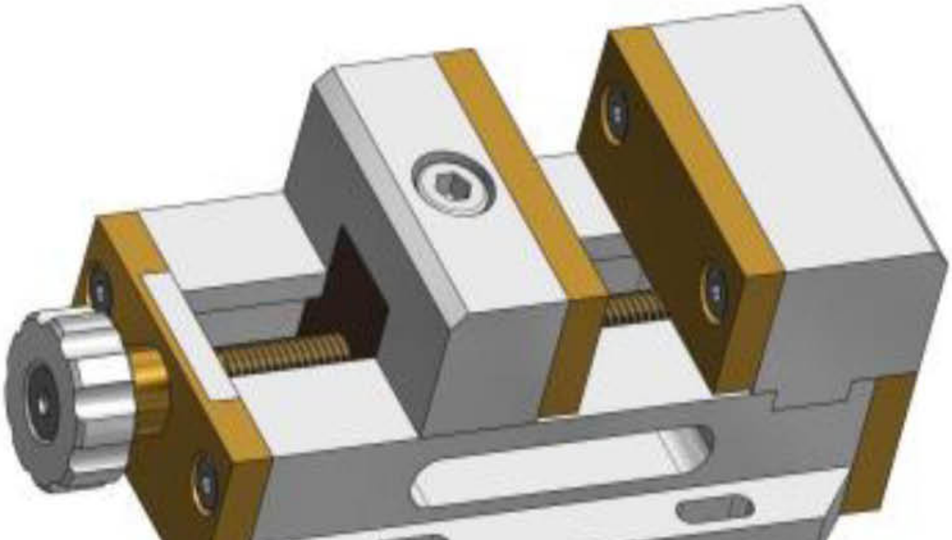
1. Choose **Assemblies** tab→**Component Position** group→**Show and Hide Constraints** .

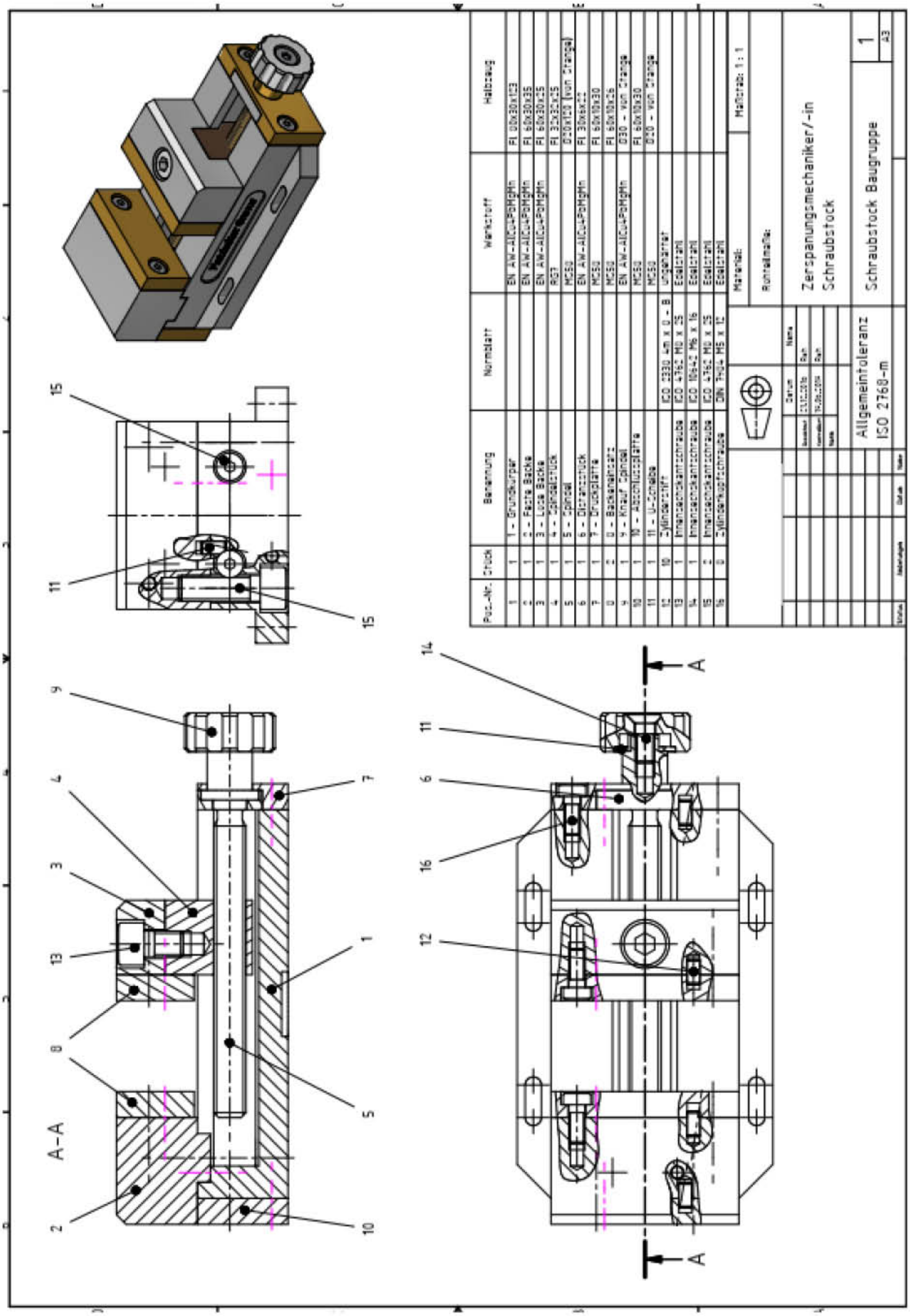
   The **Show and Hide Constraints** dialog box opens.

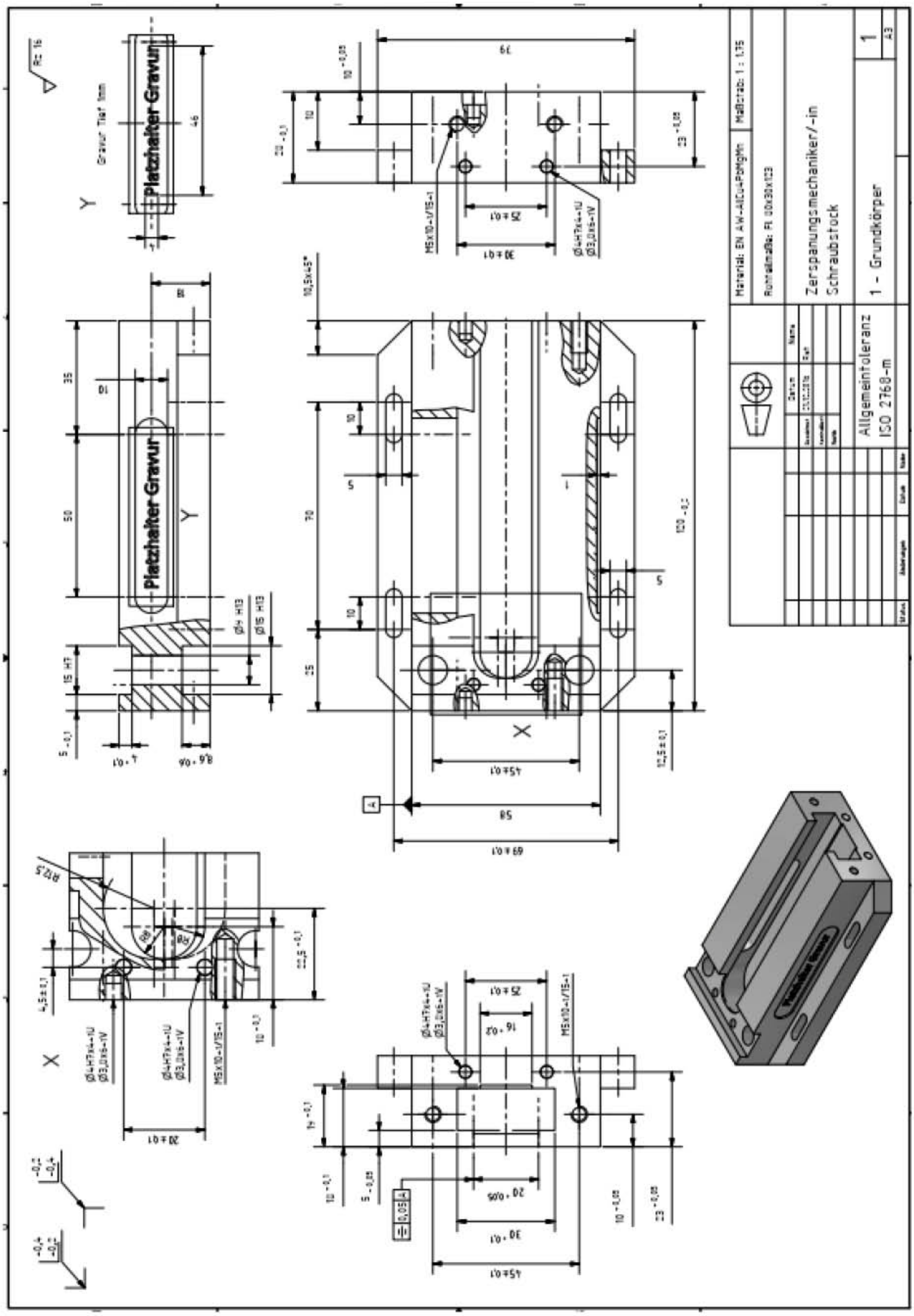
2. While **Select Component or Constraints** is active, select one or more of the following in the graphics window:
   - Components whose constraints you want to show or hide
   - Constraints you want to show besides the constraints affected by selected components
3. In the **Settings** group, set **Visible Constraints** to one of the following:
   - Choose **Between Components** if you want only constraints between your selected component, plus any constraints you select directly in step 2, to be visible.
   - Choose **Connected to Components** if you want all constraints that belong to your selected components, plus any constraints you select directly in step 2, to be visible.
4. Select the **Change Component Visibility** check box if you want the components that are not a part of the results to be hidden.
5. Select the **Filter Navigator** check box if you want components that are not a part of the results to be placed under a More node in the **Assembly Navigator**.
6. Choose **OK** or **Apply**.

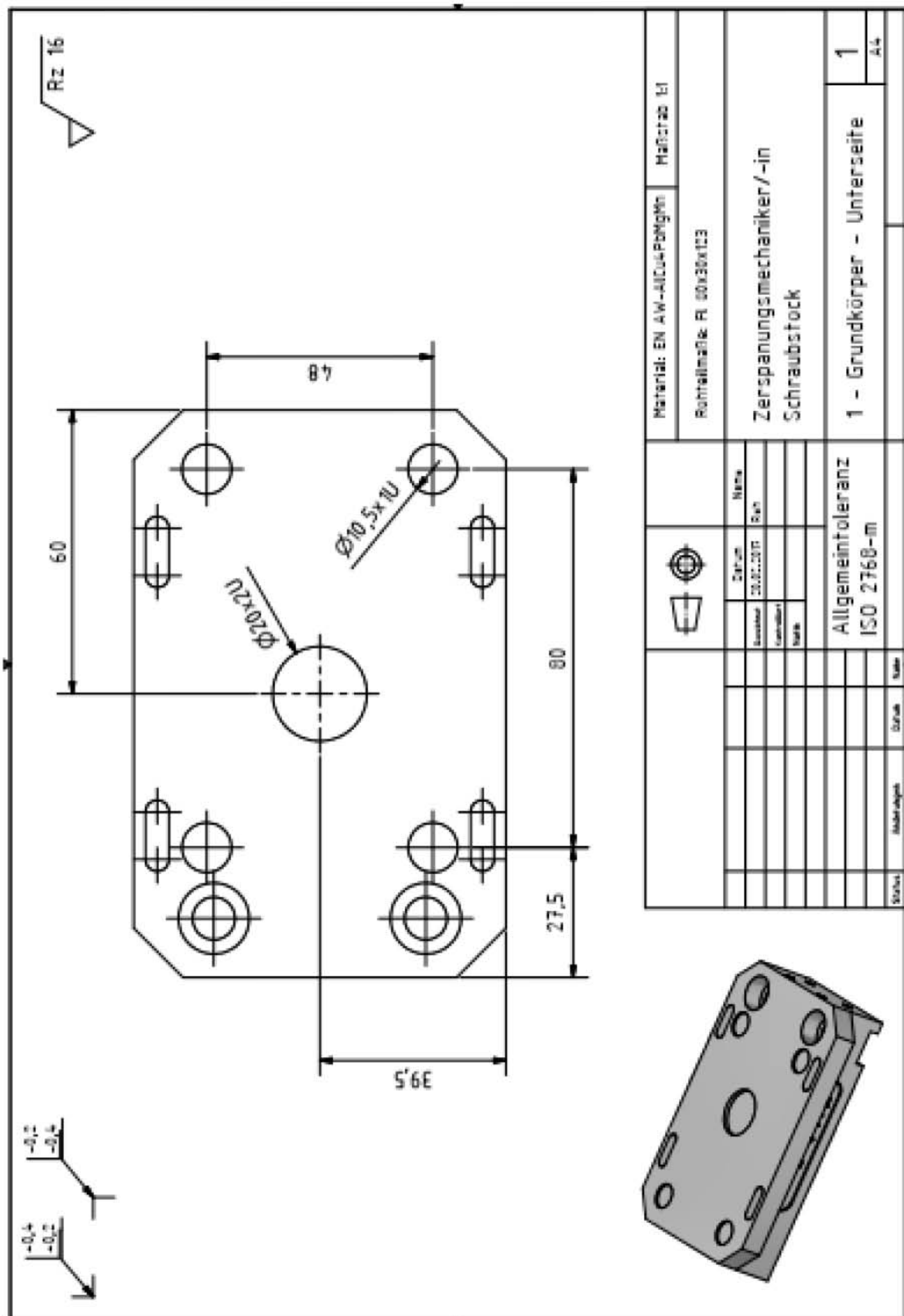
1.6 Application exercises

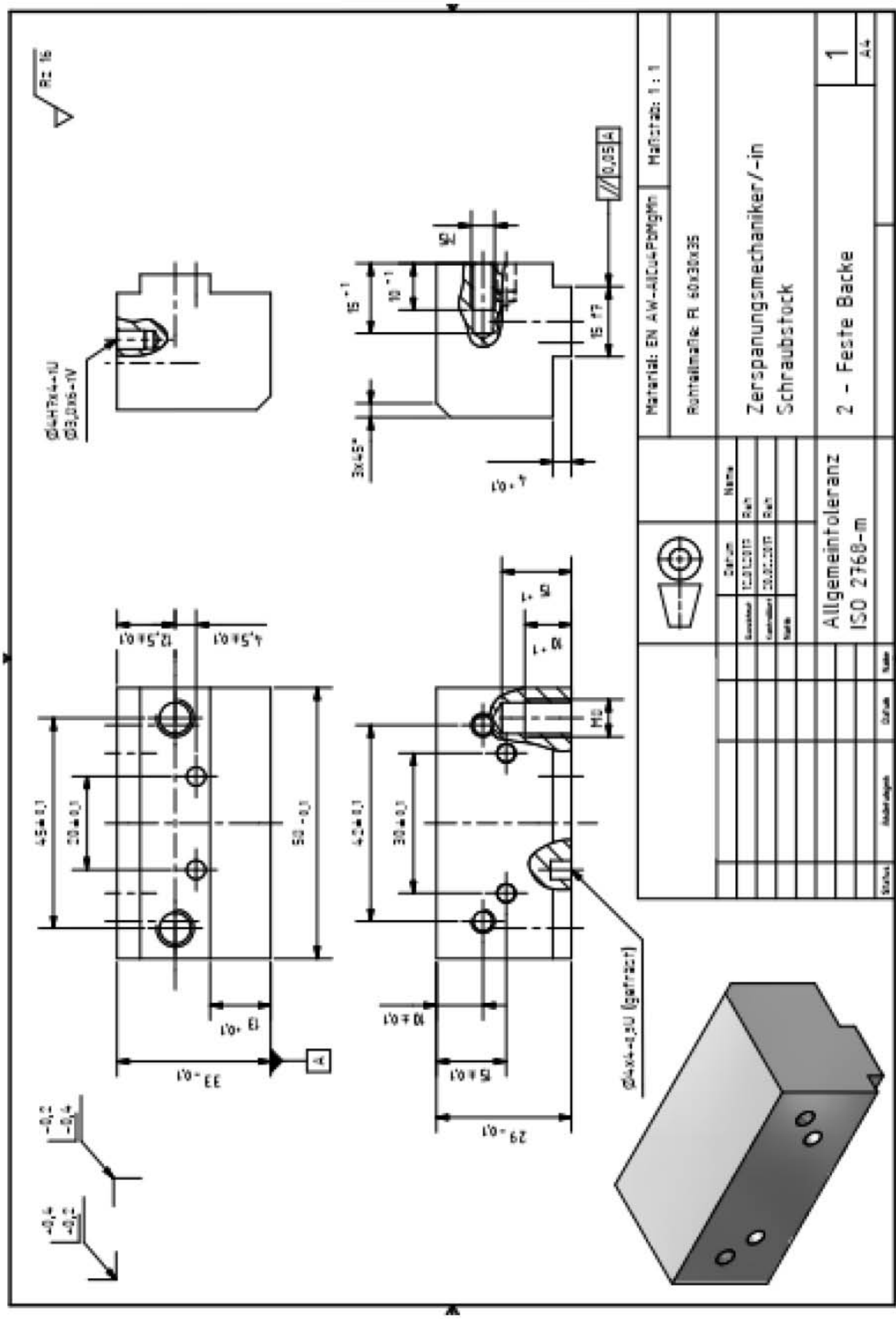


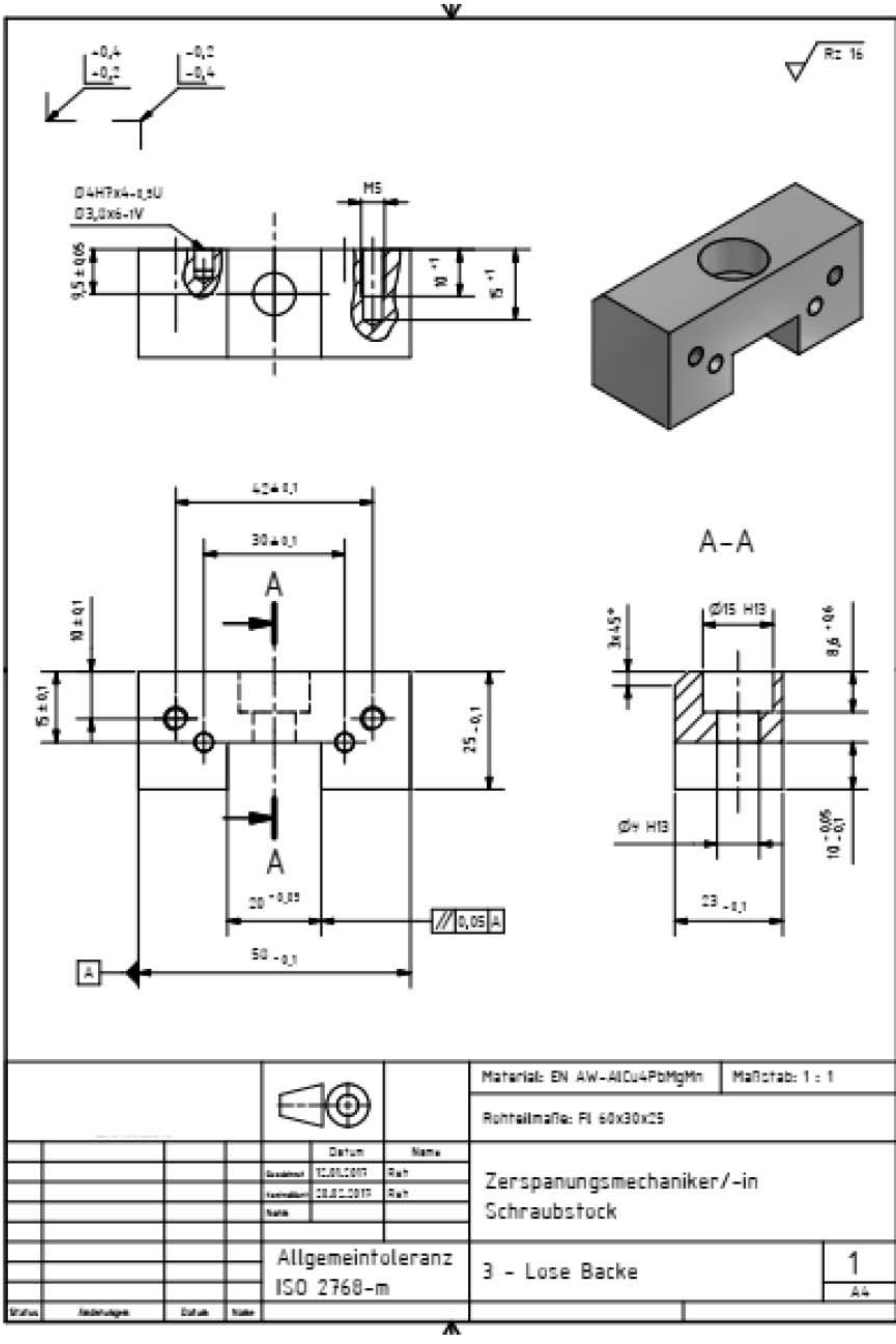



A-A
Zerspanungsmechaniker/-in
Schraubstock
Allgemeintoleranz
ISO 2768-m
Schraubstock Baugruppe

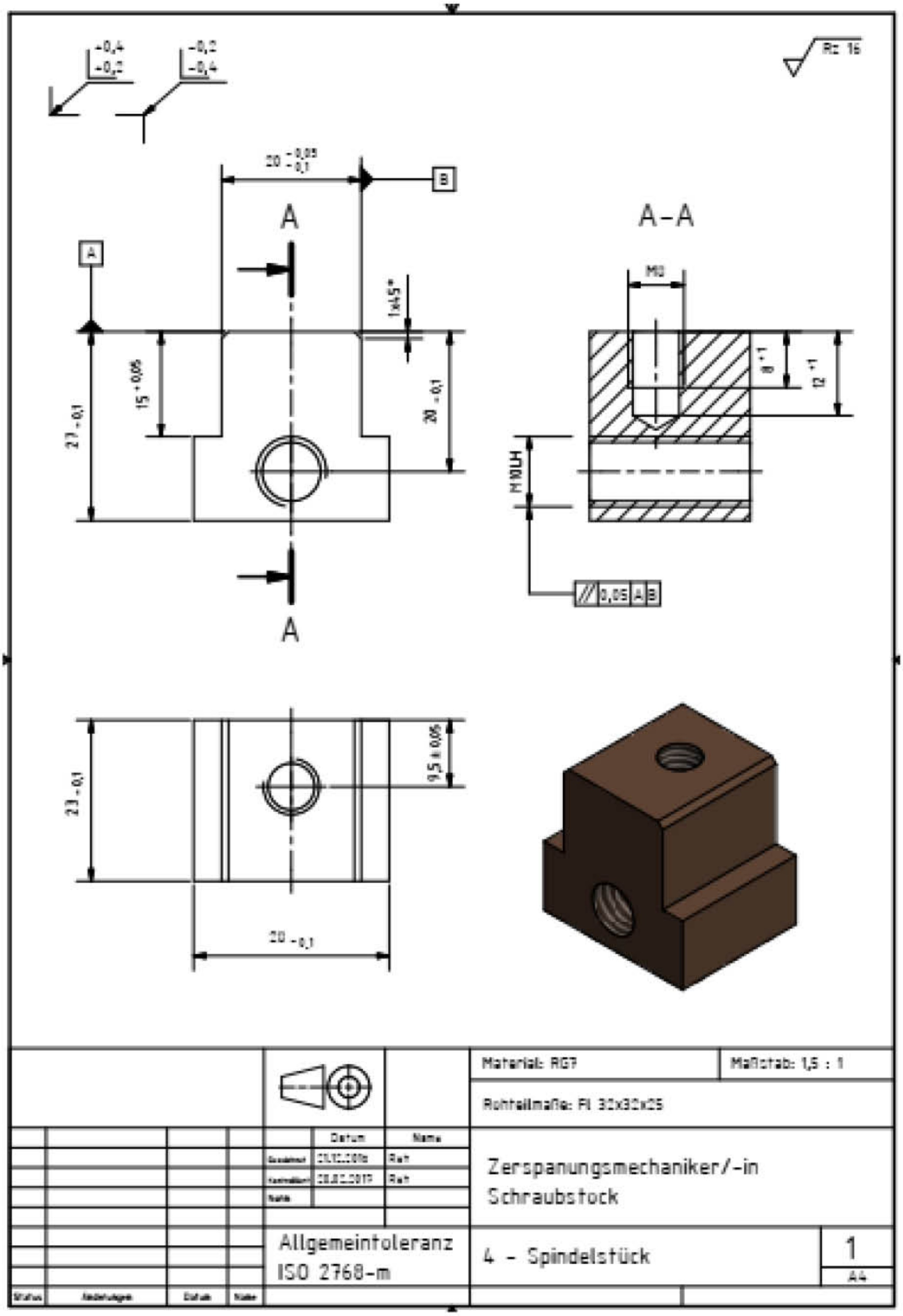


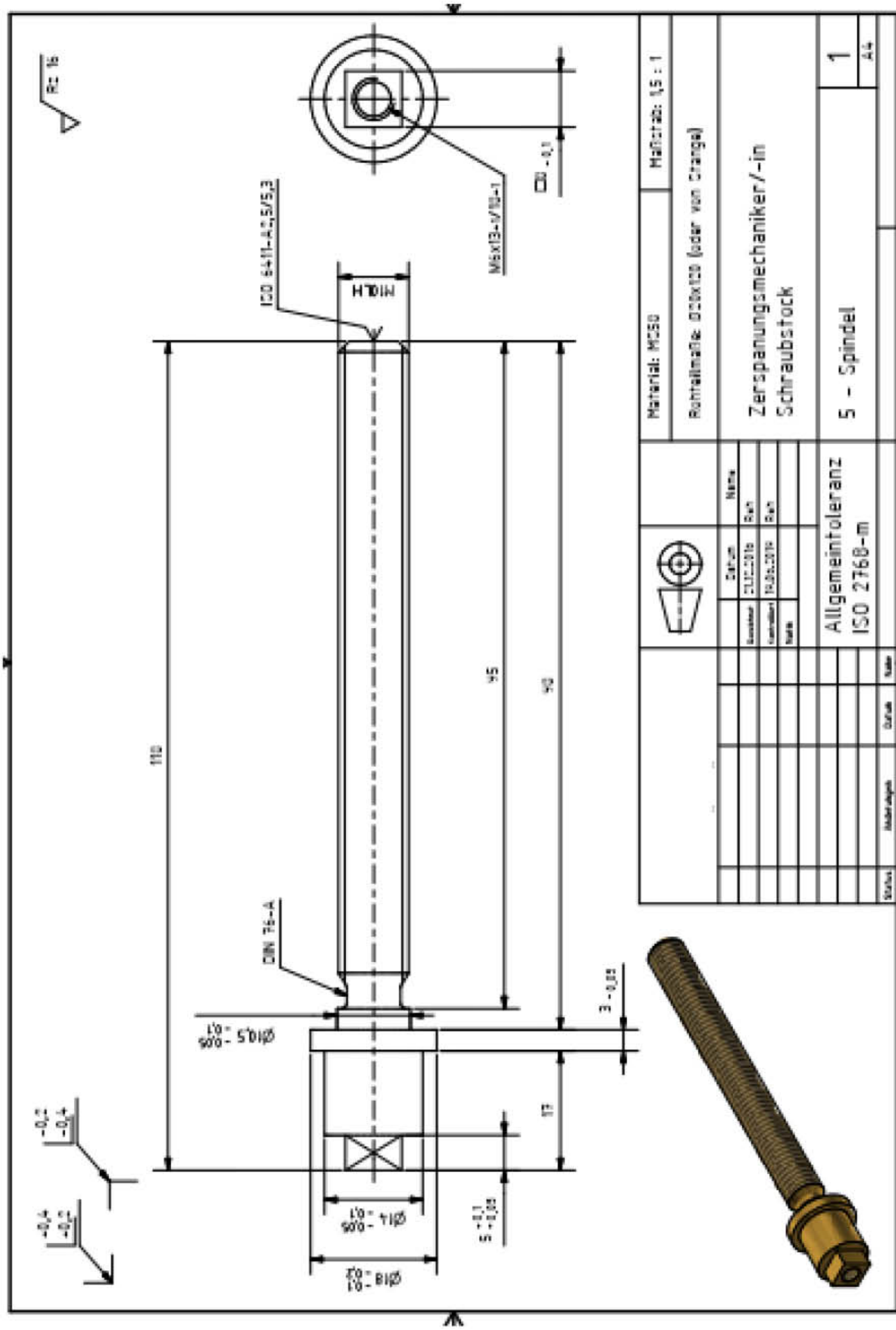

Rz 16
48
60
Ø20x2U
Ø10,5x1U
80
27,5
39,5
Maßstab 1:1
Zerspanungsmechaniker/-in
Schraubstock
Allgemeintoleranz
ISO 2768-m
1 - Grundkörper - Unterseite
1
A4

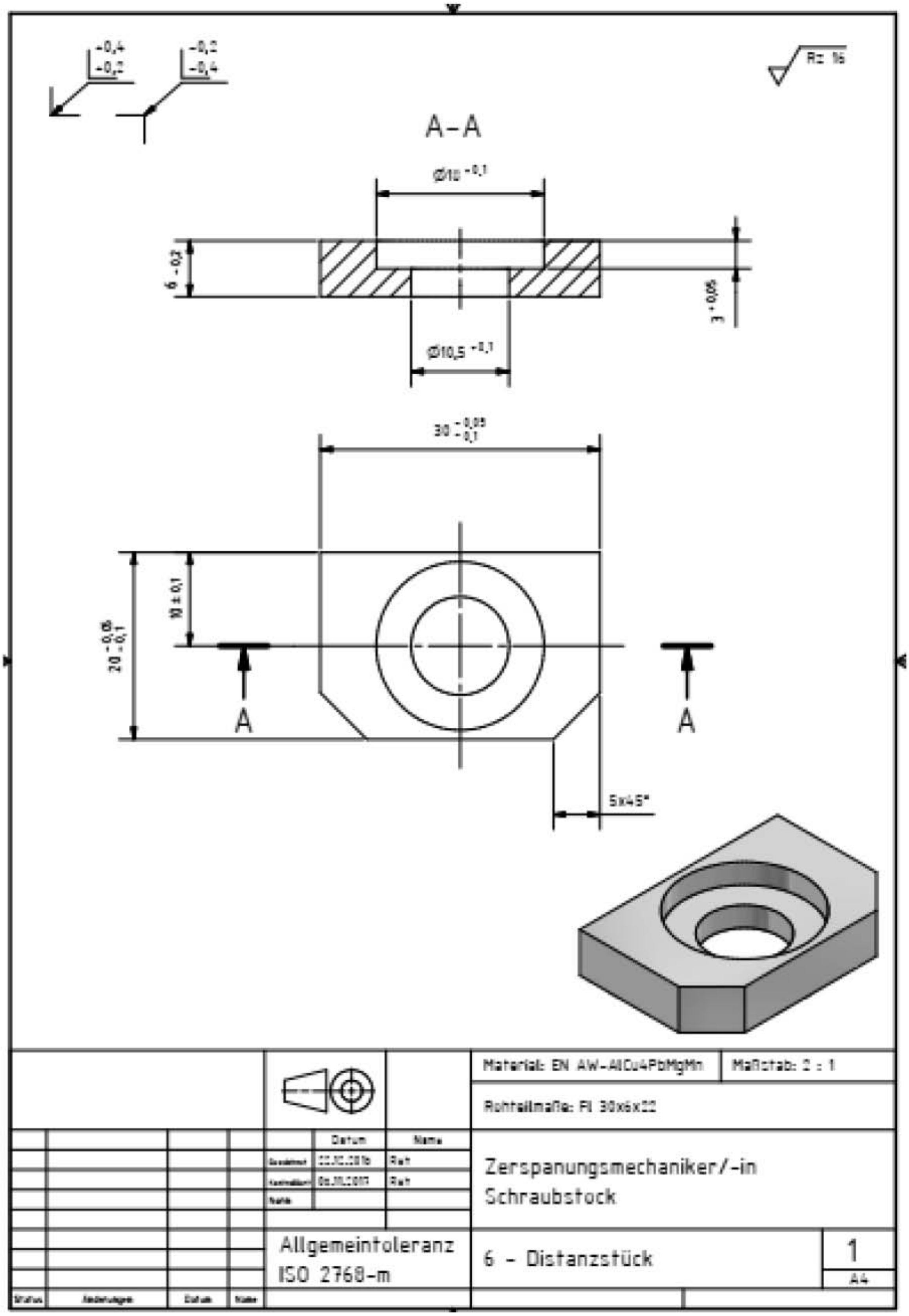

Material: EN AW-AlCu4PbMgMn
Maßstab: 1 : 1
Rohteilmaße: Fl 60x30x35
Zerspanungsmechaniker/-in
Schraubstock
2 - Feste Backe
1
A4
Allgemeintoleranz
ISO 2768-m
Ø4x4-0,5U (gefräst)
3x45°
M8
Rz 16

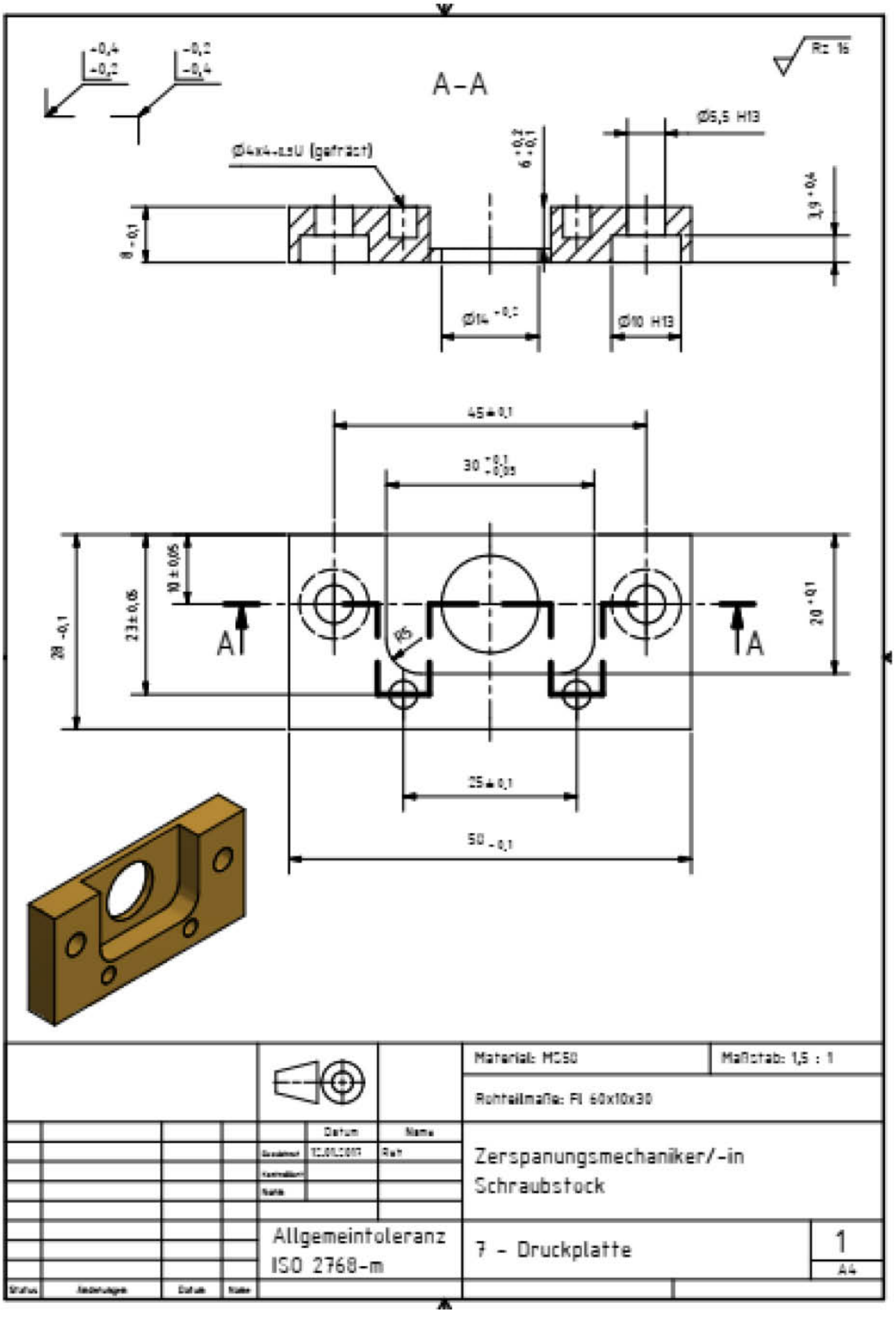


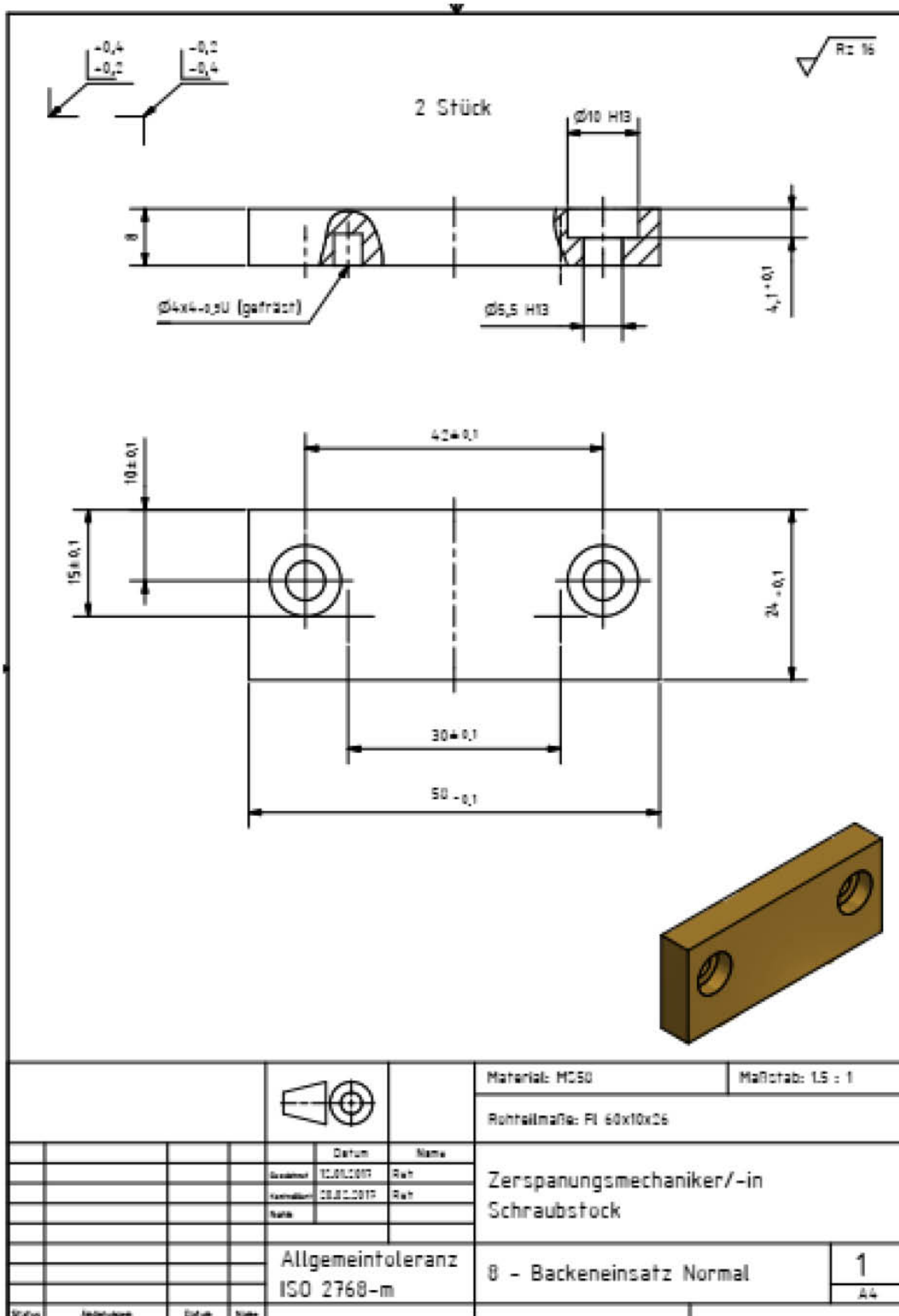
Rz 16
M5
A-A
A
A
Ø15 H13
Ø4 H13
3x45°
25 -0,1
50 -0,1
23 -0,1
Material: EN AW-AlCu4PbMgMn
Maßstab: 1 : 1
Rohteilmaße: Fl 60x30x25
Datum
Name
Zerspanungsmechaniker/-in
Schraubstock
Allgemeintoleranz
ISO 2768-m
3 - Lose Backe
1
A4

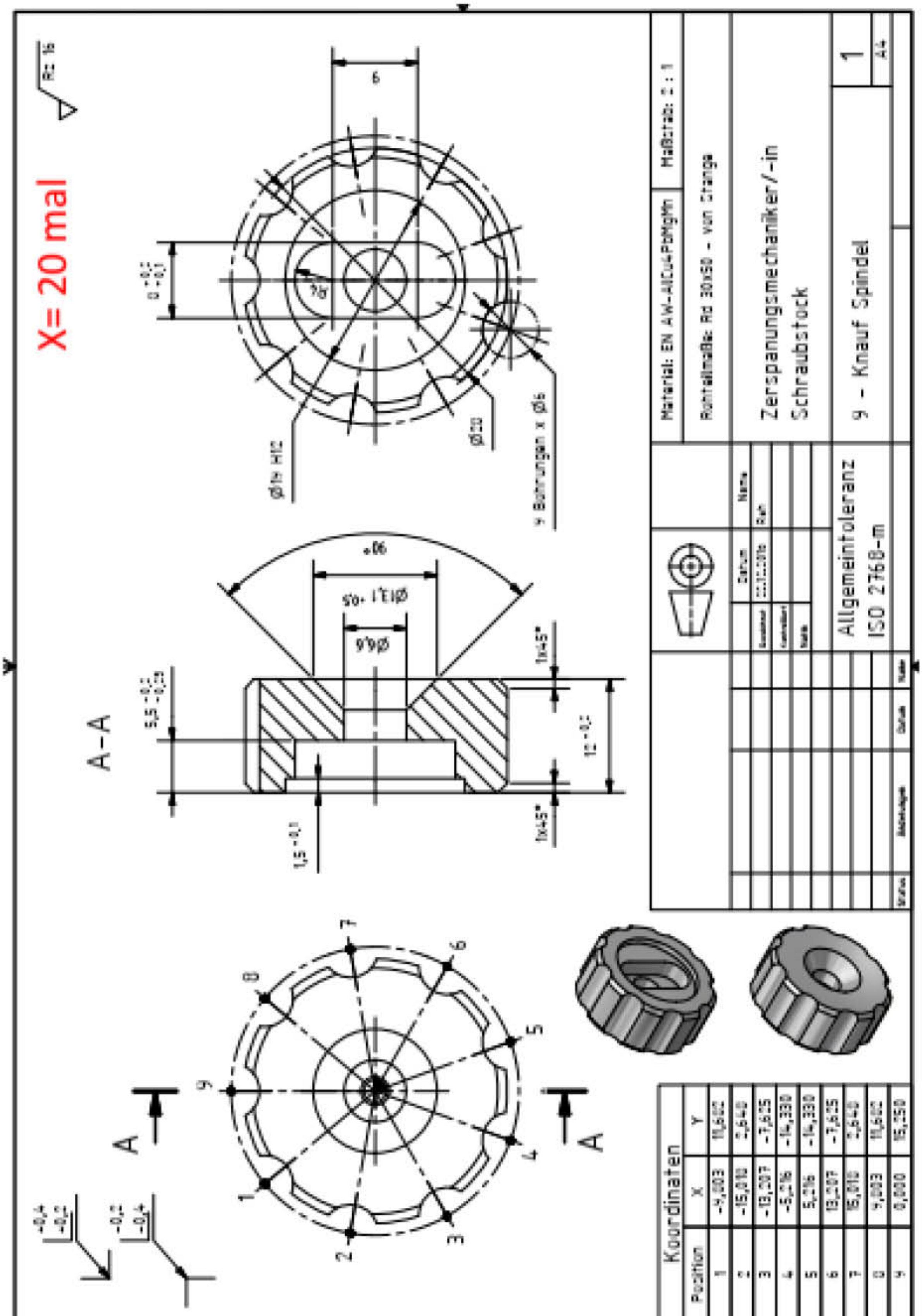

+0,4
+0,2
-0,2
-0,4
Rz 16
20 -0,05 -0,1
B
A
A
A-A
M8
1x45°
15 +0,05
27 -0,1
20 -0,1
8 +1
12 +1
M10LH
0,05 A B
A
23 -0,1
9,5 ± 0,05
20 -0,1
Material: RG7
Maßstab: 1,5 : 1
Rohteilmaße: Fl 32x32x25
Datum
Name
Zerspanungsmechaniker/-in
Schraubstock
Allgemeintoleranz
ISO 2768-m
4 - Spindelstück
1
A4
Status
Änderungen
Datum
Name

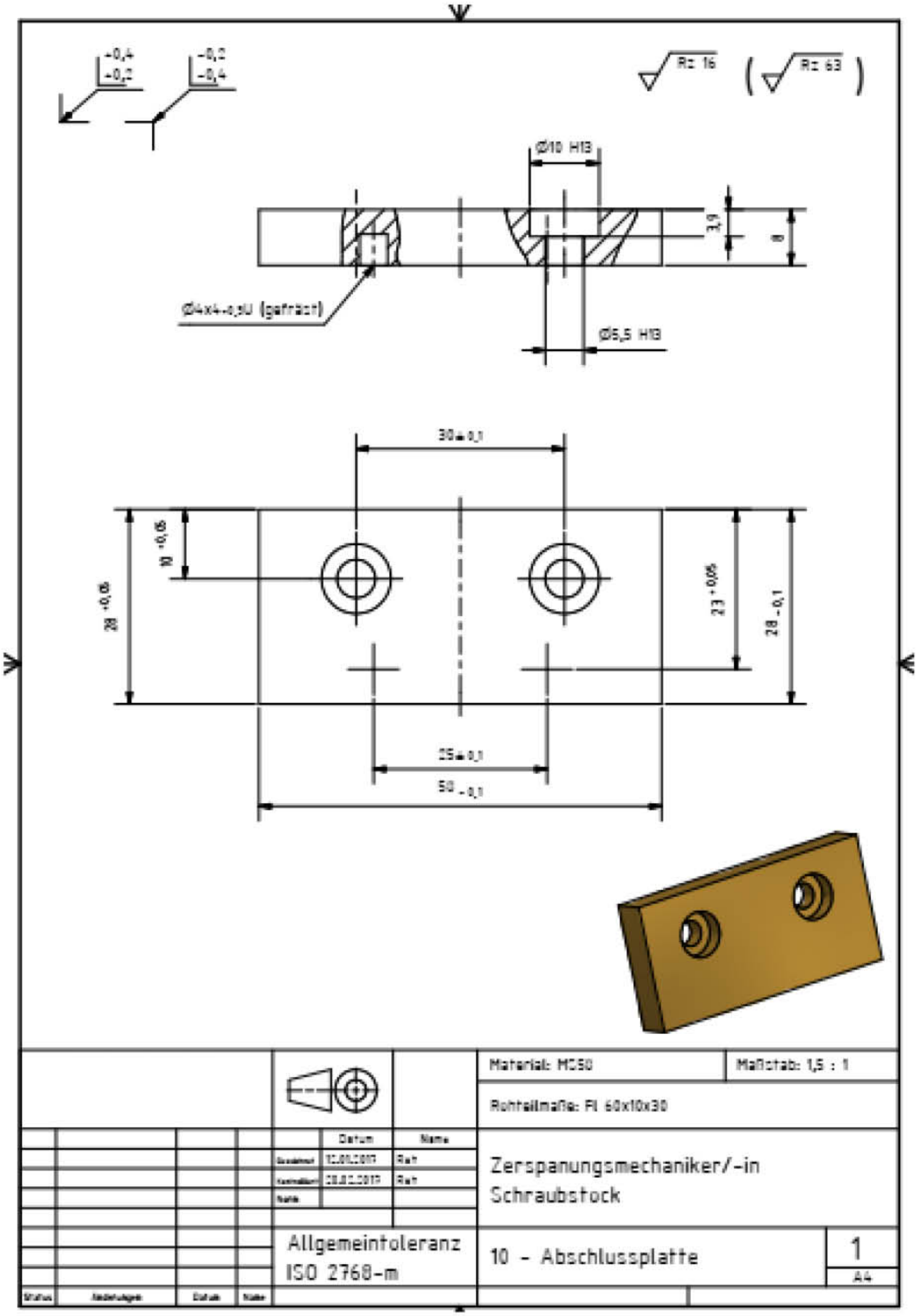

Zerspanungsmechaniker/-in
Schraubstock
5 - Spindel
Allgemeintoleranz
ISO 2768-m
DIN 76-A
1
A4

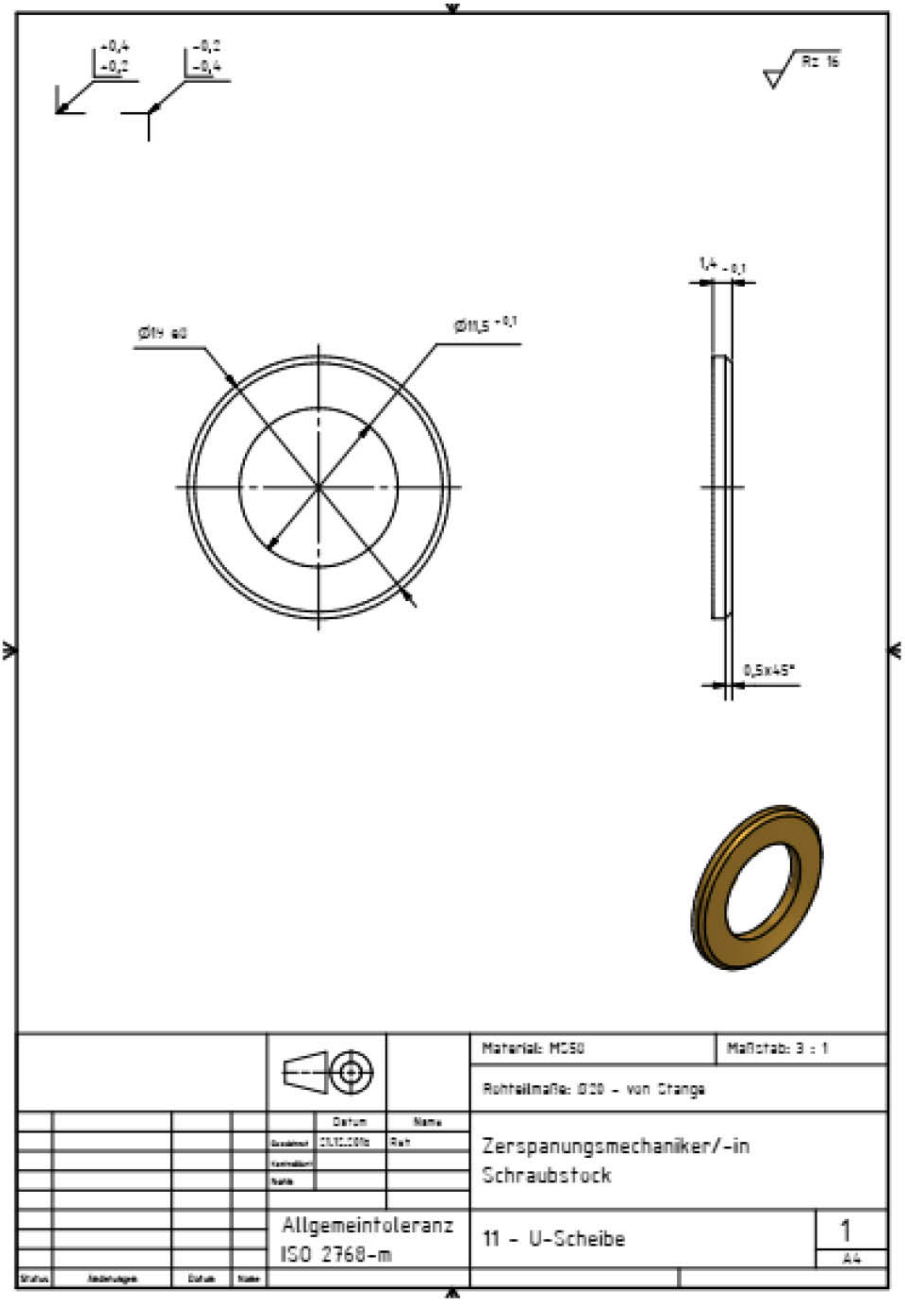

A-A
Rz 16
Ø10,5 +0,1
30
20
10 ± 0,1
A
A
5x45°
Material: EN AW-AlCu4PbMgMn
Maßstab: 2 : 1
Rohteilmaße: Fl 30x6x22
Datum
Name
Zerspanungsmechaniker/-in
Schraubstock
Allgemeintoleranz
ISO 2768-m
6 - Distanzstück
1
A4


A-A
Rz 16
Ø6,5 H13
Ø10 H13
45 ± 0,1
25 ± 0,1
50 -0,1
28 -0,1
23 ± 0,05
10 ± 0,05
Material: MC50
Maßstab: 1,5 : 1
Rohteilmaße: Fl 60x10x30
Datum
Name
Zerspanungsmechaniker/-in
Schraubstock
Allgemeintoleranz
ISO 2768-m
7 - Druckplatte
1
A4


2 Stück
Rz 16
Ø10 H13
Ø6,5 H13
4,1 +0,1
8
42±0,1
10±0,1
15±0,1
24 -0,1
30±0,1
50 -0,1
Material: MC50
Maßstab: 1,5 : 1
Rohteilmaße: Fl 60x10x26
Datum
Name
Zerspanungsmechaniker/-in
Schraubstock
Allgemeintoleranz
ISO 2768-m
8 - Backeneinsatz Normal
1
A4


X= 20 mal
A–A
A
Koordinaten
Position X Y
1 -9,003 11,602
2 -15,018 2,640
3 -13,307 -7,655
4 -5,216 -14,330
5 5,216 -14,330
6 13,307 -7,655
7 15,018 2,640
8 9,003 11,602
9 0,000 15,250
9 Bohrungen x Ø6
Ø16 H12
Ø6,6
Ø13,1 +0,5
90°
1x45°
Material: EN AW-AlCu4PbMgMn
Rohteilmaße: Rd 30x30 – von Stange
Zerspanungsmechaniker/-in
Schraubstock
9 – Knauf Spindel
Allgemeintoleranz
ISO 2768-m
1
A4


Ø10 H13
3,9
8
Ø4x4-0,3U (gefräst)
Ø6,5 H13
30±0,1
10 +0,05
28 +0,05
23 +0,05
28 -0,1
25±0,1
50 -0,1
Material: MC50
Maßstab: 1,5 : 1
Rohteilmaße: Fl 60x10x30
Datum
Name
12.01.2017
Rat
28.02.2017
Rat
Zerspanungsmechaniker/-in
Schraubstock
Allgemeintoleranz
ISO 2768-m
10 - Abschlussplatte
1
A4
Status
Änderungen
Datum
Name



Rz 16
Ø14 e8
Ø11,5 +0,1
1,4 -0,1
0,5x45°
Material: MC50
Maßstab: 3 : 1
Rohteilmaße: Ø20 - von Stange
Datum
Name
Zerspanungsmechaniker/-in
Schraubstock
Allgemeintoleranz
ISO 2768-m
11 - U-Scheibe
1
A4

LESSON 2: ADVANCED SHEET METAL DESIGN WITH NX: TOOLS AND TECHNIQUES

Objective: By the end of this training unit, the trainees will be able to:

- Understand the sheet metal model, the process of creating products from sheet metal
- Understand the material standards applied in sheet metal
- Understand drawing commands and modifying commands with NX software to create sheet metal product models
- Draw simple and complex sheet metal product models with NX software
- Create technical drawing from sheet metal with NX software

Contents:

2. 1 Getting started with Sheet Metal with Siemen NX software

2.1.1 Sheet Metal

The Sheet Metal application provides an environment for the design of sheet metal parts used in machinery, enclosures, brackets, and other parts normally manufactured with a brake press.

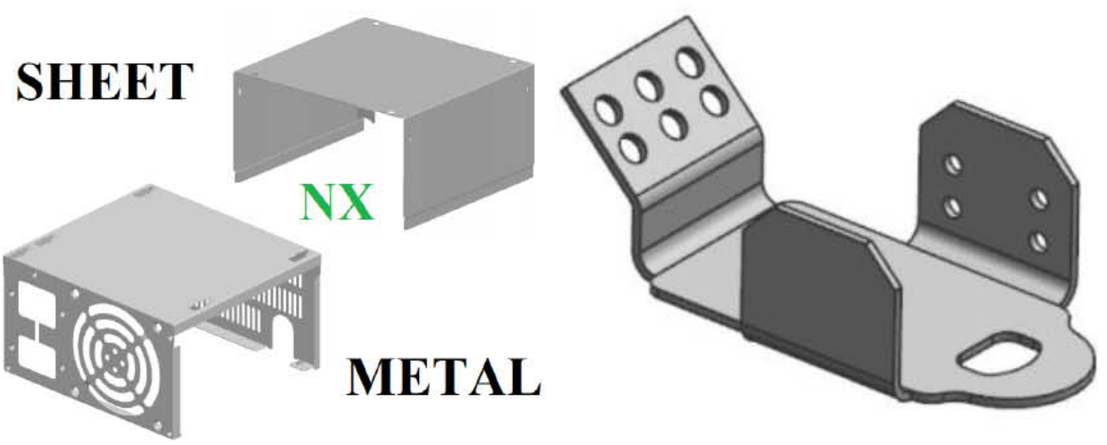


Figure 2.1: Product design sheet metal

You can:

- Create uniform thickness sheet metal parts using basic features such as flanges, tabs, and jogs. You can also close corners, and create cutouts and stiffening features.
- Create complex sheet metal parts by designing the solid model in the Modeling environment and then converting it to a sheet metal part.
- Some Modeling commands are also available in the Sheet Metal application that can be used to create holes and slots, as well as copy, paste, pattern, and mirror sheet metal features.
- Import, clean up, and convert a non-sheet metal part to a valid sheet metal part.
- Export flat pattern data of sheet metal parts to other applications.

Starting a Sheet Metal part To start a new sheet metal part, **click Home > Standard > New**

 on the ribbon, and choose **Sheet metal** template on the **New dialog/OK**

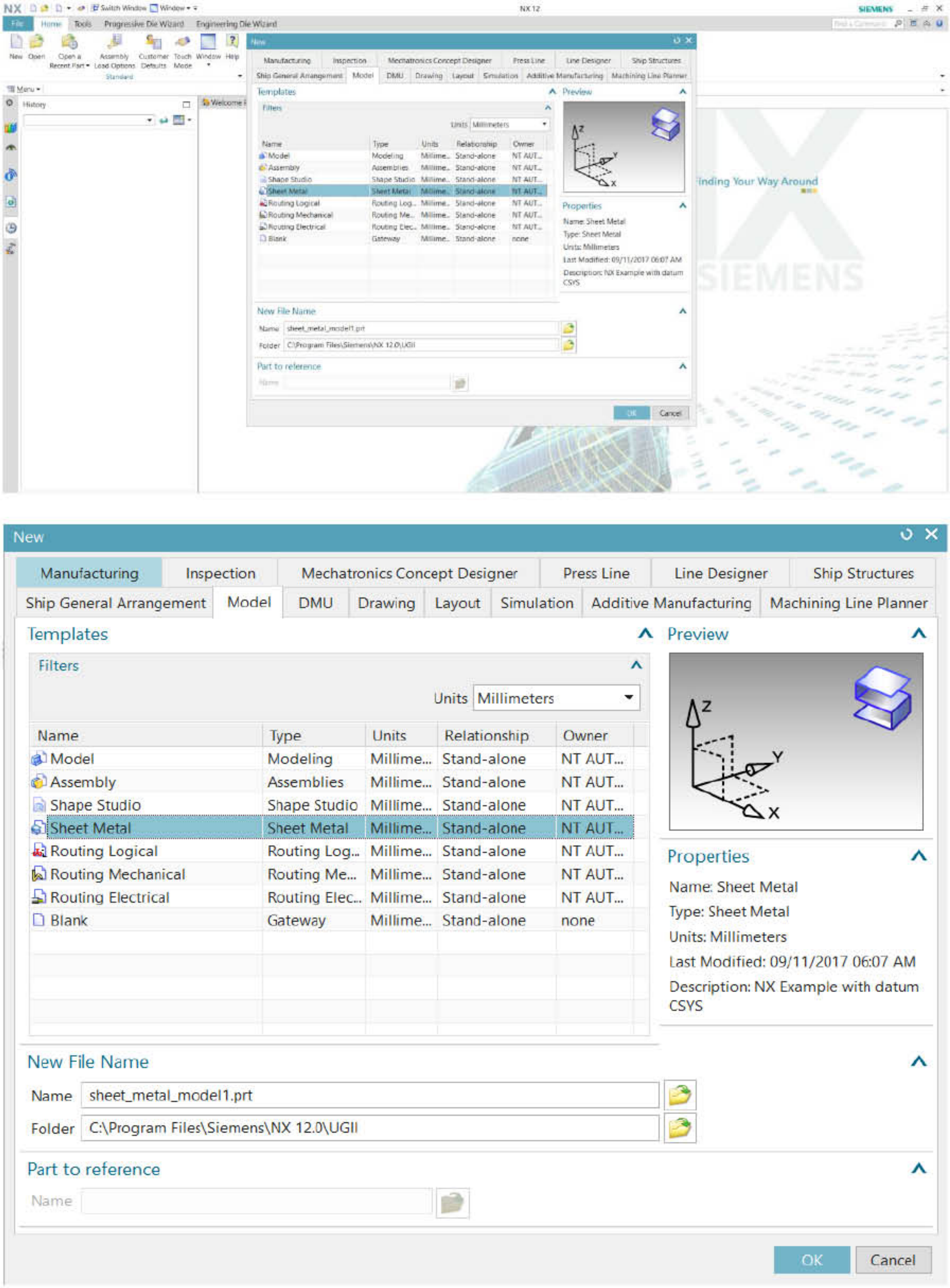
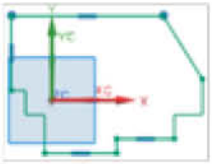


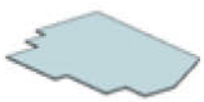
Figure 2.2: Create design a Sheet metal interface

2.1.2 Recommended workflow to create a simple part



Sketch the shape

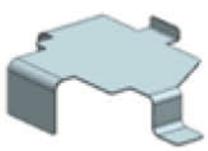
Sketch the shape of the base feature, or use an existing sketch or section.



Construct the base feature

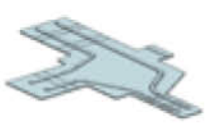
Construct the base feature to define the part's shape.

This base feature is normally a Tab, but you can also create a Contour Flange or a Lofted Flange.



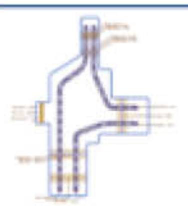
Add features

Add features such as Flanges, Jogs, or Bends, to further define the shape of the formed sheet metal part.



Unbend and add punch features or cutouts

Unbend the bend areas where needed, and place Holes, Cutouts, Beads, Dimples, or Louver features on the part.

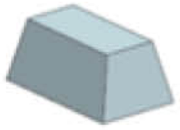


Create a Flat Pattern

Create a Flat Pattern of the part for drafting and later manufacturing.

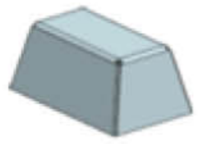
Chart 2.1: Recommended workflow to create a simple part

2.1.3 Recommended workflow to create enclosures with odd shapes or angles



Create a solid body

Create a solid body that encompasses the interior volume of the final enclosure shape, using tools in the Modeling application.

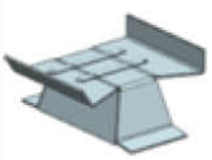


Wrap the solid body with sheet metal

Use the **Sheet Metal from Solid** command to wrap the solid body with sheet metal.

Add features

Add features such as Flanges, Jogs, or Bends, to further define the basic shape of the formed sheet metal part.

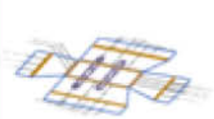


Unbend to add more features

Unbend the bend areas where needed, and place Holes, Cutouts, Beads, Dimples, or Louver features on the part.

Rebend the unbent areas

Rebend the flattened bend areas to complete the part.



Create a Flat Pattern

Create a Flat Pattern of the part for drafting and later manufacturing.

Chart 2.2: Recommended workflow to create enclosures with odd shapes or angles

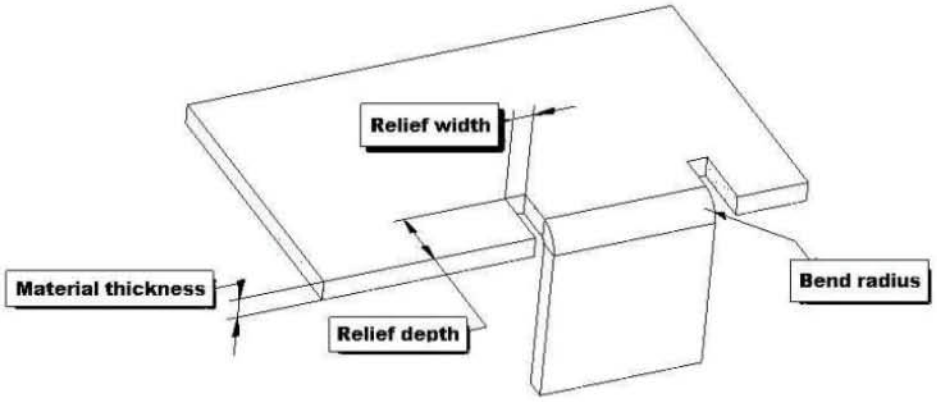
2.2. Sheet metal material standards and sheet metal preferences

2.2.1 Sheet metal material standards

Safely withstanding the expected maximum load without permanent deformation (or to stay within the specified deflection) is a basic requirement for any product. The "resistance" against the load is a function of the cross-section and the mechanical properties (or in loose terms the "strength") of the material. The most important mechanical properties are the yield strength, tensile strength, and elongation.

The most common Sheet Metal Part Properties are type of material and bend allowances.

You can define these properties by clicking **File > Preferences > Sheet Metal.** On the **Sheet Metal Preferences** dialog, select **Parameter Entry > Value Entry** and type-in values in the **Material thickness**, **Bend radius**, **Relief depth**, and **Relief width** boxes. These parameters are illustrated in the following figure.



$$BA = \frac{\pi(R + KT)A}{180}$$

BA = Bend Allowance
R = Bend Radius
K = Neutral Factor = t/T
T = Material Thickness
t= Distance from inside face to the neutral sheet
A = Bend Ang

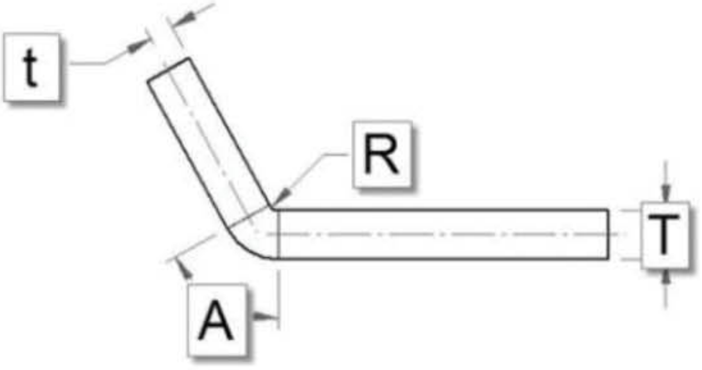


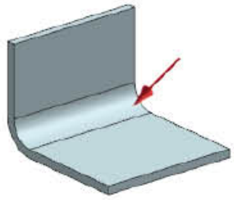
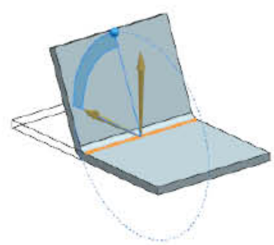
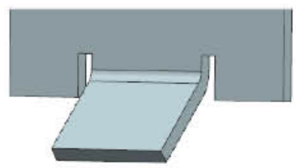
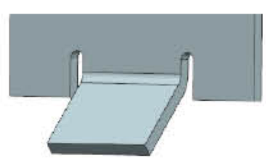
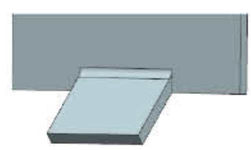
Figure 2.3: These parameters are illustrated in the following figure

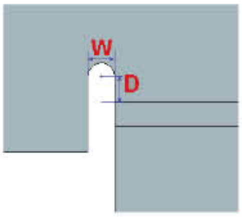
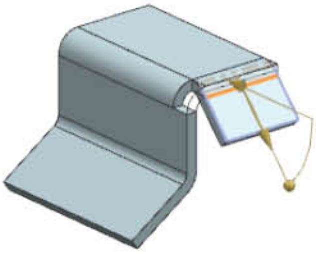
You can also define the bend allowance by using a bend table or your own bend allowance formula. To enter a bend allowance formula, select the **Bend Allowance Formula** option and type-in a value in the box. Click **OK** after specifying values on the **Sheet Metal Preferences** dialog.

Next, type-in a value in **Neutral Factor Value** box. The **Neutral Factor** is the ratio that represents the location of neutral sheet measured from the inside face with respect to the thickness of the sheet metal part. The **Neutral Factor** defines the bend allowance of the sheet metal part. The standard formula that calculates the bend allowance is given below.

2.2.2 Sheet metal preferences

Bend parameters for sheet metal features

Bend parameter	Description
Bend Radius	Sets the inside radius of a bend. 
Bend Angle	Sets the bend angle for the feature. The value must be greater than zero and less than 180 degrees. 
Bend Relief	**Square**  **Round**  **None** 
Extend Relief check box	Specifies whether to extend the bend relief to the edge of the part.

	 Extend Relief = ☑  Extend Relief = ☐
Depth, **Width** of the bend relief	Sets the depth (D) and width (W) of the reliefs. 
Neutral Factor	Sets a value for the neutral factor. The neutral factor depends on the properties of the material that is being bent. While bending, the outer surface of the bend is under tension and the inner surface is under compression. There is a layer in between that is neither in compression nor tension. The ratio of the distance of this surface from the inner bend to the thickness is the neutral factor. The default value is 0.33 and the value typically ranges from 0 to 1.
Corner Relief	Specifies that you want to apply corner relief to features that are adjacent to the feature that you are constructing. **Bend Only** Applies corner relief only to the bend portion of the adjacent features.  **Bend/Face** Applies corner relief to both the bend and face portions of the adjacent features.

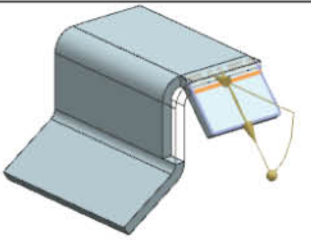
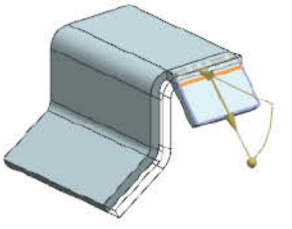
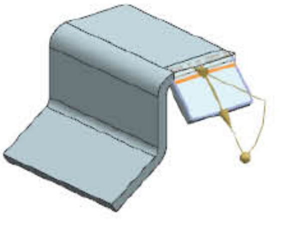
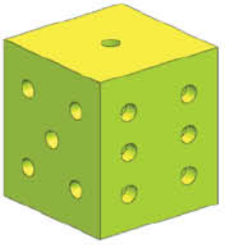
	 **Bend/Face Chain** Applies corner relief to the entire chain of bends and web faces of the adjacent features.  **None** Applies no corner relief. 

Table 2.1: Bend parameters for sheet metal features

2.3 Creating sheet metal parts and features

2.3.1 Sheet Metal from Solid

Use the **Sheet Metal from Solid** command to create a sheet metal body using a solid body to model it. The solid represents the inner void of an enclosure. You select the faces from the solid to be panel (web) faces, and edges between the faces to be bend regions. You specify bend properties, or use the ones you established in the **Sheet Metal Preferences** dialog box. NX creates the model based on your inputs. You can then treat this body as you would any other sheet metal body.



Solid

Resulting sheet metal body

Steps to do

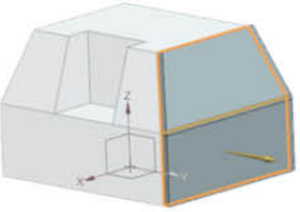
1. Create a solid body in Modeling.
2. Start Sheet Metal.

3. Choose **Home** tab→ **Convert** group→ **Sheet Metal from Solid** .
4. Select the web faces that you want to include in your sheet metal body.

   As you select the web faces, NX performs the following actions:

   - Infers the bend edge and adds it to the **List** group.
   - Shows a preview of the result (if there is only one logical bend option).
   - Displays a message informing you that the sheet metal from solid preview may not exactly match the resultant feature.

   In this example, two faces are selected and only one bend edge is shared between them. NX displays a preview.



5. Click **OK** in the message box.
6. In the **List** group, under the **ID** column, select the newly added bend edge (**Bend1**) and specify the values for bend parameters and bend relief.

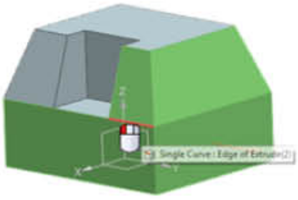
   In the **Bend Parameters** subgroup:
   - **Bend Radius = 5**

   In the **Relief** subgroup:
   - **Bend Relief = Square**
   - **Depth = 5**
   - **Width = 5**

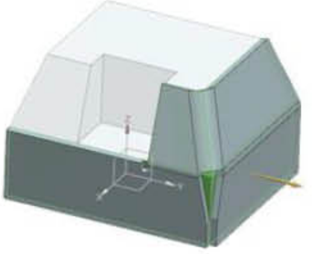
   The selected bend edge and the **Bend Radius**, **Bend Relief**, **Width**, and **Depth** columns under the **List** group reflect the changed values.

7. Select more web faces that you want to include in your sheet metal body.
   As you select web faces, NX infers the bend edge and adds it to the **List** group.
   If you select faces where there might be more than one bend edge, you must select one of the bend edges.

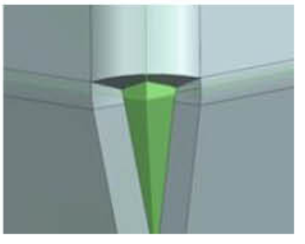
8. Select the bend edge.



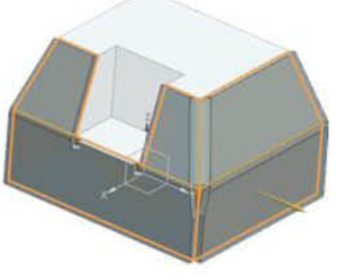
If you select an appropriate combination of faces and edges (creating a body that can be flattened), the preview reappears.



NX creates a clean open corner at the three bends.



9. Select more web faces that you want to include in your sheet metal body.



10. In the **List** group, under the **ID** column, individually select the bend edges (**Bend2**, **Bend3**, and **Bend4**), and specify the values for bend parameters and bend relief.

For this example:

ID	Bend Radius	Bend Relief	Width	Depth
Bend2	5	Square	5	5
Bend3	5	Round	5	5
Bend4	5	Square	4	4

11. (Optional) In the **Thickness** group, click **Reverse Direction** to thicken the sheet metal body in the opposite direction.

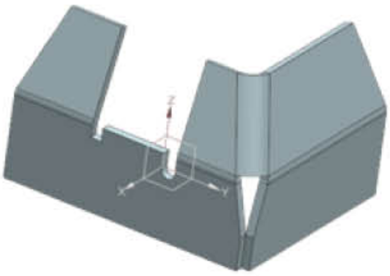
Note:

By default, the sheet metal body is thickened on the outside of the solid model.

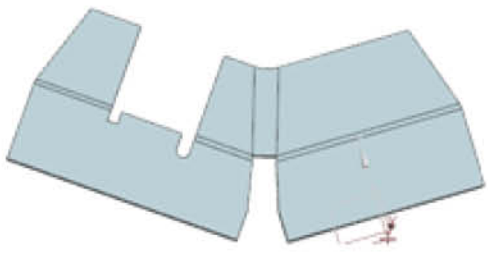
12. In the **Settings** group, select the **Hide Original** check box.

NX hides the parent body and you can view the resultant sheet metal body clearly.

13. Click **OK** to create the sheet metal body.



14. (Optional) Create a flat solid view of the sheet metal body.



2.3.2 Basic sheet metal commands

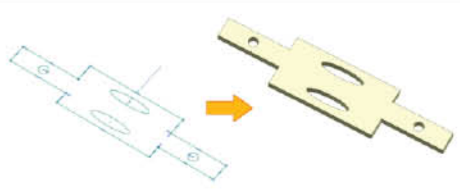
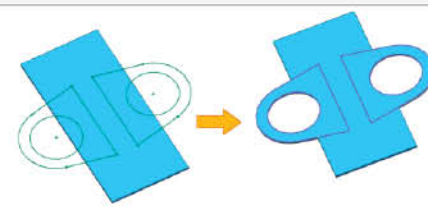
2.3.2.1. Tabs and flanges

2.3.2.1.1 Tab

Use the **Tab** command to create a base feature or a tab by extruding a sketch along a vector by a specified thickness value. You can also use this command to create a secondary tab if the work part includes existing sheet metal features. You can do this by adding material to existing faces and bend regions of an existing sheet metal part.

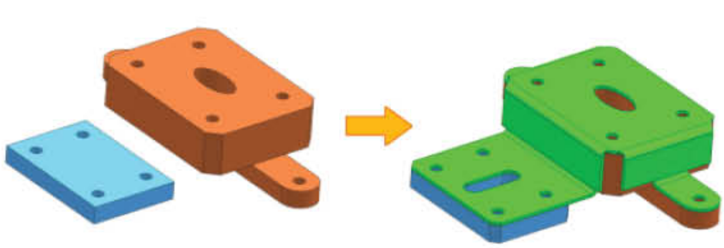
Tab with multiple cutouts

You can create a tab with multiple cutouts by selecting multiple inner closed profiles.

	
Base tab	Secondary tab

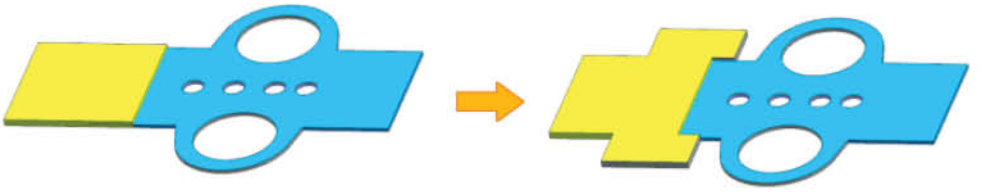
Tab with multiple bends

You can create a tab with multiple bends, such as a mating bracket in an assembly, using the assembly references.



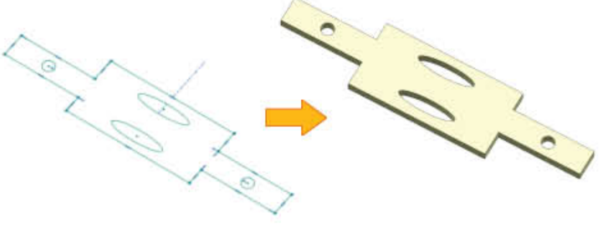
Adding material to an existing sheet metal body

You can add material to a specified sheet metal body when multiple bodies are present in a part file.

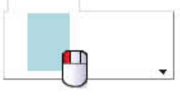


a. Create a base tab with cutouts

This example shows how to create a tab using a section.



1.

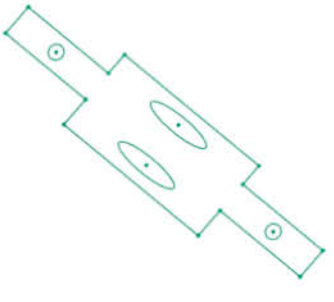


Choose **Home** tab→ **Base** group→ **Tab** .

2.

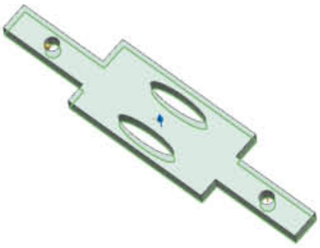
In the **Section** group, click **Sketch Section**  to create the tab section.

3.



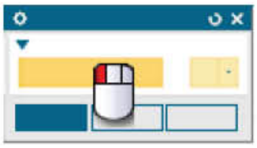
Create the sketch shown.

4.



Finish the sketch.

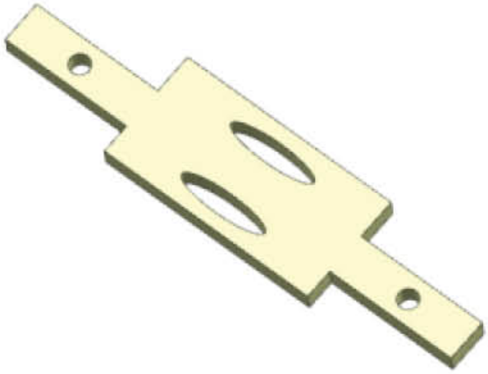
5.



In the **Thickness** group, set the **Thickness** value to **4**.

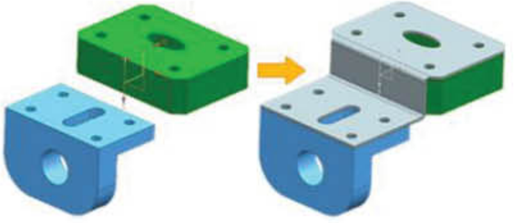
6.

Middle-click to create the tab.

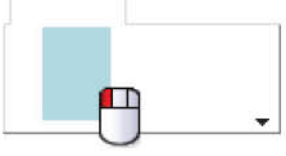


b. Create a multi-bend tab

This example shows how to create a mating bracket with multiple bends.

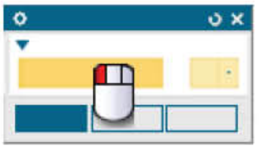


1.



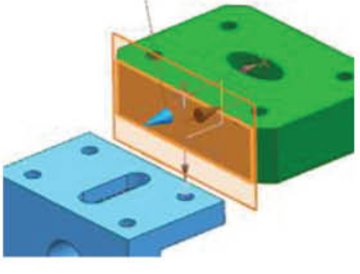
Choose **Home** tab→ **Base** group→ **Tab** .

2.



In the **Multi-Bend References** group, click **Specify Plane**.

3.



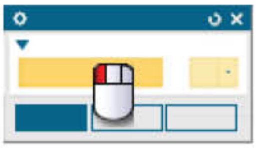
Note:

If the plane direction is not as shown, double-click the arrow to reverse it.
Select the face shown to specify the first reference plane.

4.

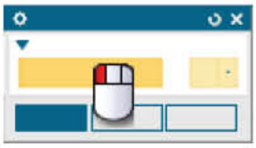
From the **Inset** list, select **Material Outside** .

5.



Click **Add New Set** to add a new plane to the **List** box.

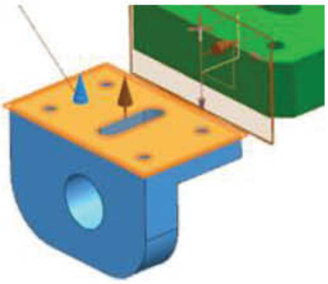
6.



From the **Parent** list, select **Plane1**.

In the **List** box for the **New** plane, the **Parent** changes from **Sketch Plane** to **Plane1**.

7.

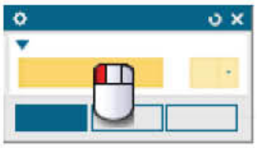


Select the face shown to specify the second reference plane.

8.

From the **Inset** list, select **Material Inside**.

9.

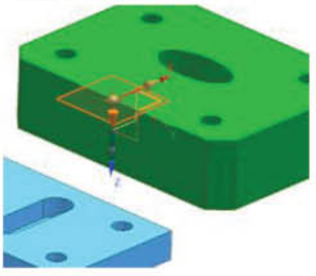


In the **Section** group, click **Sketch Section**.

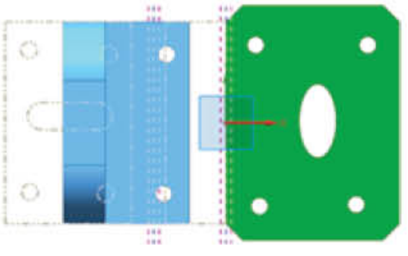
10.

Note:

Make sure that the sketch plane normal direction and the bend direction are the same. In this example, both the sketch plane and the Z-axis of the CSYS (which indicates the bend direction) are pointing downwards.



Select the XY plane of the datum coordinate system and click **OK** in the **Create Sketch** dialog box.

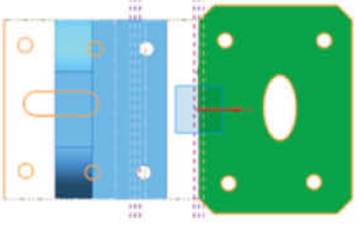


The sketch plane reorients and NX displays the bend center and bend tangent curves in infinite lines.

11.

Tip:

The selected curves are highlighted in the following graphic.
Choose **Home** tab→ **Include** group→ **Project Curve** and select the curves as shown to project onto the sketch plane.



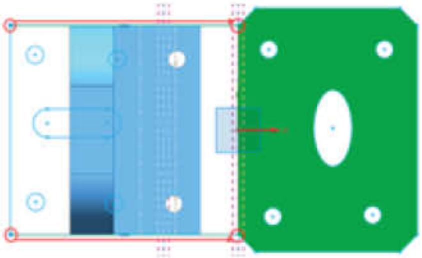
12.

Click **OK** in the **Project Curve** dialog box.

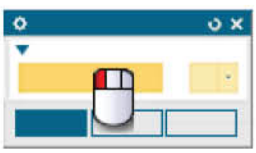
13.

Note:

Make sure that you close the open ends of the sketch and create a closed loop.
Choose **Home** tab→ **Curve** group→ **Line** and create two lines to complete the sketch as shown.



14.



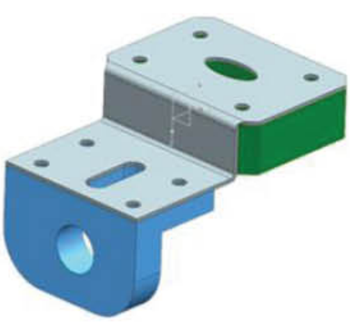
Right-click and choose **Finish Sketch** .

15.

In the **Thickness** group, click **Reverse Direction** .

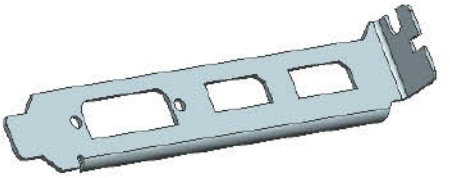
16.

Click **OK** to create the multi-bend bracket.

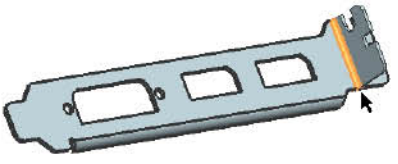


c. Create a secondary tab across a bend

To create a secondary tab across a bend region, you need to flatten the bend area, create the tab, then rebend the unbent region. This example uses a part from a computer video card.



1. Choose **Home** tab→ **Bend** group→ **Unbend** .
2. Select the flat face of the tab as the stationary face, then select the bend.

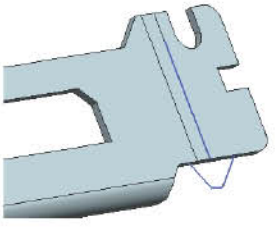


3. Click **OK**.

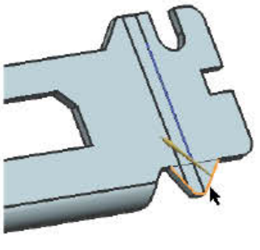
4. Choose **Home** tab→ **Base** group→ **Tab** .

   In the **Type** group, the **Secondary** type is selected by default.

5. Click **Sketch Section**  and sketch the profile of the tab you want to add.

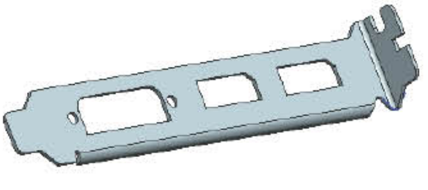


6. Select the sketch.



7. Click **OK**.

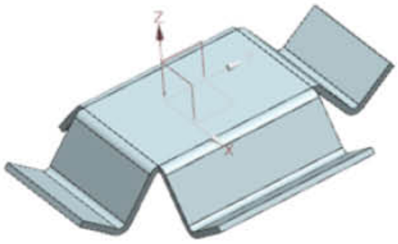
8. Use the **Rebend**  command to return the part to its original shape.



2.3.2.1.2 Flange

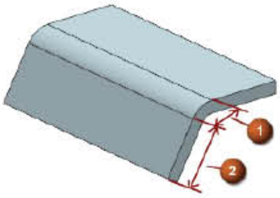
Use the **Flange** command to create a single flange instance or multiple instances by selecting a single or multiple base edges.

NX creates multiple flange instances in a single flange feature.



Note:

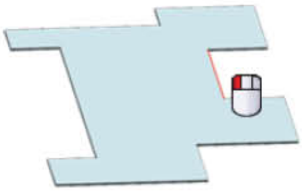
- NX successfully creates the flange feature, even if some of the flange instances in a set fail.
- A flange consists of a bend region (1) and a web (2). You can add a web at an angle to a planar face and add a bend between the two.



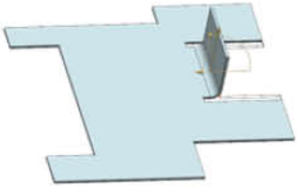
a. Create a flange

This example shows how to create a flange.

1. Choose **Home** tab→ **Base** group→ **Flange** .
2. Select the edge to which you want to add a flange.



A preview of the flange appears.



Tip:

If required, click **Reverse Direction**  to reverse the direction of the flange.

3. In the **Width** group, from the **Width Option** list, select **At Center**.

4. In the **Flange Properties** group, set the flange properties.

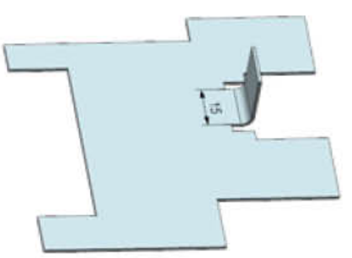
   For this example:

   - **Width = 15**
   - **Length = 20**
   - **Angle = 90**
   - **Length Reference = Inside**
   - **Inset = Material Inside**
5. In the **Bend Parameters** group, specify values for the bend radius and the neutral factor.
6. In the **Relief** group, set the following.

   For this example:

   - **Bend Relief = Square**
   - **Depth = 3**
   - **Width = 3**
   - **Corner Relief = Bend Only**
   - **Extend Relief =** ☑
   - **Include Relief In Width =** ☐
7. (Optional) In the **Geometry Properties** group, select the **Use Geometry Mirror and Pattern** ☑ check box.
8. Click **OK** or **Apply**.

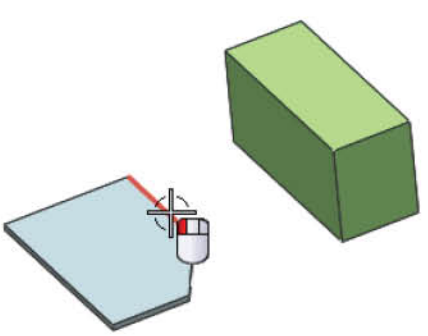
   NX creates the flange.



b. Create a flange matched to a face

This example shows how to create a flange by matching it to the planar face of either the same part or from other part in design context.

1. Choose **Home** tab→ **Base** group→ **Flange** .
2. Select the edge to which you want to add a flange.

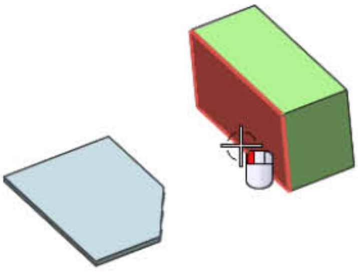


3. In the **Width** group, from the **Width Option** list, select **Full**.
4. In the **Flange Properties** group, set the flange properties.

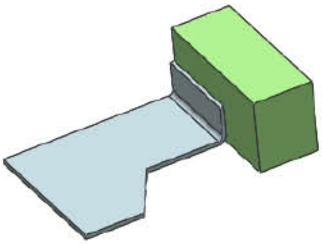
   For this example:

   - **Length = 20**
   - **Match Face = Until Selected**
   - **Length Reference = Inside**
   - **Inset = Material Outside**
5. Click on the planar face of the other part.

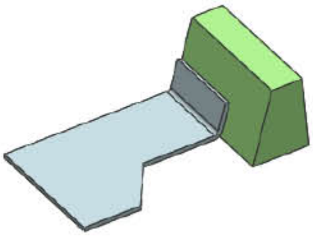
   If the face you select is from another component in an assembly, and you want the flange to update associatively, on the Top Border bar, turn on **Create Interpart Link**  before selecting the face.



6. Click **OK** to create the flange.



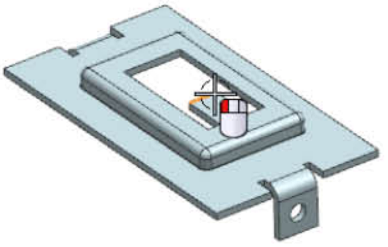
The flange is associative to the planar face. If the face changes, the flange changes with it.



c. Create a flange on the edge of a deformed face

This example shows how to create a flange on the edge of a deformed face of a dimple feature. The dimple feature is of the closed section type and the cutout is created on the face parallel to the planar face on which the dimple feature is created.

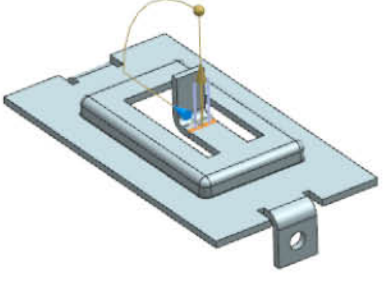
1. Choose **Home** tab→ **Base** group→ **Flange** .
2. Select the edge to which you want to add a flange.



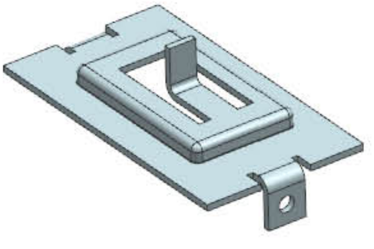
3. In the **Width** group, from the **Width Option** list, select **Full**.
4. In the **Flange Properties** group, set the flange properties.

   For this example:

   - Flange direction = +ZC
   - **Length = 20**
   - **Match Face = None**
   - **Angle = 90**
   - **Length Reference = Inside**
   - **Inset = Material Inside**
5. In the **Offset** group, enter an offset value of **10**.



6. Click **OK** to create the flange.



d. Create multiple flange instances

You can create multiple flange instances by selecting multiple base edges. The flange instances are created within a single flange feature.

1. Choose **Home** tab→ **Base** group→ **Flange** .
2. Select the edge to which you want to add a flange.
3. In the **Flange Properties** group, set the flange properties.

   For this example:

   - **Width Option = Full**

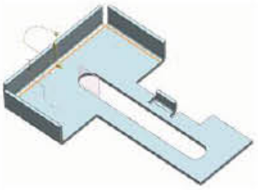
Note:

You can create multiple flange instances only when you set the width option to **Full** or **At Center**.

- **Length = 20**
- **Angle = 90**
- **Length Reference = Inside**
- **Inset = Bend Outside**

4. Select one or more edges.

   As you select edges, you can preview the flange instances.

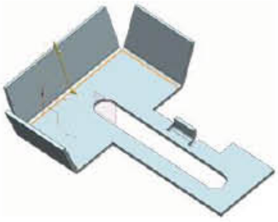


5. Change the flange properties.

   For this example:

   - **Length = 40**
   - **Angle = 70**

   The flange instances, as well as the **Length** and **Angle** columns under the **List** group, reflect the changed values.



6. Click **Add New Set** ⊕.

   You can add a new set of edges with unique feature parameters to create another set of flange instances.

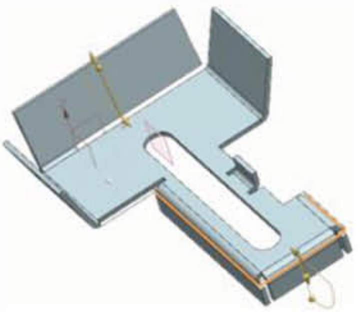
7. Select one or more edges.

   As you select edges, you can preview the flange instances.

8. In the **Flange Properties** group, set the flange properties.

   For this example:

   - **Width Option = Full**
   - **Length = 16**
   - **Angle = 90**
   - **Length Reference = Inside**
   - **Inset = Material Inside**



Note:

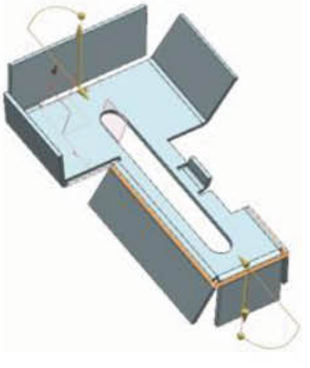
The values of the first set remain unchanged. Each set of edges has unique feature parameters.

9. Change the flange properties for the second set of flange instances.

   For this example:

   - **Length = 35**
   - **Angle = 75**

   The flange instances, as well as the **Length** and **Angle** columns under the **List** group, reflect the changed values.

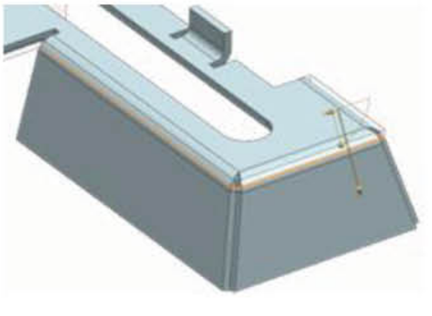


Note:

If there is an interference between flanges on adjacent edges within a set, you can avoid the interference by applying a miter.

10. In the **Geometry Properties** group, select the **Miter** ☑ check box.

    NX applies the miter and removes the interference.

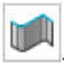


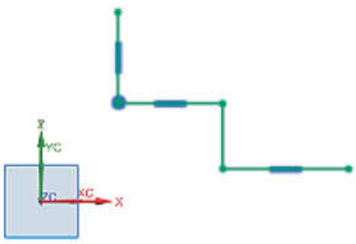
11. Click **OK** to create the flange instances.

    NX creates the flange instances in a single flange feature under the **Model History** node.

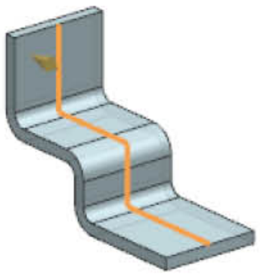
2.3.2.1.2.1 Contour Flange

a. Create a base contour flange

1. Choose **Home** tab→ **Base** group→ **Contour Flange** .
2. In the **Section** group, click **Sketch Section** , and specify the plane on which you want to sketch the section geometry.
3. Sketch the open section.



4. In the **Width** group, from the **Width Option** list, select **Symmetric**.



5. In the **Thickness** group, specify the thickness value.

   For this example, the default thickness of 3 is used.

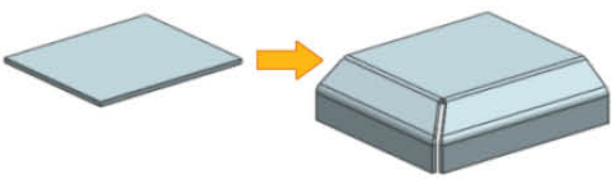
   You can modify the options available in the **Bend Parameters**, **Relief**, **Miter**, and **Corner** groups. For this example, the default values are used.

6. Click **OK**.

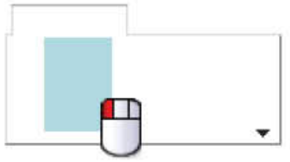
b. Create a secondary contour flange

This example shows how to create a chained secondary contour flange. You will close and miter corners where the two adjacent bends meet. You will also create a gap equal to the sheet metal thickness.

You will start with a base sheet metal tab.

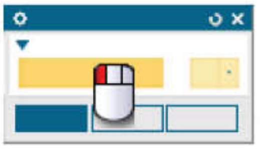


1.



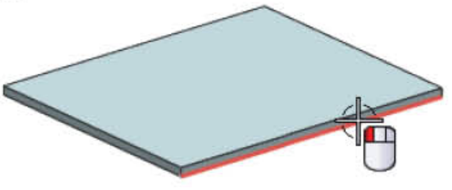
Choose **Home** tab→ **Base** group→ **Contour Flange** .

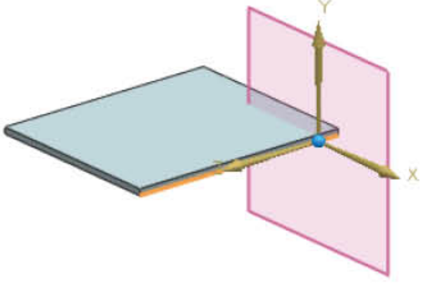
2.



In the **Section** group, click **Sketch Section**  to create the contour flange section.

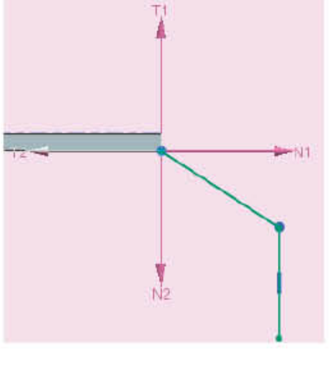
3.





Select the edge of the tab as the sketch path and then middle-click.

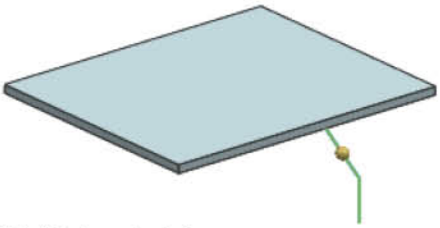
4.



Create the sketch shown.

The start point of the sketch must be on the selected edge.

5.



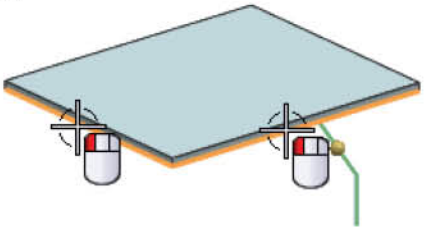
Finish the sketch.

6.

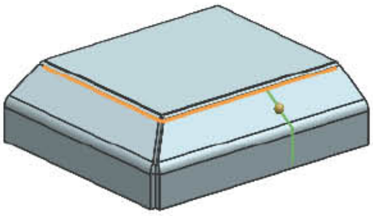
In the **Width** group, do the following:

- **Width Option** = **Chain**
- Click **Select Edge** 

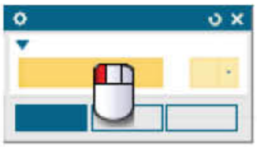
7.

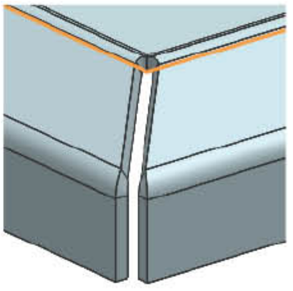


Select the chain of edges on which you want to create the contour flange.



8.





Do the following:

Corner group:

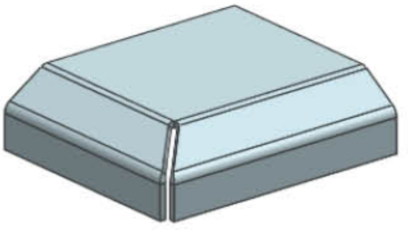
- **Close Corner** = ☑
- **Treatment = Circular Cutout**
- **Miter Corner** = ☑

Corner Setting group:

Gap = Sheet_Metal _Material_Thickness

9.

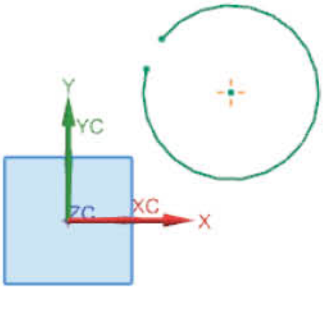
Middle-click to create the contour flange.



c. Create wrapped features

This procedure shows how to use the **Contour Flange** command to construct features that are wrapped around a cylinder. For example, you can use this procedure to create parts that are made by rolling perforated material.

1. Choose **Home** tab→ **Base** group→ **Contour Flange** .
2. In the **Type** group, select **Base**.
3. Sketch an arc that has an included angle of slightly less than 360 degrees.



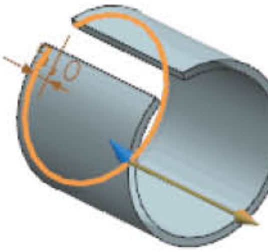
When you create the arc:

- There must be a slight gap where the ends of the rolled material meet, else the part will not unbend.
- The material side and the material thickness of the contour flange must be defined such that it does not close the gap.

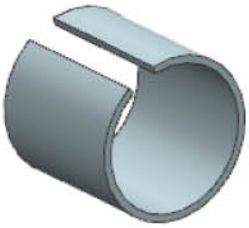
4. In the **Width** group, set the width of the contour flange.

   For this example:

   - **Width Option = Finite**
   - **Width = 60**

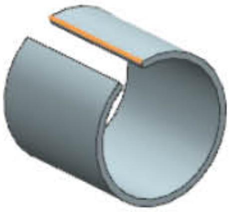


5. Click **OK**.

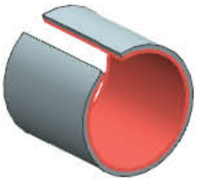


   After you have constructed the contour flange, you can unroll it using the **Unbend** command.

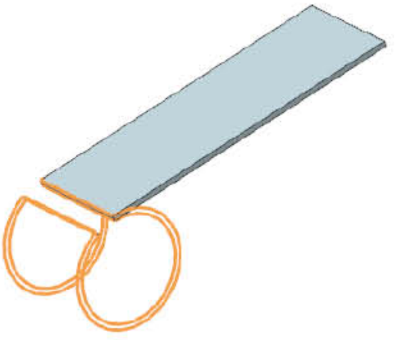
6. Choose **Home** tab→ **Bend** group→ **Unbend** .
7. Select an edge of the part, as the stationary edge.



8. Select the bend as shown in the following graphic.

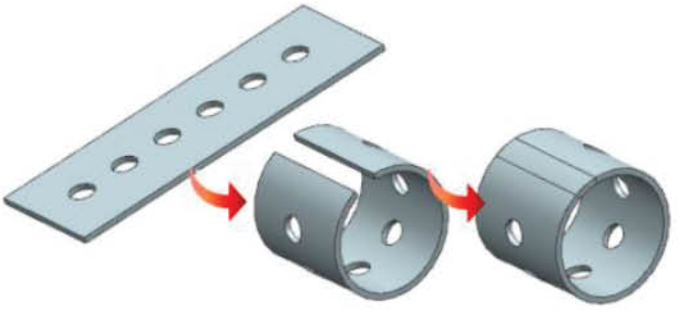


If the **Preview** check box is selected, the unrolled state of the part appears as shown below.



9. Click **OK** to unbend the part.

   You can add features, such as a pattern of holes, to the flattened part, and then use the **Rebend** command to re-roll the part. You can fill the gap by adding an extrude.

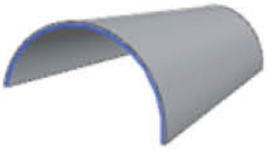


2.3.2.1.2.2 Lofted Flange

Use the **Lofted Flange** command to create a base or secondary feature between two sections where the lofted shape is a linear transition between the sections. The two sections must be open and on parallel reference planes.

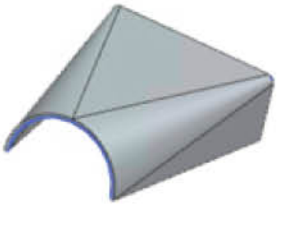
You can:

- Create conical bend regions.



- Create transition regions between two different sections.

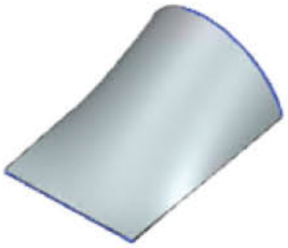
  The following example shows a round to square transition that is used in HVAC technology.



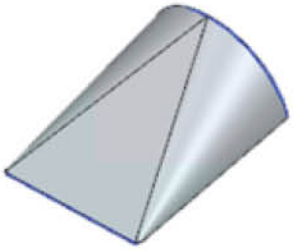
You can create a lofted flange by using bend segments to generate a flange surface that has a constant bend radius. This flange is developable.

You can use different bending methods to create the following:

- Non-planar, non-cylindrical, and non-conical faces.

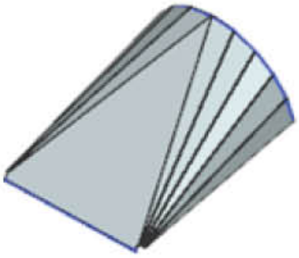


- Planar, cylindrical, and conical faces.



- Planar faces and cylindrical bends.

  When you create a lofted flange that has a conical region or a cylindrical region, NX triangulates that region to planar faces and cylindrical bends.

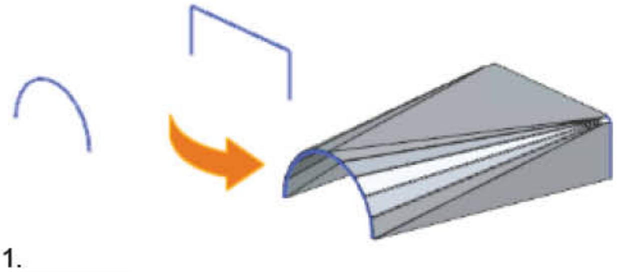


When you create this type of a flange, you can:

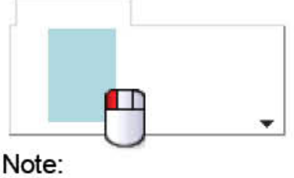
- Create a relief on the bend segments. When you use a relief, NX trims the interfering bends and helps to easily flatten the resulting model. You can specify a linear or a spherical relief.
- Specify how each bend can be divided using the following parameters.
  - **Bend segments**
  - **Chord height**
  - **Segment length**
  - **Segment angle**

Create a lofted flange

This example shows how to create a transition from a square section to a round section by creating a lofted flange between two sections.



1.



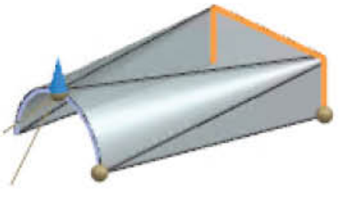
Note:
If there are no sheet metal features in the part, a base lofted flange is created by default.

Choose **Home** tab→**Base** group→**Lofted Flange** .

2.

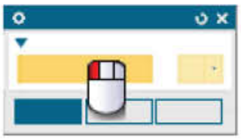
In the **Start Section** group, select the start curve of the flange that you want to create.

3.



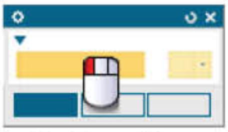
In the **End Section** group, click **Select Curve**, and then select the end curve of the flange that you want to create.

4.



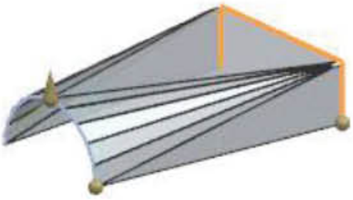
In the **Thickness** group, accept the default thickness.

5.

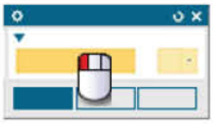


In the **Bend Segments** group, set the following.

- **Bending Method = Bends**
- **Divide Parameter = Bend Segments**
- **Number of Bend Segments = 10**

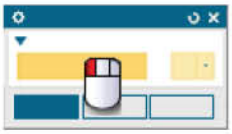


6.



In the **Bend Parameters** group, accept the default values of the **Bend Radius** and **Neutral Factor**.

7.

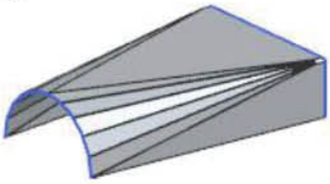


In the **Relief** group, set the following.

For this example:

- **Auto Relief = Linear** 
- **Trim End Plates =** 

8.



Click **OK**.

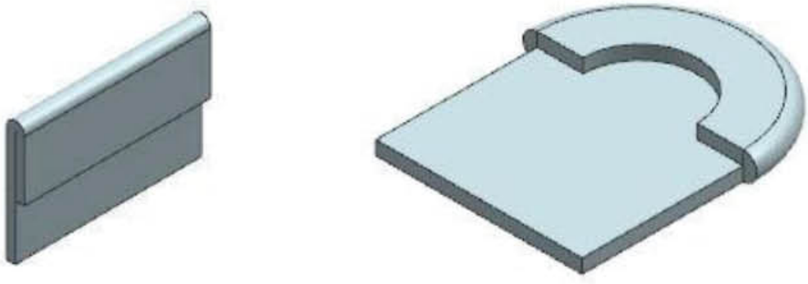
2.3.2.1.2.3 Hem Flange

Use the **Hem Flange** command to modify an edge of a sheet metal part by folding it onto itself for safe handling or to increase the stiffness of the edge.

The hem flange is always created as a secondary feature on a base part.

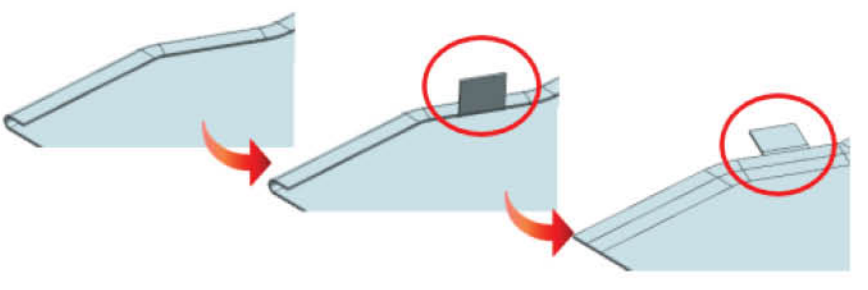
You can create hem flanges on:

- Linear edges and curved edges of a part.



- Connected curves that include a curved edge. You can also create a flange on the linear edge of such hem flanges.

  In the following example, a flange is created on the linear edge of a hem flange, and then both the flanges are flattened.



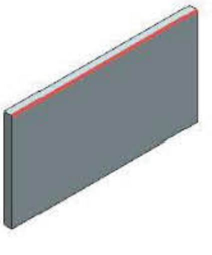
Create a hem flange

This example shows how to create an S-type hem flange on a tab .

1. Choose **Home** tab→ **Bend** group→ **Hem Flange** .
2. In the **Type** group, from the list, select **S-Type** .

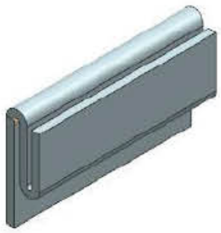
   In the **Edge to Hem** group, the **Select Edge** option is active.

3. Select the edge of the tab on which you want to create the S-type hem flange .



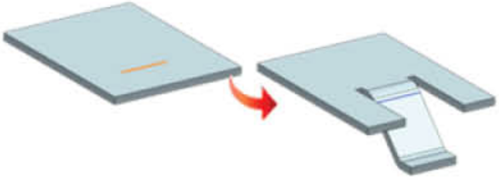
4. In the **Inset Options** group, from the **Inset** list, select **Bend Outside** .
5. In the **Bend Parameters** group, set the following:

   Equal Radii =
   1. Bend Radius = 1
   2. Flange Length = 20
   4. Flange Length = 20
6. In the **Relief** group, set the following.
   - **Bend Relief = Round**
   - **Depth = 3.0000**
   - **Width = 3.0000**
7. Click **OK** to create the S-type hem flange .

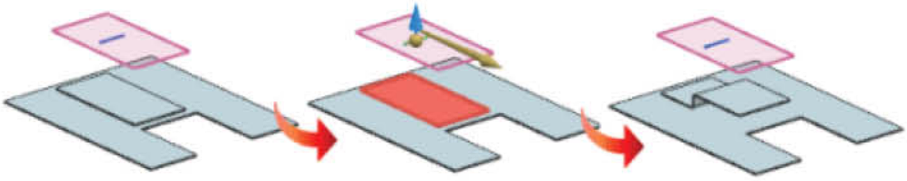


2.3.2.1.2.4 Jog

Use the **Jog** command to create an offset or a step in a tab. You can use small jogs to provide clearance or rigidity to a part. The section for the jog must consist of a single profile that is linear or oblique across a planar face. NX creates the jog at any angle between 0 and 180° by lifting material on one side of the jog line and adding a flange between the two sides.

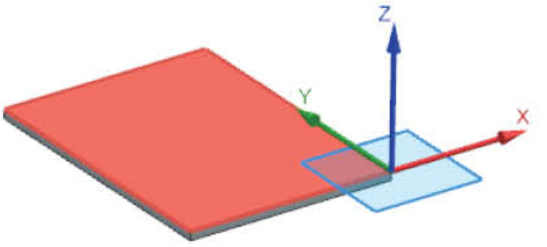


If the part has multiple sheet metal bodies, you can specify the target face on which you want to create the jog.

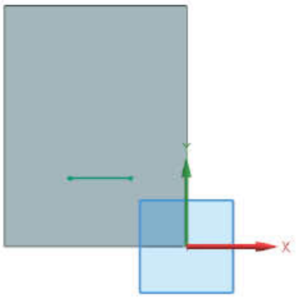


Create a Jog on a tab

1. Choose **Home** tab→ **Bend** group→ **Jog** .
2. Select the top face of the tab to create a section for the jog.



a. Sketch a line for the jog section as shown.



3. In the **Jog Properties** group, set the following:
   a. **Height = 20**
   b. **Angle = 45**

      Note:

      You can specify an angle between 0 and 180°. Ensure that the angle you specify supports the creation of an offset web.
   c. **Height Reference = Outside**
   d. **Inset = Material Inside**
   e. **Extend Section =** ☐
4. In the **Bend Parameters** group, accept the default values for **Bend Radius** and **Neutral Factor**.
5. In the **Relief** group, set the following:
   a. **Bend Relief = Square**
   b. **Depth = 3.0000**
   c. **Width = 3.0000**
   d. **Corner Relief = Bend Only**
6. Click **OK** to create the jog.

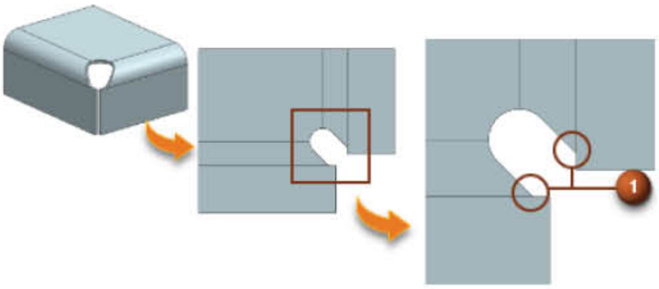
2.3.2.2. Corners

2.3.2.2.1 Closed Corner

Use the **Closed Corner** command to modify two flanges in one operation and to close the corner where the two flanges meet. During flattening, the corner and relief geometry are made appropriate for a flat pattern.

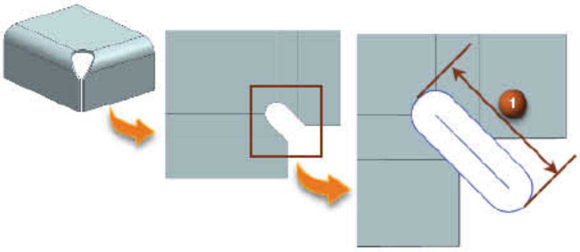
When you use the **Closed Corner** command, you can create:

- Mitered closed corners using the finite cutting methods for the **U Cutout** and **V Cutout** type of corner treatment options.
  - Cutting by limiting the cut up to the bend area. This method is intended for laser cutouts.



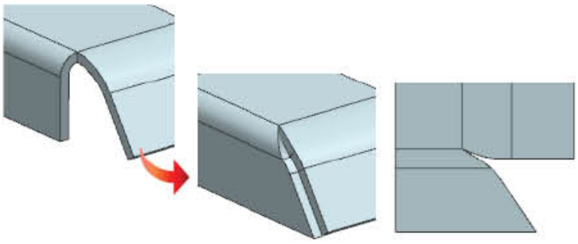
(1) depicts a cut that stops at the boundary of the bend.

- Cutting by controlling the length of the cutout. This method is intended for punching operations.

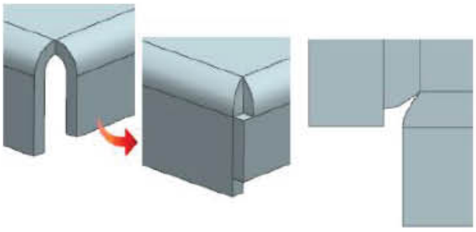


(1) depicts a cut limited by the specified length.

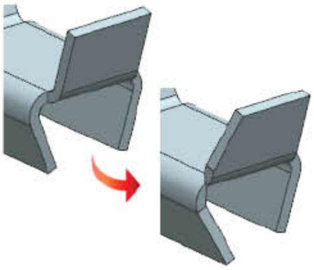
- Mitered closed corners on bends that have unequal bend radii or bend angles.



- Overlapping webs with mitered corners. This is applicable for the **Close and Relief** type of Closed Corner feature.

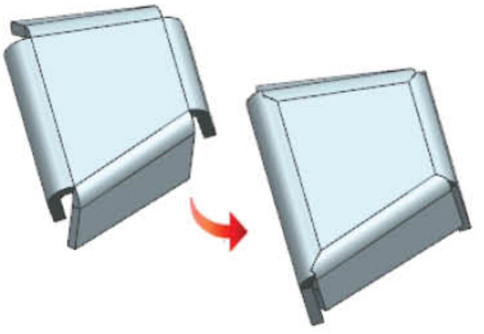


- Corners between flanges that bend in opposite directions.



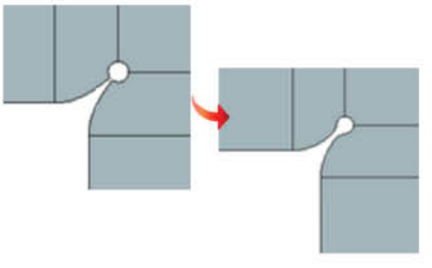
- Corners between a wide range of corner angles.

  The example shows closed corners where all the corner angles have different values.

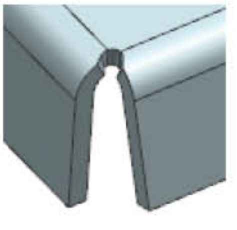


- Mitered closed corners with a smooth transition from the miter to the cutout edges for the **Circular Cutout**, **U Cutout**, **V Cutout**, or **Rectangular Cutout** type of corner treatment options.

  This smooth transition ensures that any jerks or jitters while tracing the tool path are avoided.

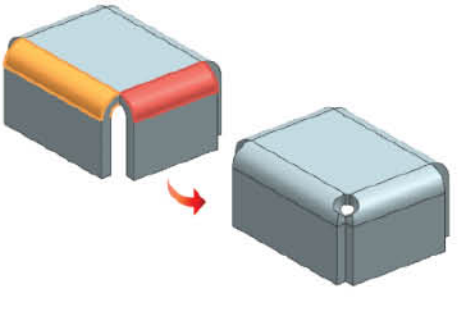


- Reliefs without creating a corner.



a. **Close a corner across two adjacent bends**

This example shows how to close a corner between two bends. The closed corner has a circular cutout type of corner relief treatment.



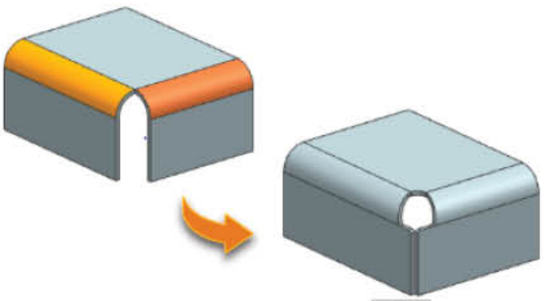
1. Choose **Home** tab→ **Corner** group→ **Closed Corner** .
2. From the **Type** list, select **Corner and Relief**.
3. Select the bends adjacent to the corner you want to close.
4. In the **Corner Properties** group, specify the corner parameters.

   For this example:

   - **Treatment = Circular Cutout**
   - **Overlap = Closed**
   - **Gap** = 0
   - **Miter Corner** = ☑
   - **Blend Miter** = ☑
5. In the **Relief Properties** group, specify the relief parameters.
   - **Origin = Bend Center**
   - **Diameter** = 5
   - **Offset** = 2.5
6. Click **OK** to close the corner.

b. Close a corner across two adjacent bends using a finite cutting method

This example shows how to close a corner between two bends using a finite cutting method. The closed corner has a V cutout type of corner relief treatment.



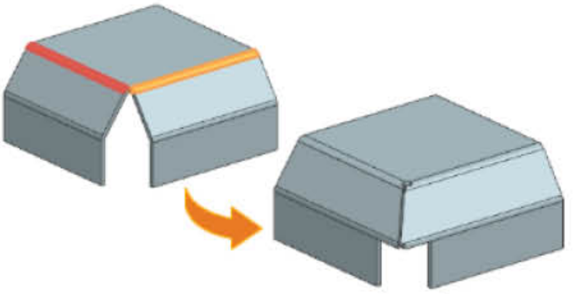
1. Choose **Home** tab→ **Corner** group→ **Closed Corner** .
2. From the **Type** list, select **Corner and Relief**.
3. Select the bends adjacent to the corner that you want to close.
4. In the **Corner Properties** group, specify the parameters for the corner.

   For this example:

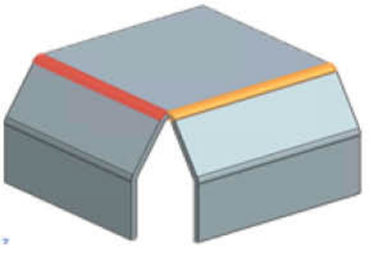
   - **Treatment = V Cutout**
   - **Overlap = Closed**
   - **Gap** = 1
   - **Miter Corner** = ☑
   - **Blend Miter** = ☑
   - **Blend Miter Radius = 5**
5. In the **Relief Properties** group, specify the parameters for creating the relief.
   - **Cut Method = By Path**
   - **Origin = Corner Point**
   - **Diameter = 15**
   - **Offset = 0**
   - **Angle 1 = 25**
   - **Angle 2 = 20**
6. Click **OK** to close the corner.

c. Close a corner by limiting miter propagation

This example shows how to close a corner by limiting miter propagation.



1. Choose **Home** tab→ **Corner** group→ **Closed Corner** .
2. From the type list, select **Corner and Relief**.
3. Select the bends adjacent to the corner that you want to close.

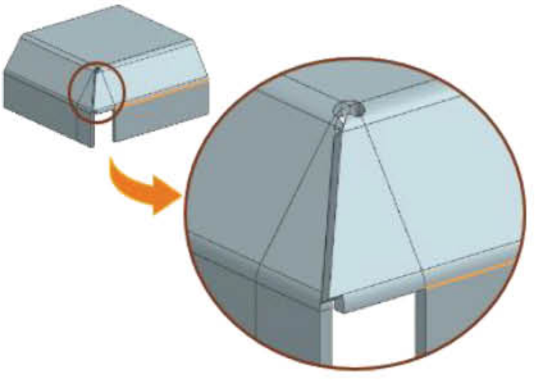


4. In the **Corner Properties** group, specify the parameters for the corner.

   For this example:

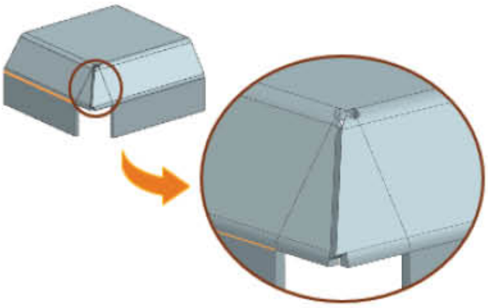
   - **Treatment = Circular Cutout**
   - **Overlap = Side 1**
   - **Gap** = 1
   - **Miter Corner** = ☑
   - **Blend Miter** = ☑
   - **Blend Miter Radius = 0.5**
5. In the **Miter Limits** subgroup, select the first edge to limit the miter.

   NX creates the miter and stops at the first selected edge.



6. (Optional): In the **Miter Limits** subgroup, select the second edge to limit the miter.

NX creates the miter and stops at the second selected edge.



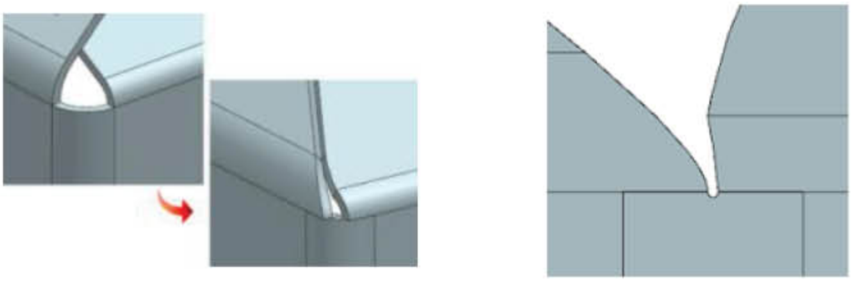
7. In the **Relief Properties** group, specify the parameters for creating the relief.
    - **Origin = Corner Point**
    - **Diameter = 5**
    - **Offset = 0**
8. Click **OK**.

    The corner is closed and the miter is applied at both the sides.

2.3.2.2.2 Three Bend Corner

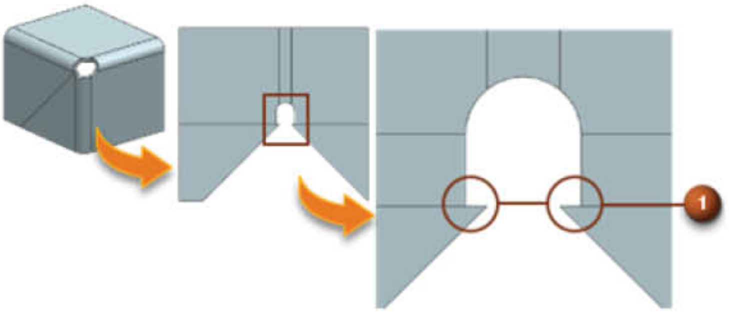
Use the **Three Bend Corner** command to close corners where three adjacent bends with equal or unequal bend radii or bend angles meet. The corner is closed by extending the bends and flanges.

In the example, the three adjacent bends have unequal bend radii and bend angles.



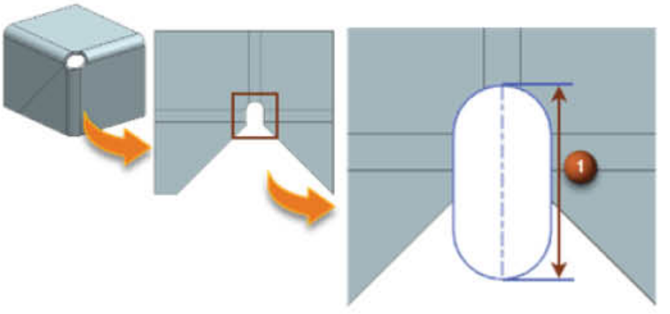
You can:

- Create mitered three bend corners using the finite cutting methods for the **U Cutout** and **V Cutout** type of corner treatment options.
    - Cutting by limiting the cut up to the bend area. This method is intended for laser cutouts.



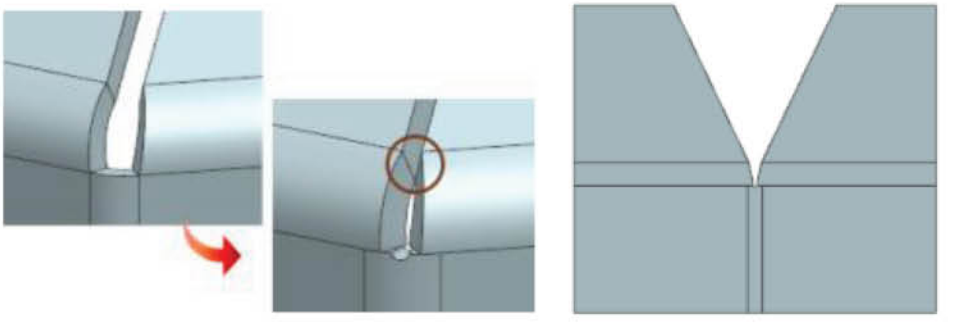
(1) depicts a cut that stops at the boundary of the bend.

- Cutting by controlling the length of the cutout. This method is intended for punching operations.

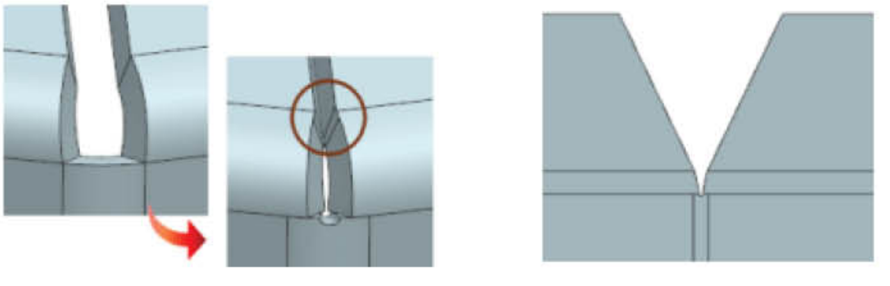


(1) depicts a cut that stops at the boundary of the bend.

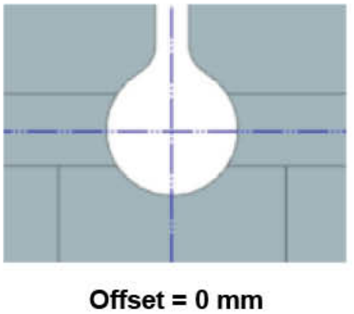
- Create a mitered three bend corner to avoid intersecting flanges.



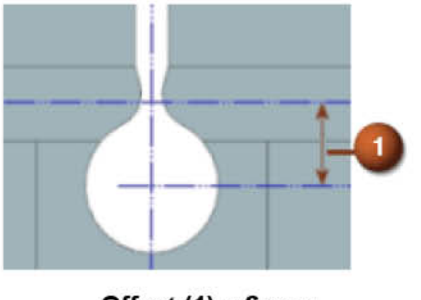
- Blend the mitered edges and create a smooth transition from the mitered edge to the cutout edge to avoid jerks or jitters while tracing the tool path.



- Add offsets to the cutouts that are created at the corner where three adjacent bends meet. You can create the offset from the center of the bend or from the corner point of the bend.



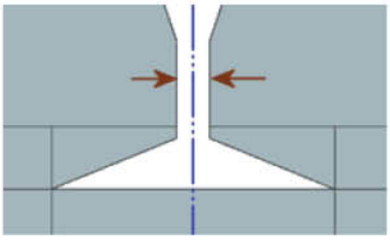
Offset = 0 mm



Offset (1) = 6 mm

- Specify the width of the gap between overlapping flanges in the flattened state by setting a **Flange Clearance** value.

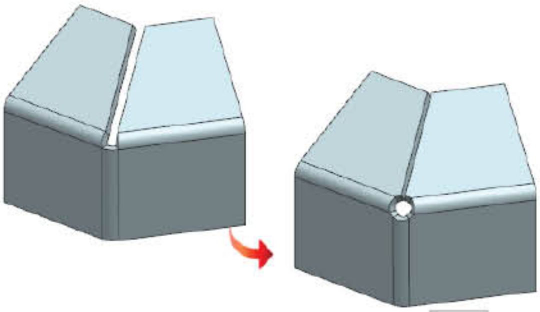
Flange Clearance = 0 mm



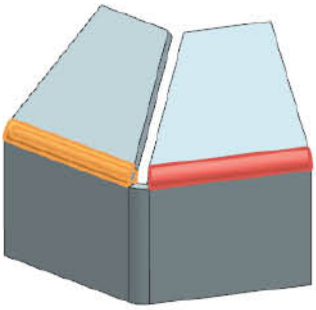
Flange Clearance = 3 mm

a. Close a corner where three adjacent bends meet

This example shows how to close a corner where three adjacent bends that have equal bend radii meet. You will close the corner using the **Circular Cutout** corner treatment option.

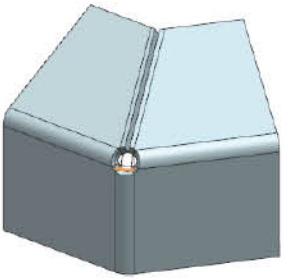


1. Choose **Home** tab→ **Corner** group→ **Three Bend Corner** .
2. Select the bend faces adjacent to the corner.

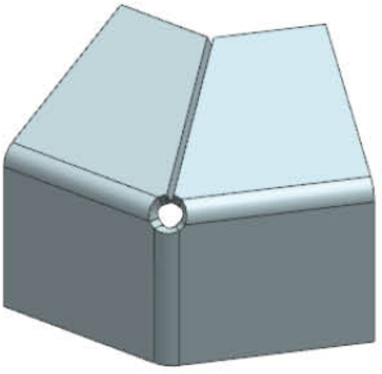


3. In the **Corner Properties** group, from the **Treatment** list, select **Circular Cutout** .
4. In the **Relief Properties** group, specify the diameter of the circular cutout.

   For this example, **Diameter** = **8**.

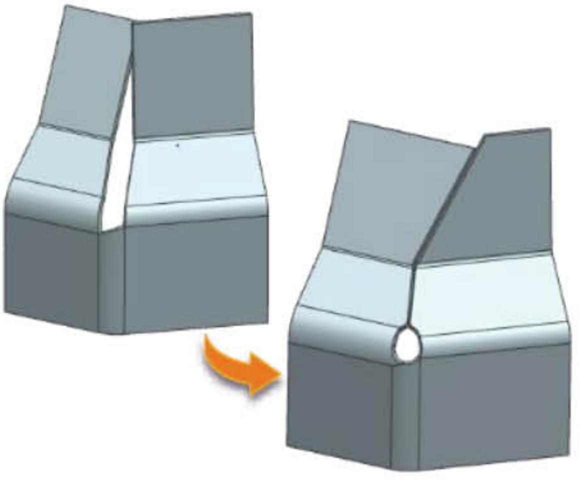


5. Click **OK** to create the Three Bend Corner feature.

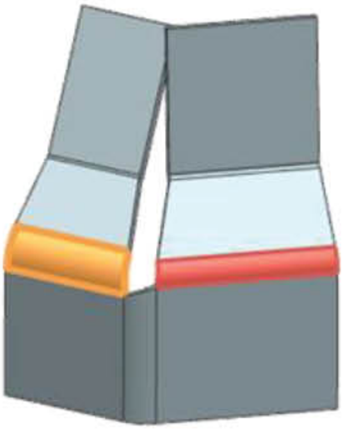


B. Close a three bend corner with mitered edges using a finite cutting method

This example shows how to close a corner where three adjacent bends with unequal bend radii meet. You will create a mitered three bend corner and blend the mitered edges using a finite cutting method to create a smooth transition from the mitered edge to the cutout edge.



1. Choose **Home** tab→ **Corner** group→ **Three Bend Corner** .
2. Select the bend faces that are adjacent to the corner.



3. In the **Corner Properties** group, specify the parameters for the corner.

For this example:

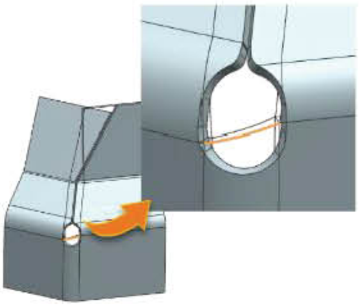
- **Treatment = U Cutout**
- **Miter Corner =** ☑
- **Blend Miter =** ☑
- **Blend Miter Radius = 2.5**

4. In the **Relief Properties** group, specify the parameters for creating the relief.

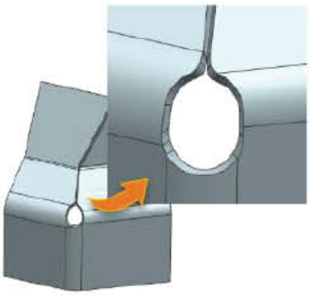
For this example:

- **Diameter = 15**
- **Cut Method = By Tool**
- **Origin = Corner Point**
- **Offset = 0**
- **Length = 20**

5. In the **Settings** group, in the **Flange Clearance** box, enter **2.5**.

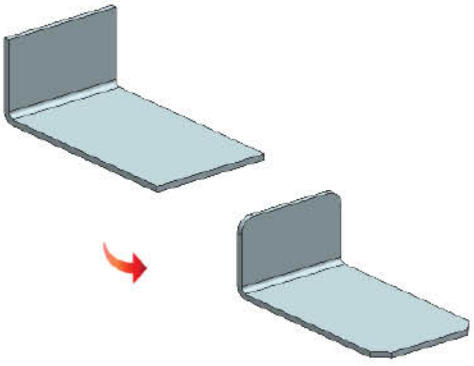


6. Click **OK**.



2.3.2.2.3 Break Corner

Use the **Break Corner** command to round or chamfer sharp corners of a tab or flange.

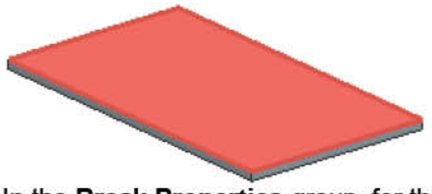


When you create blends on thick edges, to save design time, you can filter out invalid edges.

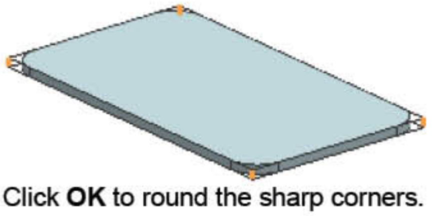
Round a sharp corner on a tab

This example shows how to round the sharp corners of a tab.

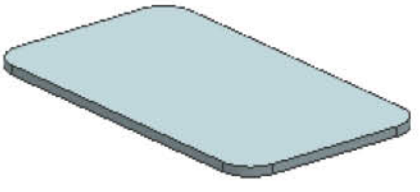
1. Choose **Home** tab→ **Corner** group→ **Break Corner** .
   In the **Edge to Break** group, the **Select Face or Edge** option is active.
2. Select the face of the tab.



3. In the **Break Properties** group, for this example, set the following:
   Method = **Blend**
   Radius = 10



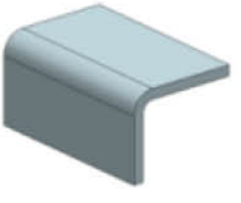
4. Click **OK** to round the sharp corners.



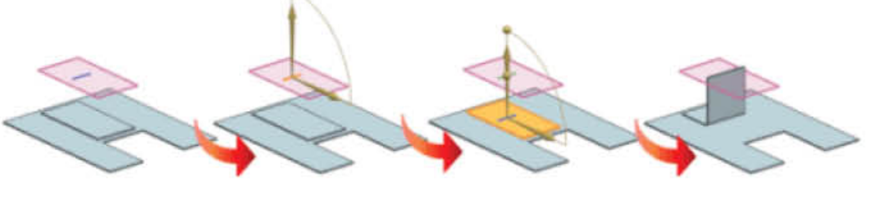
2.3.2.3. Bend, unbend, and rebend
2.3.2.3.1 Bend

Use the **Bend** command to modify the model by bending material on one side of a sketch line, adding a bend between the two sides.

You can create a bend on a planar face of a sheet metal body.

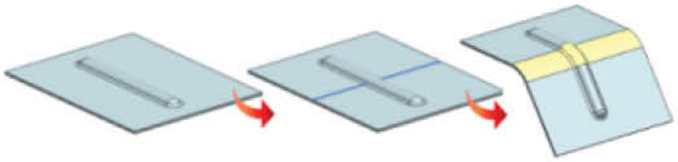


If the part has multiple sheet metal bodies, you can specify the target face on which you want to create the bend.



You can create a bend across existing Bead, Dimple, or Drawn Cutout features. When you create such bends, the bend line must pass completely through the deform features.

In the example, a bend is created across an existing Bead feature.

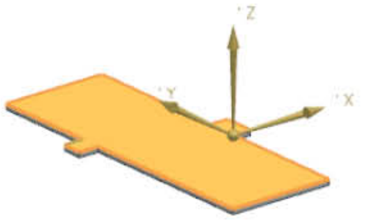


Note To create a cutout on a bend that you want to joggle, you must first joggle the bend before you create the cutout.

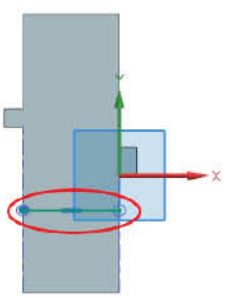
a. Create a bend on a tab

1. Choose **Home** tab→ **Bend** group→ **Bend** .
2. Click **Sketch Section** to specify the plane on which you want to sketch the section geometry.

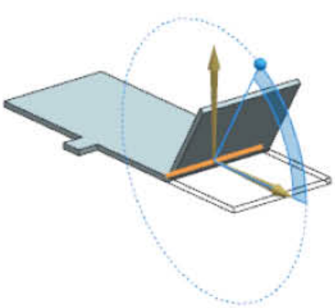
   For this example, the planar face of the Tab feature is selected.



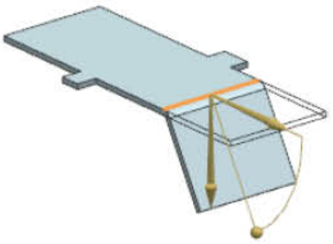
3. Create the sketch to define the bend center line.



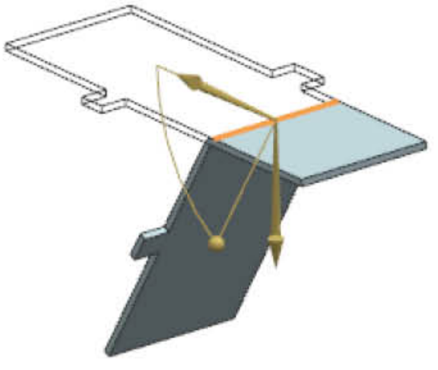
4. Click **Finish** to exit the Sketch.



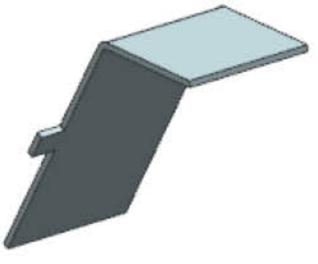
5. In the **Bend Properties** group, set the following:
   a. **Angle = 60**
   b. Click **Reverse Direction** to reverse the bend direction.



   c. Click **Reverse Side** to change the side of the part to move. The direction arrow points towards the portion that will move.



6. Click **OK** to create the bend.



b. Create cuts in bend regions

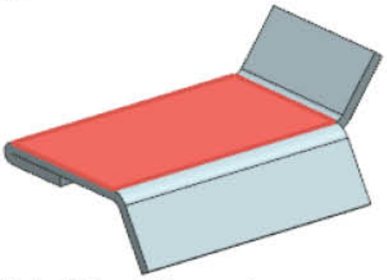
This example shows the typical workflow for making holes or similar cuts in Sheet Metal. You unbend any bend regions that the cuts will intersect, make the cuts, and then rebend the bend areas.

You will use the **Unbend**, **Hole**, and **Rebend** commands.

1.

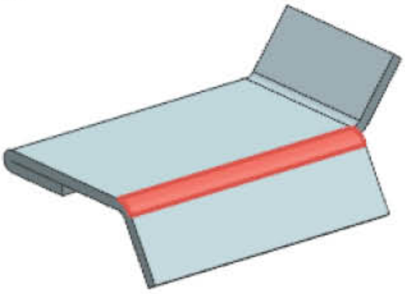
Click **Unbend** .

2.



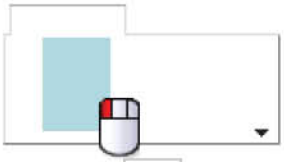
Select the stationary face.

3.

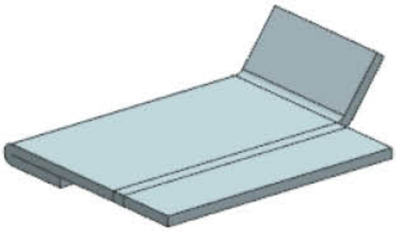


Select the bend region that you want to unbend.

4.

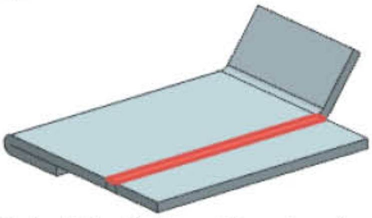


Click **Hole** .



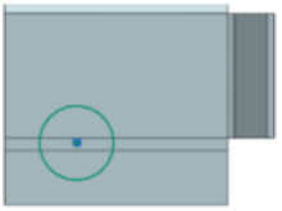
NX unbends the bend and opens the **Hole** dialog box.

5.



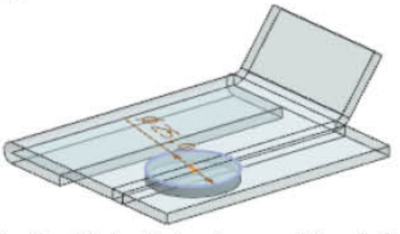
Select the flattened bend region as the sketch placement plane.

6.



In the Sketch, create a circle as shown, and then click **Finish** .

7.

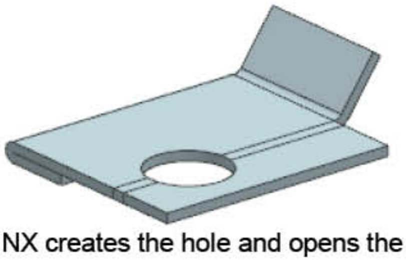


In the **Hole** dialog box, set the following:

- **Type = General Hole**
- **Form and Dimensions** group
  Form = Simple
- **Dimensions** subgroup
  Depth Limit = Through Body

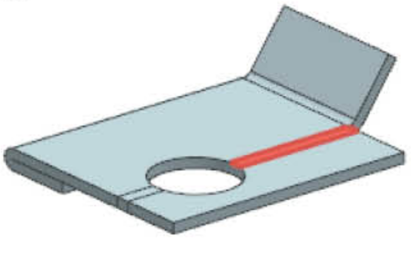
8.

Click **Rebend** .



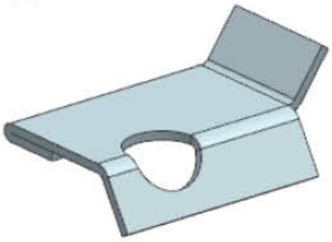
NX creates the hole and opens the **Rebend** dialog box.

9.



Select the bend region that you want to rebend.

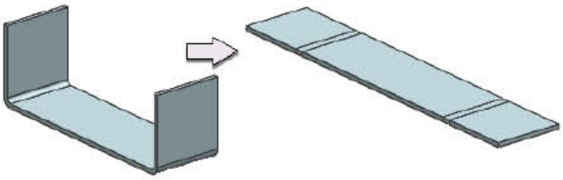
10.



Click **OK** to rebend the part and close the **Rebend** dialog box

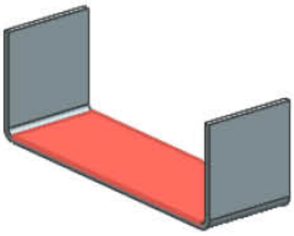
2.3.2.3.2 Unbend

Use the **Unbend** command to unbend a portion of the part to construct cutout features, hole features, or deform features across a bend. You can later reform the bend to represent the true formed state of the model.

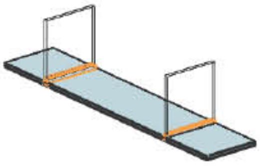


a. Unbend a part

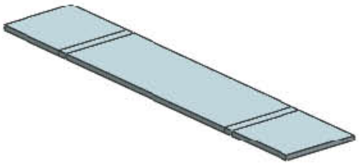
1. Choose **Home** tab→ **Bend** group→ **Unbend** .
2. Select a planar face that remains stationary during the unbending operation.



3. Select the bends you want to unbend.



4. Click **OK** to unbend the part.



Note:

If you create a part that intersects itself at any point during its development, either when creating it or when unbending or rebending, any further unbend or rebend operations may result in an error. If this happens, roll the part back to the step before the self-intersection and try to correct the self-intersecting condition.

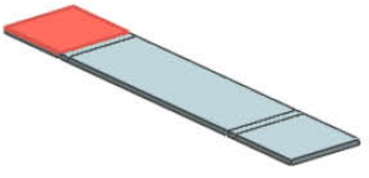
2.3.2.3.3 Rebend

Use the **Rebend** command to return an unbend feature to its previous bent state. Any features added after the unbend operation are properly repositioned during the rebend operation.

a. Rebend a part

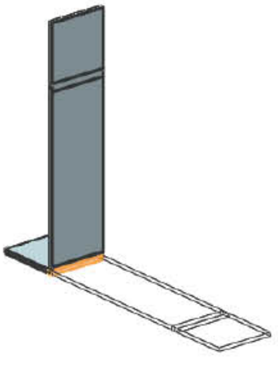
This example shows how to rebend and unbend feature using a stationary face for reference.

1. Choose **Home** tab→ **Bend** group→ **Rebend** .
2. In the **Rebend** dialog box, in the **Stationary Face or Edge** group, click **Select Face or Edge** .
3. In the graphics window, select the face to be retained as the stationary face during the rebend operation.

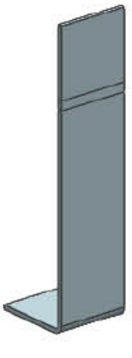


4. Select the unbend feature that you want to rebend.

   A preview of the rebend operation appears.



5. Click **OK** to rebend the selected bend.

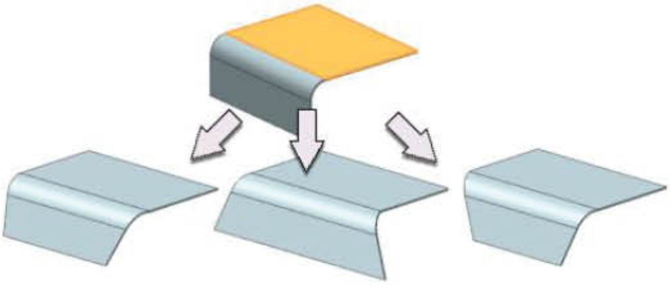


Note:

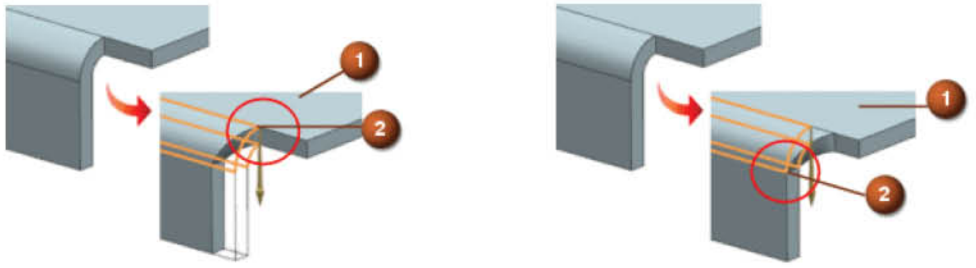
If you create a part that intersects itself at any point during its development, either when creating it or when unbending or rebending, any further unbend or rebend operations may result in an error. If this happens, use the **Edit with Rollback** command to roll the part back to the step before the self-intersection, and try to correct the self-intersecting condition.

2.3.2.4. Bend Taper

Use the **Bend Taper** command to taper one or both sides of the bend or web by adding or removing material.



You can create tapers that start from the bend or the web.



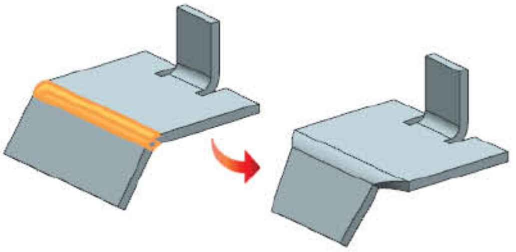
Taper from Bend **Taper from Web**

(1) Stationary face
(2) Start point of the taper

You can create tapers on:

- Flanges, contour flanges, jogs, or any other bend or web.

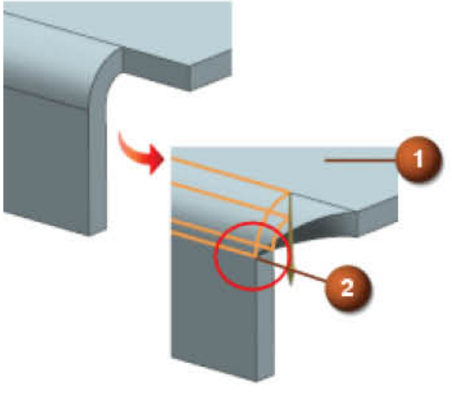
  In the example, the top face is stationary, and a taper is created symmetrically on both sides of the bend.



- Partial width flange.

  You can create tapers that extend up to the end of the base flange or the tab.

In the example, a taper that extends up to the end of the stationary face is created.



1 Stationary face

2 Start point of the taper

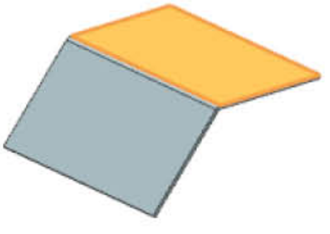
Note:

You cannot use this command to create tapers on non-cylindrical bends.

a. Create a symmetric taper on a bend and the web

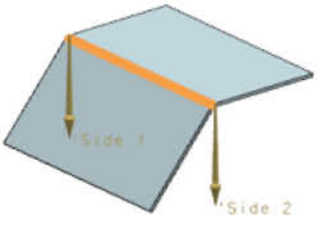
This example shows how to create a symmetric taper on both sides of a selected bend and the web.

1. Choose **Home** tab→ **Corner** group→ **Bend Taper** .
2. Select a face that remains stationary during the taper operation.



You can select a face or an edge. NX uses the selected face or edge to infer the side that remains fixed during the taper operation.

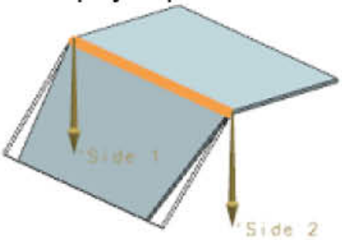
3. Select the bend that you want to taper.



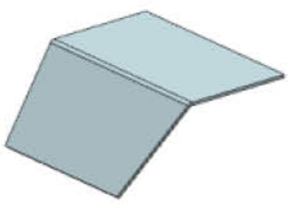
Tip:

The arrows indicate the direction in which the edges of the bend and web are tapered. To create a taper on the other side of the bend, you must change your selection of the stationary face.

4. In the **Taper Properties** group, from the **Taper Sides** list, select **Symmetric**.

5. In the **Bend** subgroup, click **Taper from Bend** .
   NX will create a taper that starts at the bend.
6. Set the following parameters:
   - In the **Bend** subgroup:
     Taper = Linear
     Input Method = Angle
     Taper Angle = 10
   - In the **Web** subgroup:
     Taper = Face
     Taper Angle = 10
7. NX displays a preview of the bend taper feature.
8. 

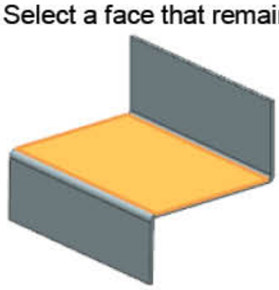
   Click **OK** to create the taper.



b. Create an asymmetric taper on a bend and the web

This example shows how to create an asymmetric taper on either side of a selected bend and the web.

1. Choose **Home** tab→ **Corner** group→ **Bend Taper** .
2. Select a face that remains stationary during the taper operation.

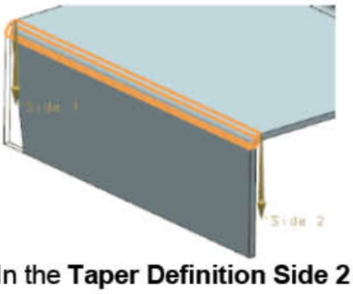


   You can select a face or an edge. NX uses the selected face or edge to infer the side that remains fixed during the taper operation.
3. Select the bend that you want to taper.

Note:
The arrows indicate the direction in which the edges of the bend and web are tapered. To create a taper on the opposite side of the bend, you must change your selection of the stationary face.

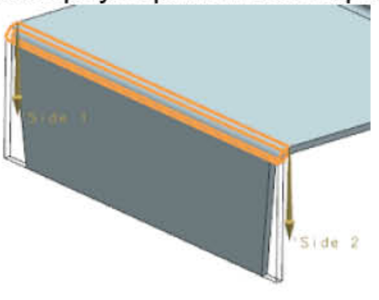
4. In the **Taper Properties** group, from the **Taper Sides** list, select **Both**.
5. In the **Taper Definition Side 1** group, set the taper parameters for side 1.
   For this example:
   - **Bend** subgroup:
     Taper from Bend = ◉
     Taper = **Tangent**
     Taper Distance = **5**
     Infer Radius = ☑
   - **Web** subgroup:
     Taper = **Face**
     Taper Angle = **10**

   NX displays a preview of the taper created on side 1.

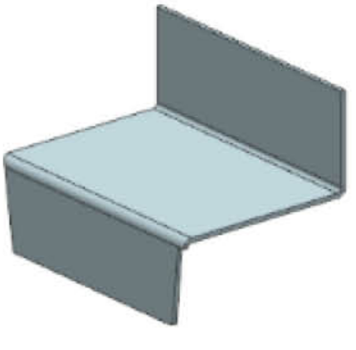


6. In the **Taper Definition Side 2** group, set the taper parameters for side 2.
   For this example:
   - **Taper from Bend** = ◉
   - **Bend** subgroup:
     Taper = **Square**
     Taper Distance = **5**
   - **Web** subgroup:
     Taper = **Face**
     Taper Angle = **5**

   NX displays a preview of the taper created on side 1.

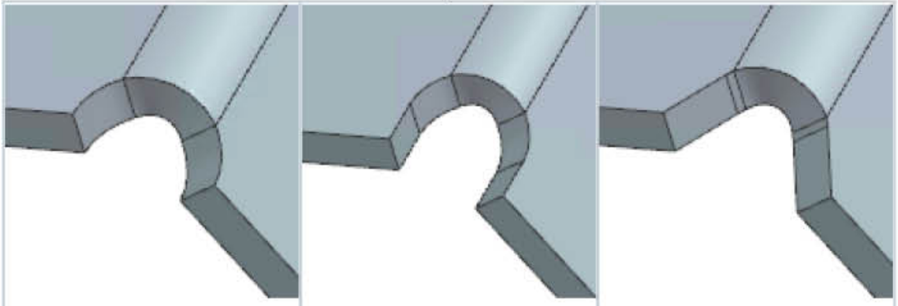


7. Click **OK** to create the taper.



2.3.2.5. Bend Bulge Relief

Use the **Bend Bulge Relief** command to create bend reliefs that let you avoid bulging, deformation, and excessive material collection at specified bends.



Circular relief	U-shaped relief	V-shaped relief

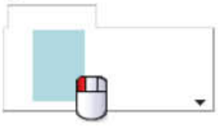
You can create bend bulge reliefs only for:

- Cylindrical bends that are not deformed.
- Bend edges that are not connected to other bend edges.

Create a bend bulge relief

This example shows how to create a relief at the ends of selected bends to avoid the bulging of bends.

1.

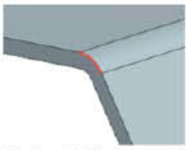


Choose **Home** tab→ **Bend** group→ **Bend Bulge Relief** .

2.

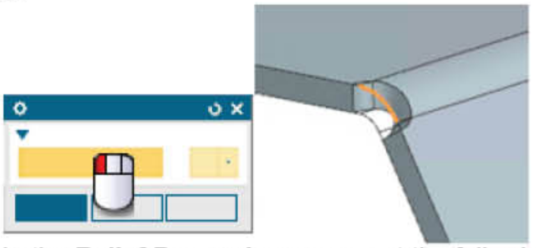
In the **Bend Edge** group, click **Select Edge** .

3.



Select the bend edge where you want to create the relief.

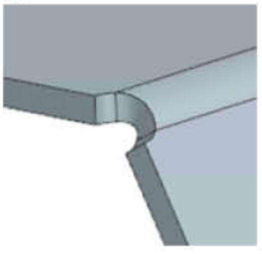
4.



In the **Relief Properties** group, set the following.

- **Relief Type = Circular** 
- **Width = 6**
- **Depth = 3**

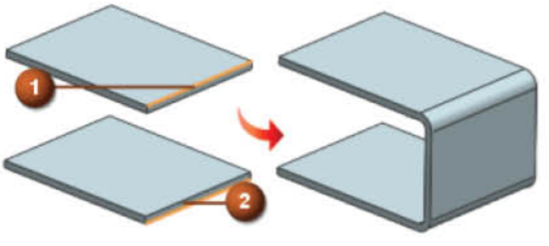
5.



Click **OK**.

2.3.2.6. Bridge Bend

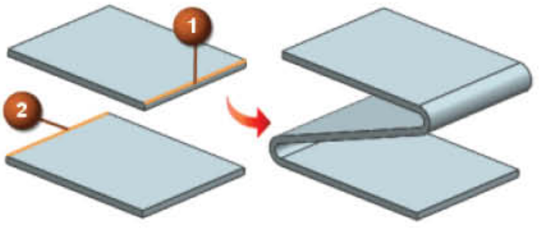
Use the **Bridge Bend** command to create transition geometry by joining two edges on different sheet metal bodies and then uniting the bodies. You can join two unconnected sheet metal bodies by creating a Z-shaped, U-shaped, or fold type transition between them.



U-shaped

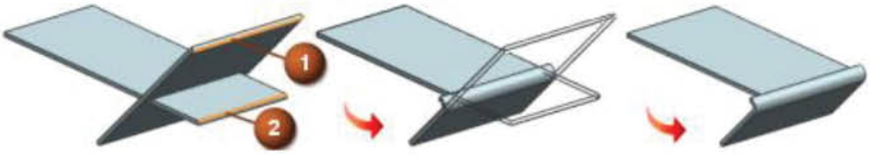
1Start Edge

2End edge



Z-shaped

If the two sheet metal bodies intersect or overlap, you can create fold transitions to join them.

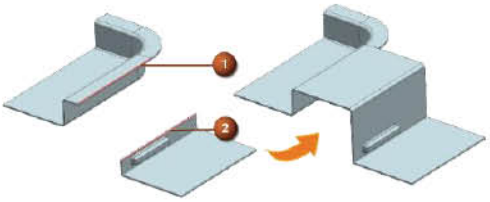


Fold transition

(1) Start edge

(2) End edge

If one of the sheet metal bodies has a chained advanced or chained contour flange, you can select a linear edge as the start edge from the planar web face of the flange and create a fold transition to join the two sheet metal bodies.



Fold transition defined by bend

(1) Start edge

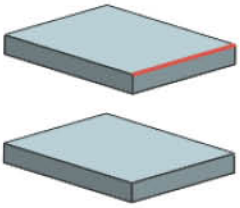
(2) End edge

This command works the same as the **Bridge Transition** command in the

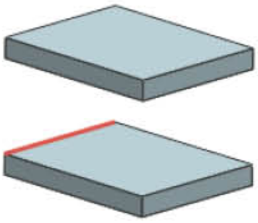
a. Create a bridge bend

This example shows how to join two unconnected sheet metal bodies by creating a Z-shaped bridge bend between them.

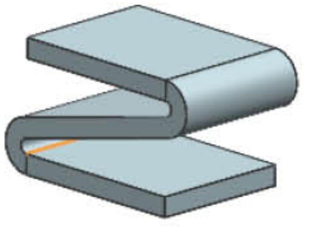
1. Choose **Home** tab→ **Bend** group→ **Bridge Bend** .
2. In the **Bridge Bend** dialog box, from the **Type** list, select **Z or U Transition** .
3. Specify the start edge for the transition.



4. Specify the end edge for the transition.



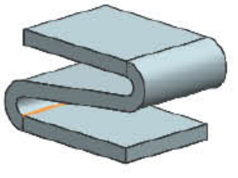
5. In the **Width** group, from the **Width Option** list, select **Full Both Edges** .



6. In the **Bend Properties** group, select the **Start and End Parameters Equal** check box.
7. In the **Start Edge** subgroup, under **Bend Parameters**, in the **Bend Radius** row, click **Launch the formula editor** , and select **Use Local Value**.

   This option overrides the sheet metal preferences and uses the bend radius value that you enter in the dialog box.

8. In the **Bend Radius** box, specify the radius of the bend and press Enter.

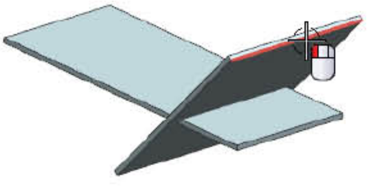


9. Click **OK**.

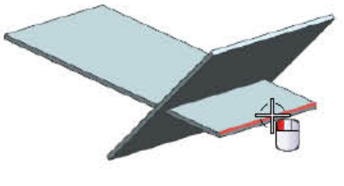
b. Create transitions between intersecting sheet metal bodies

This example shows how to create a fold transition between two intersecting sheet metal bodies.

1. Choose **Home** tab→ **Bend** group→ **Bridge Bend** .
2. In the **Bridge Bend** dialog box, from the **Type** list, select **Fold Transition** .
3. Specify the start edge for the transition bend.

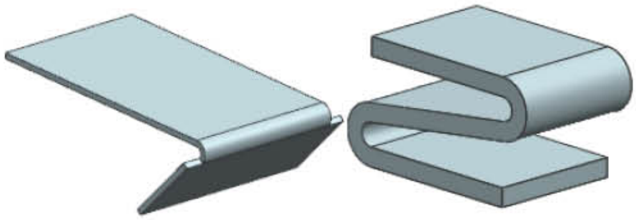


4. Specify the end edge for the transition bend.



5. To trim the bodies and join them with a bend, do the following:
   - In the **Bend Properties** group, from the **Define By** list, select **Bend**.
   - In the **Width** group, select the **Trim or Extend to Bend Tangent** check box.
6. Click **OK**.

   NX trims the bodies and creates a bend between the two intersecting bodies.



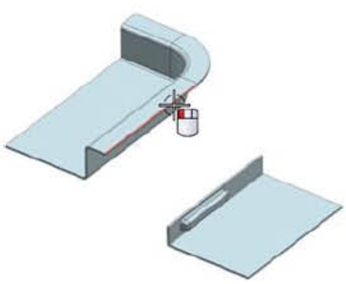
c. Create a fold transition between planar webs of chained bends

This example shows how to create a fold transition between the planar web faces of chained bends by selecting a linear edge as the start edge from the planar web face of a chained contour flange.

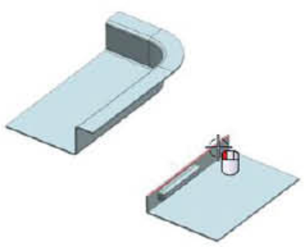
1. Choose **Home** tab→ **Bend** group→ **Bridge Bend** .
2. In the **Bridge Bend** dialog box, from the type list, select **Fold Transition** .
3. Specify the start edge for the transition.

   Note:

   Make sure that you select the start edge from the planar web face of the chained contour flange.



4. Specify the end edge for the transition.

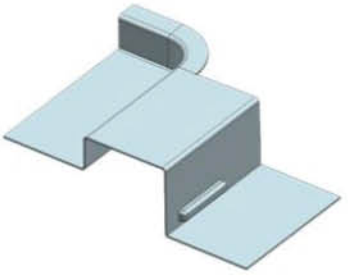


   Note:

   Make sure that you select the end edge from a body that does not contain a complex feature such as a chained advanced or a chained contour flange, or a lightening cutout, or a joggle.
5. In the **Width** group, from the **Width Option** list, select **Full Both Edges** .
6. In the **Bend Properties** group, from the **Define By** list, select **Bend**.

7. Click **OK**.

   NX creates a bend between the two planar web faces.

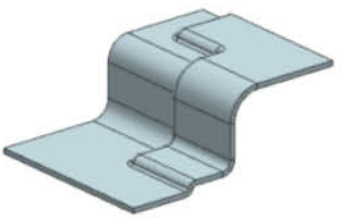


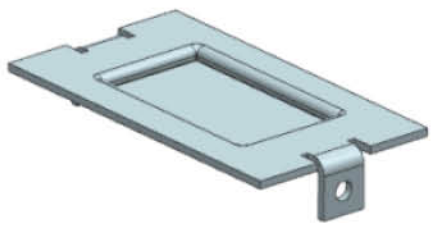
2.3.2.7. Punch

2.3.2.7. 1 Dimple

Use the **Dimple** command to lift an area of the model inside a sketch that simulates a stamping tool. You can create an indentation in a sheet metal part using a sketched section as the dimple outline.

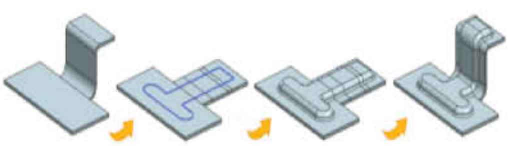
Like the **Drawn Cutout** command, the **Dimple** command makes an indentation in the sheet metal part, using a sketched section as its outline. The difference between the two is that a Dimple feature has a bottom face whereas a Drawn Cutout feature does not.



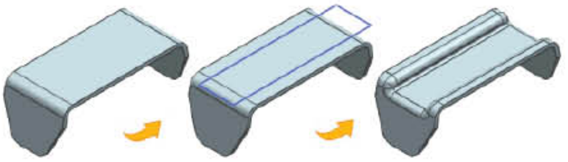


You can create dimples on bends in the folded or flattened state. A dimple created on a bend in the folded state does not wrap around the bend. You can also create dimples on deform features in sheet metal parts.

Dimple on a flattened cylindrical bend



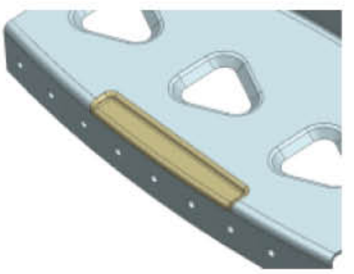
Dimple on a formed cylindrical bend

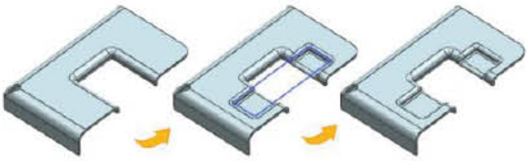


Dimple across deform features

Dimple on a flattened non-cylindrical bend

Dimple on a formed non-cylindrical bend





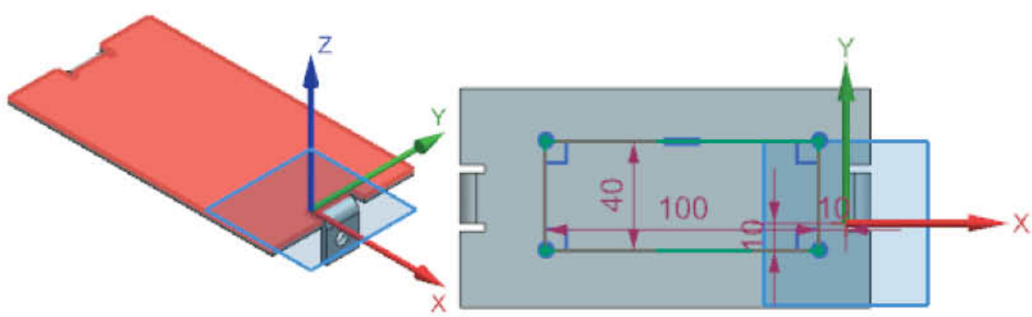
a. Create a dimple

1. Choose **Home** tab→ **Punch** group→ **Dimple** .
2. In the **Section** group, click **Sketch Section**  to create a sketch that defines the section geometry.

   For this example, a sketch is created on the top face of the tab.

   Note:

   You can also create a sketch on the bottom face of the tab. In any case, NX creates the dimple with consistent depth irrespective of the sketch position.



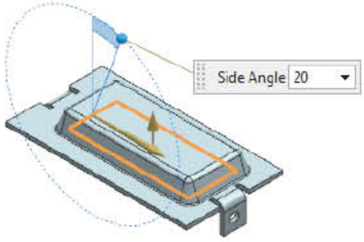
   A closed section must not touch any part edges.

   If you want to use an open section, the open ends must intersect the part edges. If you create an open section where the curves do not intersect the outer edges of the part, NX extends the ends of the section curves tangentially until they either intersect one another or intersect the edges of the part.

3. In the **Dimple Properties** group, specify the depth parameters for the dimple.

   For this example:

   - **Depth = 15**
   - **Side Angle = 20**

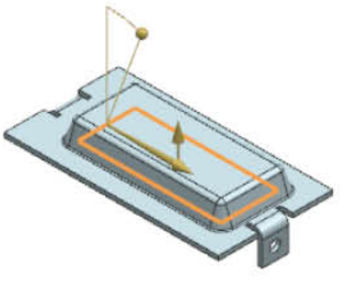


   - **Side Walls = Material Outside**

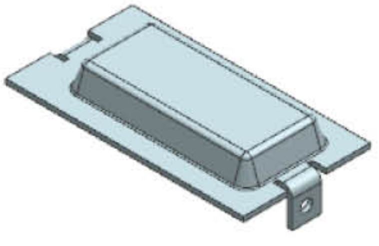
4. In the **Settings** group, specify the parameters to blend the edges of the dimple.

   For this example:

   - **Blend Dimple Edges =** ✓
   - **Punch Radius = 0.5**
   - **Die Radius = 1**
   - **Blend Section Corners =** ✓
   - **Corner Radius = 0.5**

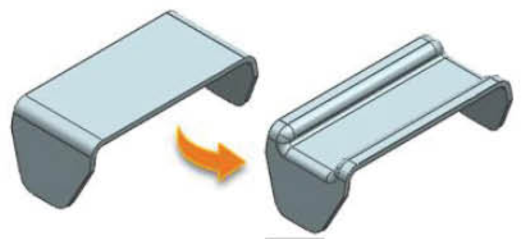


5. Click **OK** to create the dimple.

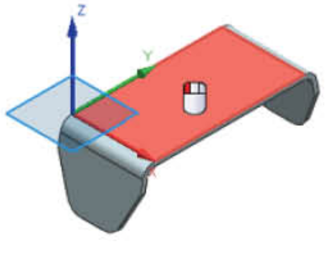


b. Create dimples across bends in the formed state

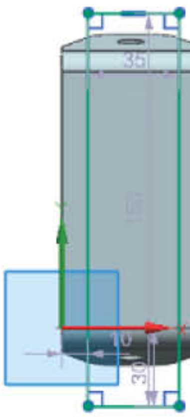
When you create dimples on bends without first flattening the bend, the dimple does not wrap around the bend.



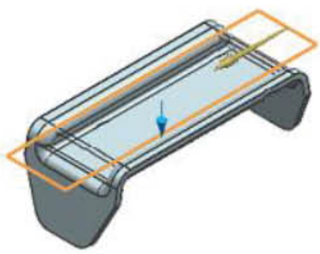
1. Choose **Home** tab→ **Punch** group→ **Dimple** .
2. In the **Section** group, click **Sketch Section** .
3. Select a face to specify the sketch plane and click **OK**.



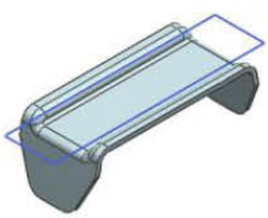
4. Sketch a rectangle.



5. Finish the sketch.

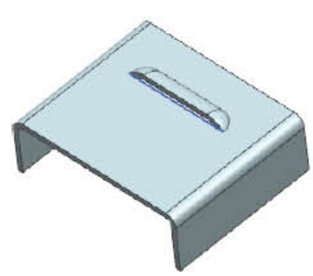


6. In the **Dimple Properties** group, set the following properties:
    - **Depth**
    - **Side Angle**
    - **Side Walls**
7. In the **Settings** group, ensure that the **Blend Dimple Edges** check box is selected.
8. Click **OK**.



2.3.2.7. 2 Louver

Use this command to create a customized vent by piercing the model along a single sketch line.

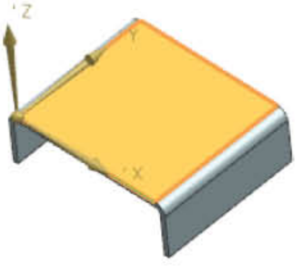


Louver features cannot be flattened.

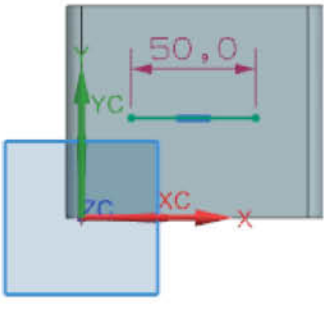
Create a louver

This procedure shows how to create a formed louver feature.

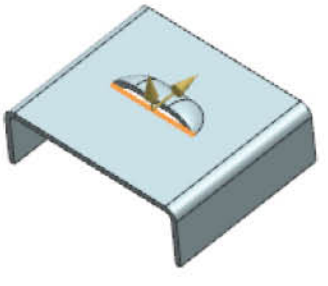
1. Choose **Home** tab→ **Punch** group→ **Louver** .
2. In the **Cut Line** group, click **Sketch Section** , and select the sketch placement face as shown in the following graphic.



3. Sketch a line as shown in the following graphic and click **Finish** to exit the Sketch.



4. In the **Louver Properties** group, set the following:
   - **Depth = 8**
   - **Width = 15**
   - **Louver Shape** = **Formed**

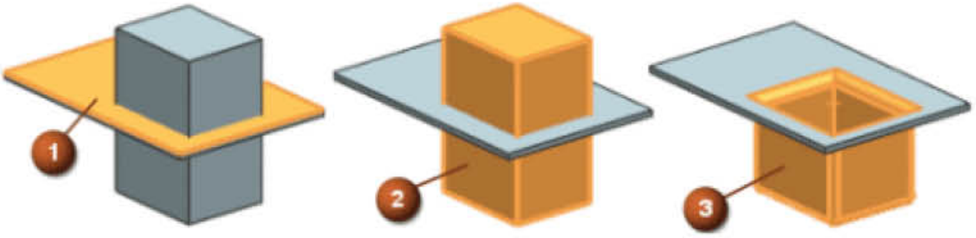


5. Click **OK** to create the louver.

2.3.2.7. 3 Solid Punch

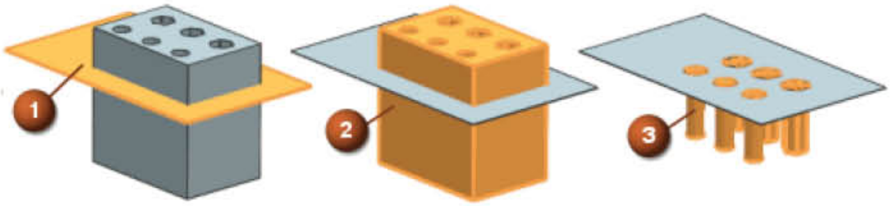
Use the **Solid Punch** command to add a sheet metal feature that inherits its shape from the tool body.

Example: Type = Punch



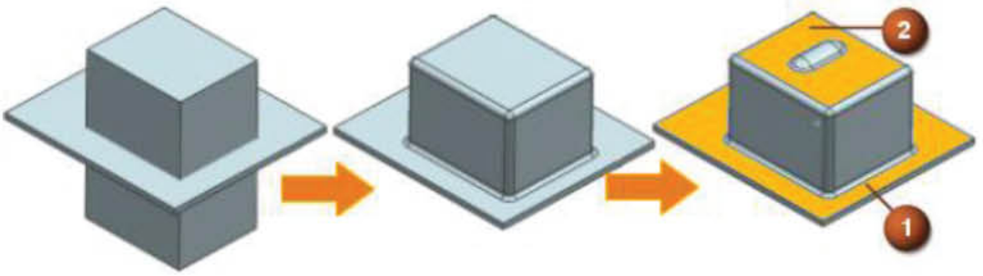
(1) Target face
(2) Tool body
(3) Resulting solid punch feature

Example: Type = Die



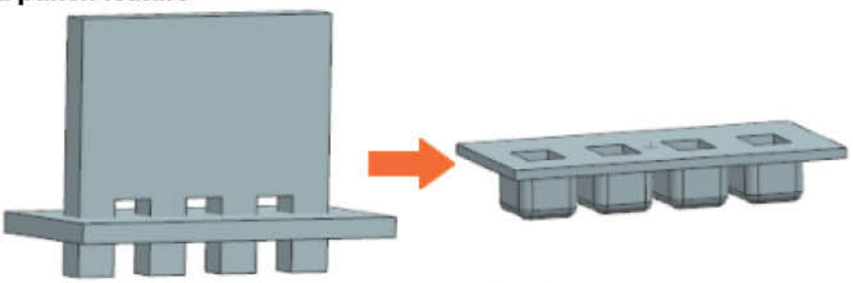
(1) Target face
(2) Tool body
(3) Resulting solid punch feature

You can create other sheet metal features such as a flange, beads, or dimples, on the face of a solid punch feature which is parallel to the target face.



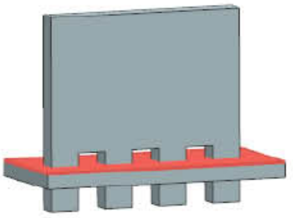
(1) Target face
(2) Tool body
(3) Resulting solid punch feature

Create a solid punch feature

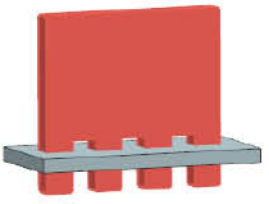


1. Choose **Home** tab→ **Punch** group→ **Solid Punch** .
2. In the **Type** group, from the list, select **Punch** .

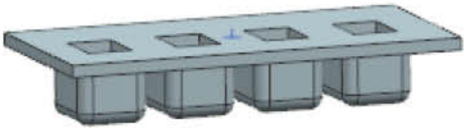
3. Select the target face that you want to punch.



4. Select the tool body.



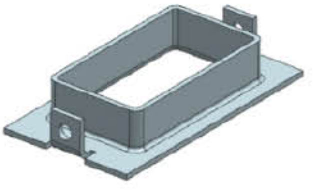
5. In the **Settings** group, set the following.
   Blend Edges = ☐
   Constant Thickness = ☑
   Centroid Point = ☑
   Hide Tool Body = ☑
6. Click **OK**.



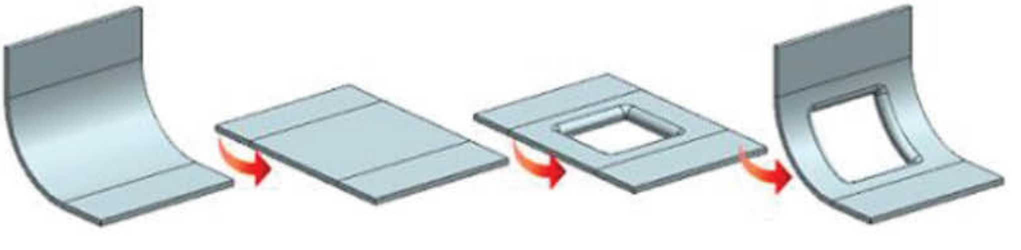
2.3.2.7. 4 Drawn Cutout

Use this command to cut an area of the model inside a sketch that simulates a stamping tool.

Similar to the **Dimple** command, the **Drawn Cutout** command makes an indentation in the sheet metal part, using a sketched section as its outline. The difference between the two is that a Dimple feature has a bottom face whereas the Drawn Cutout feature does not.



You can also create a Drawn Cutout feature across cylindrical bends in the unbent state. The Drawn Cutout feature can have an open or a closed section.



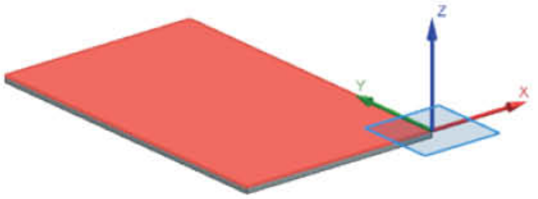
Create a drawn cutout

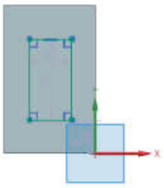
1. Choose **Home** tab→ **Punch** group→ **Drawn Cutout** .
2. In the **Section** group, click **Sketch Section**  to create a sketch that defines the section geometry.

   For this example, a sketch is created on the top face of the tab.

   Note:

   You can also create a sketch on the bottom face of the tab. In any case, NX creates the drawn cutout with consistent depth irrespective of the sketch position.

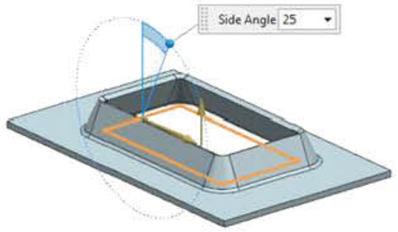
   

   A closed section must not touch any part edges.

   If you want to use an open section, the open ends of the section must intersect the part edges.

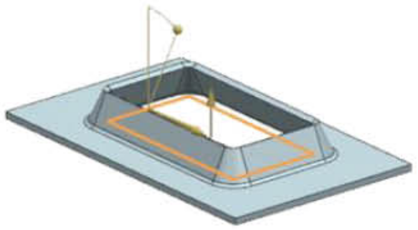
3. In the **Cutout Properties** group, specify the following parameters:
   - **Depth = 15**
   - **Side Angle = 25**

   - **Side Walls = Material Outside** 
4. In the **Settings** group, specify the parameters to blend the edges of the drawn cutout.

   For this example:

   - **Blend Drawn Cutout Edges** = ☑
   - **Die Radius = 2**
   - **Blend Section Corners** = ☑
   - **Corner Radius = 2**

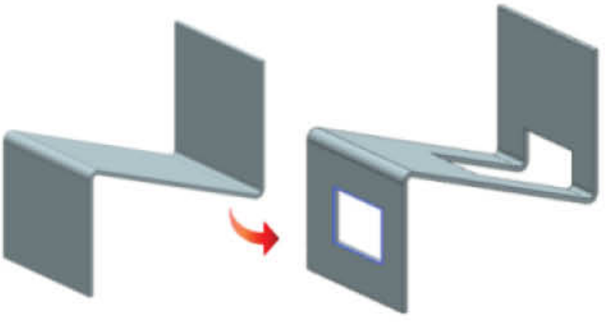
   

5. Click **OK** to finish the feature.

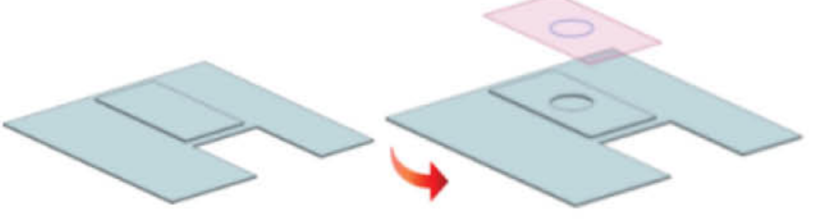
2.3.2.7. 5 Normal Cutout

Use the **Normal Cutout** command to cut material by projecting a sketch onto the model and then cutting perpendicular to the faces intersected by the projection.

You can use planar sketches or 3D curves to specify the section for the normal cutout.

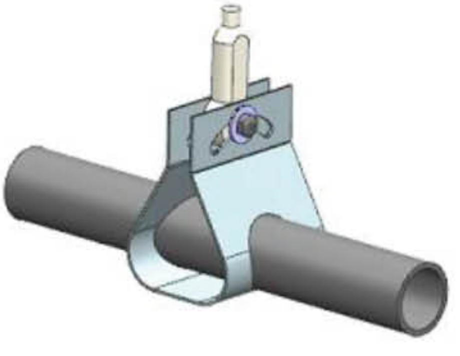


If the sheet metal part has multiple bodies, you can specify the target body on which you want to create the normal cutout.



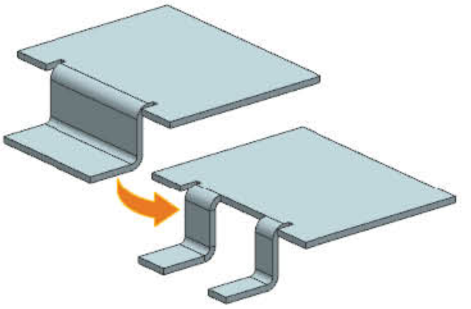
You can design the sheet metal part within the context of an assembly and create cutouts while the model is in the formed state.

In the example, the intersection curves of the two bodies are used as input to create a normal cutout. The cutout creates a void in the pipe hanger that allows the pipe to pass through it.

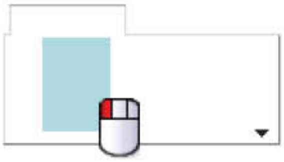


a. Create a normal cutout using a sketch

This example shows how to create a normal cutout using an open sketch section.

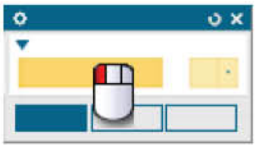


1.



Choose **Home** tab→ **Base** group→ **Normal Cutout** .

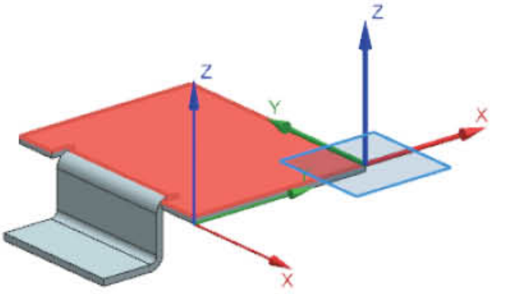
2.



In the **Normal Cutout** dialog box, set the following:

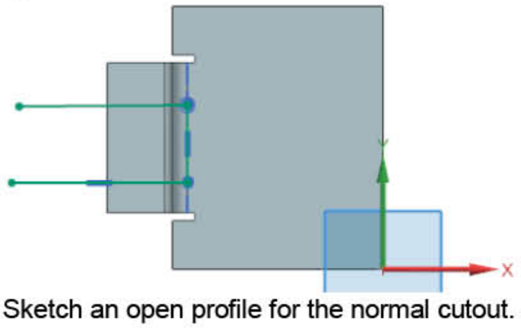
- **Type** = **Sketch**
- In the **Section** group, click **Sketch Section** .

3.



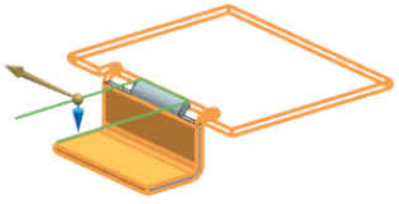
Select a plane for the sketch and then middle-click.

4.



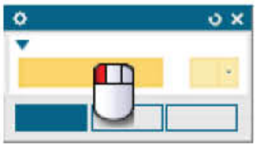
Sketch an open profile for the normal cutout.

5.



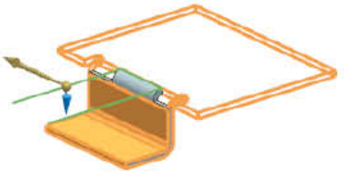
Finish the sketch.

6.

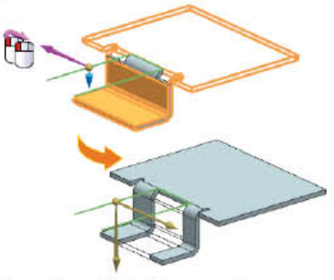


In the **Cutout Properties** group, set the following:

- **Cut Method = Thickness** 
- **Limits = Through All** 



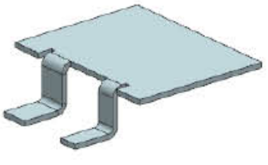
7.



Double-click the section arrow to reverse the direction of the normal cutout.

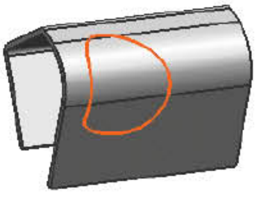
8.

Middle-click to create the normal cutout.

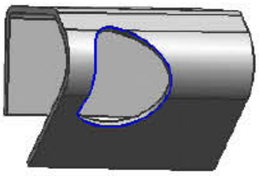


B. Create a Normal Cutout using a 3D curve

1. Choose **Home** tab→ **Base** group→ **Normal Cutout** .
2. From the **Type** list, select **3D Curves**.
3. Select the 3D curve that defines the cutout shape, as shown in the following figure.



4. Click **OK** to create the Normal Cutout feature.

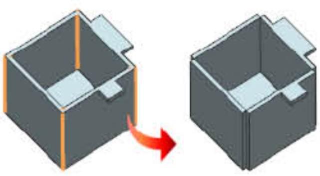


2.3.2.8. Convert to sheet metal
2.3.2.8.1 Convert to Sheet Metal Wizard

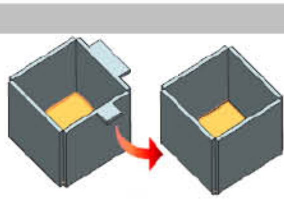
Use the **Convert to Sheet Metal Wizard** to convert a non-sheet metal part to a valid sheet metal part by ripping edges and cleaning up geometry.

The wizard streamlines the workflow by performing the cleanup and conversion tasks in the correct sequence and ensures a workable sheet metal model.

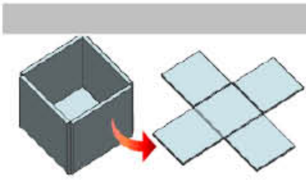
The example shows a typical workflow to convert a non-sheet metal part to a sheet metal part.



Rip along the corner edges. This step is optional, but if you do not rip edges that need to be ripped, they are defined as non-sheet metal areas that cannot be bent or unbent.



Clean up imported parts so that the cleaned up part meets the requirements of the **Convert to Sheet Metal** step. This is an optional step.



Convert the non-sheet metal part to a valid sheet metal part on which you can perform operations like bending, unbending, flattening, and so on.

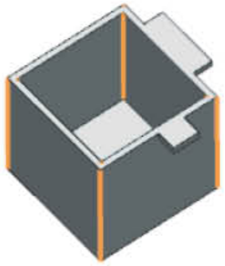
Convert a model to a Sheet Metal part

Note: You must be in the Sheet Metal application.

This example shows how to convert a non-sheet metal part to a valid sheet metal part with a thickness value of 3.

Choose **Home** tab→ **Convert** group→ **Convert to Sheet Metal Wizard** .

1. Select the corner edges to rip.



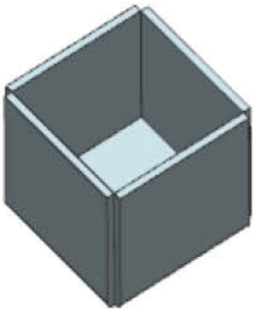
2. Click **Next** to advance to the **Cleanup Utility** step.
3. Select the base face to perform the cleanup operation.

   Because NX uses the thickness of the base face to infer the thickness of the resulting cleaned up part, select a base face that has uniform thickness.

4. In the **Thickness** group, clear the **Infer Thickness** ☐ check box as you want to enter a thickness value.
5. In the **Thickness** box, type **3**.
6. Click **Next** to advance to the **Convert to Sheet Metal** step.

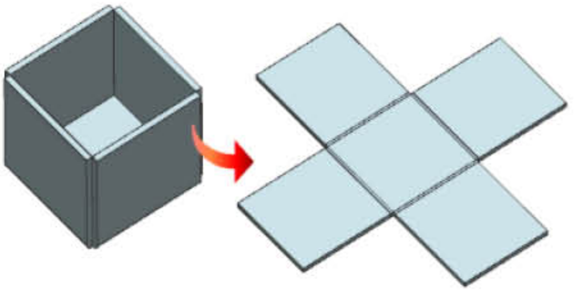
   You must perform this step to create a Convert to Sheet Metal feature. If you skip this step, the wizard will perform the cleanup operation, but it will not create a Convert to Sheet Metal feature to perform sheet metal operations like bending, unbending, flattening and so on.

7. Click **Finish** to create the Convert to Sheet Metal feature and exit the wizard.



You can later perform sheet metal operations like bending, unbending, flattening and so on.

The following graphic shows the unbent sheet metal part. The converted body is unbent using the **Unbend** command.

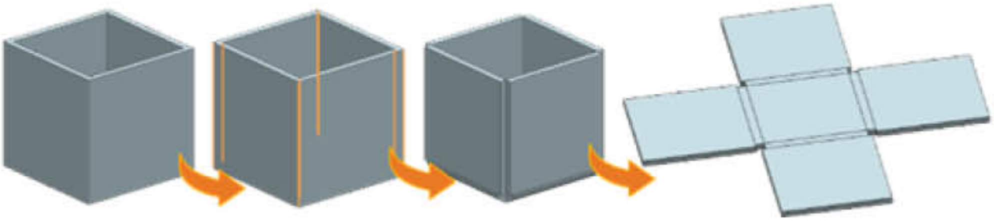


2.3.2.8.2 Rip

Use the **Rip** command to define cuts in sheet metal parts or rip corner edges when converting a solid model to a sheet metal part.

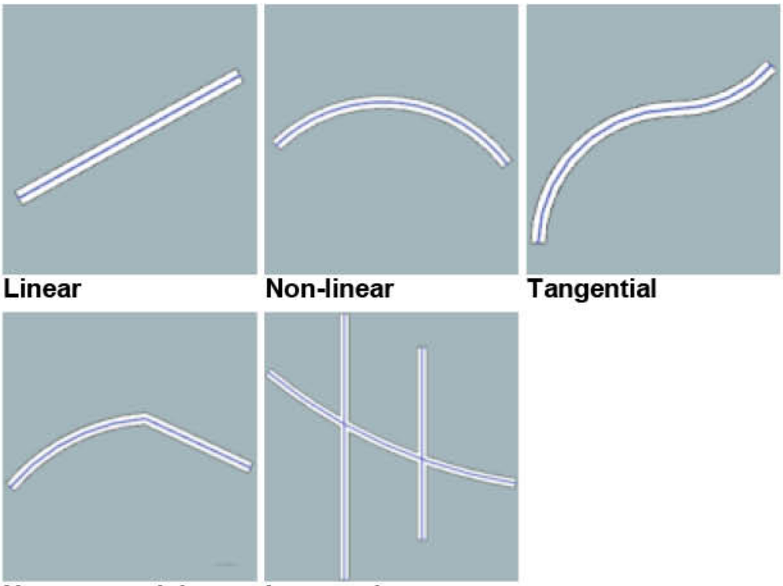
You can:

- Rip along corner edges.



- Rip along a curve.

  You can use the following types of curves to generate the rip:

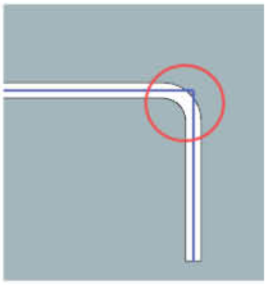


Linear **Non-linear** **Tangential**

Non-tangential **Intersecting**

When you use a curve to generate the rip, you can:

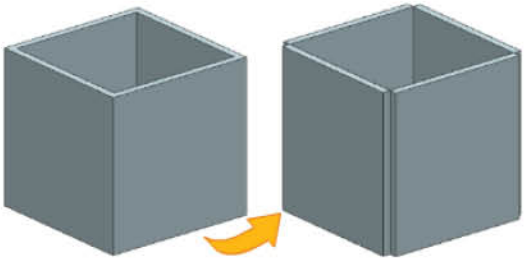
- Define the width and the shape of the end cap of the rip.
- Blend sharp corners by applying fillets.



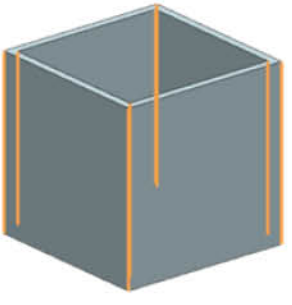
Note:

NX applies fillets only at the corners of a profile.

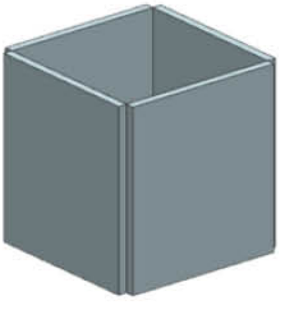
a. Rip along corner edges
This example shows how to rip the edges on a body.



1. Choose **Home** tab→ **Convert** group→ **Rip** .
2. Select the edges that you want to rip.



3. Click **OK** to rip the body along the selected edges.



B. Rip along a curve

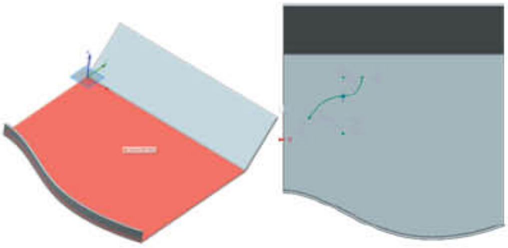
Use any of the following methods to rip the faces of a body along sketched curves.

Rip along non-linear curves

This example shows how to rip the face of the body along non-linear curves.

1. Choose **Home** tab→ **Convert** group→ **Rip** .
2. In the **Rip References** group, click **Sketch Section**  to create a non-linear curve along which you want to create a rip.

For this example, a sketch is created on the face of the body as shown.

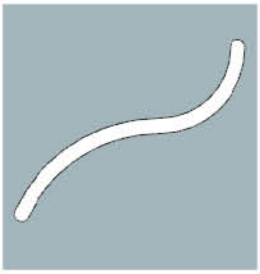


3. In the **Rip Parameters** group, set the parameters for the rip.

   For this example:

   - **Use Minimal Relief** = ☐
   - **Symmetric Offset** = ☑
   - **Rip Width = 5**
   - **End Cap Shape = Round**
   - **Blend Sharp Corner** = ☐

4. Click **OK** to rip along the sketch.

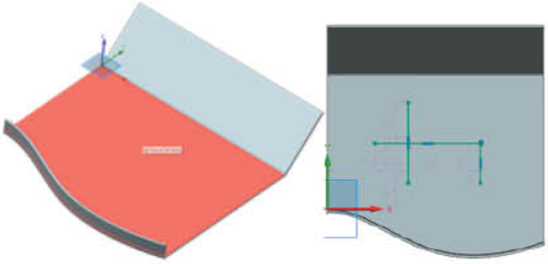


c. Rip along intersecting curves

This example shows how to rip the face of the body along the intersecting curves.

1. Choose **Home** tab→ **Convert** group→ **Rip** .
2. In the **Rip References** group, click **Sketch Section**  to create an intersecting curve sketch that defines the rip.

   For this example, the sketch is created on the face of the body as shown.



Note:

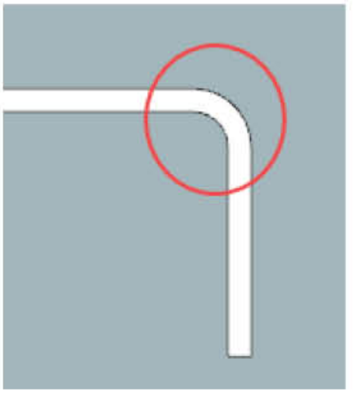
You cannot create bends at sharp corner points at which the curves intersect.

3. In the **Rip Parameters** group, set the parameters for the rip.

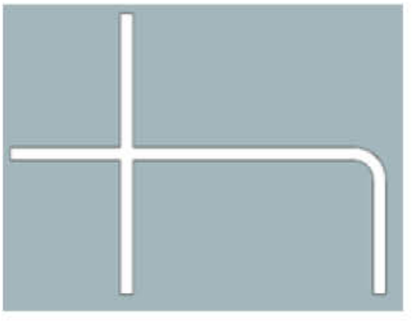
   For this example:

   - **Use Minimal Relief =** ☐
   - **Symmetric Offset =** ☑
   - **Rip Width = 5**
   - **End Cap Shape = Square**
   - **Blend Sharp Corner =** ☑
   - **Fillet Radius = 10**

NX applies fillets to blend the sharp corner only at the corners of the profile. In this example, NX blends the rightmost sharp corner by applying the specified curve radius.



4. Click **OK** to rip along the curve.



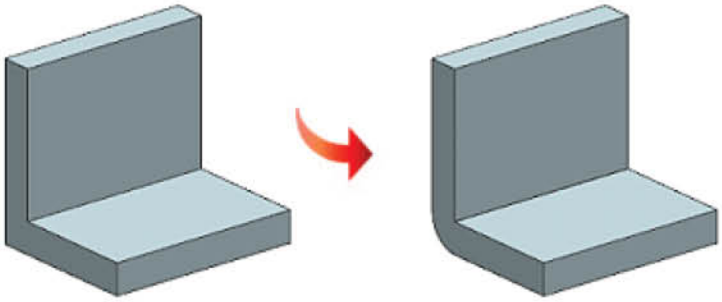
2.3.2.8.3 Cleanup Utility

Use the **Cleanup Utility** command to identify areas of a model that do not meet the requirements of the **Convert to sheet metal** command. If your part has sharp corners, you must rip the edges of the part to fix the sharp corners before you use the Cleanup Utility command.

You can clean up imported parts and make the conversion of non-sheet metal parts to sheet metal parts successful.

NX cleans up the sharp edges using a bend radius value of 0.02 mm. The model can have an inner sharp edge with an outer bend face, an outer sharp edge with an inner bend face, or both sharp edges.

For example, if a model has an inner sharp edge, the inner sharp edge is replaced with a bend face of radius 0.02 mm. The corresponding outer bend radius is equal to the sum of the inner bend radius and the input thickness.



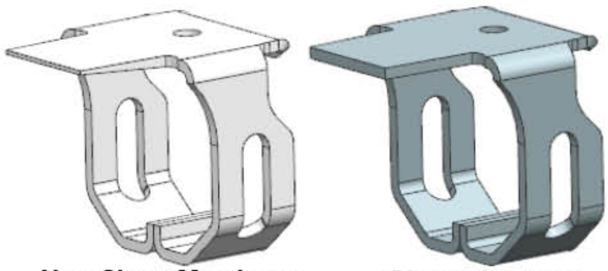
You can use customer defaults to change the value of tolerance.

Note:

- Use the **Cleanup Utility** command only if the model is not successfully converted using the **Convert to sheet metal** command.
- You can also use the **Optimize Face** command for additional cleanup.

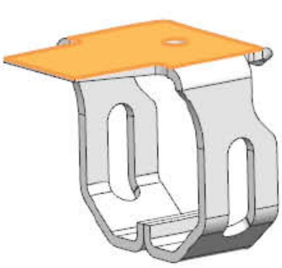
Clean up a part before converting it to a Sheet Metal part

This example shows how to clean up a non-Sheet Metal part before converting it to a Sheet Metal part.



Non-Sheet Metal part **Cleaned up part**

1. Choose **Home** tab→ **Convert** group→ **Cleanup Utility** .
2. Select the base face of the part.



3. In the **Thickness** group, specify the thickness of the sheet metal part.

   For this example, **Thickness = 1.2**.

4. In the **Sliver** group, specify the value of tolerance for identifying sliver faces.

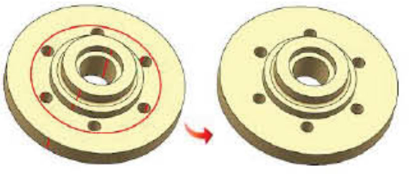
   For this example, **Sliver Tolerance = 0.01**.

5. In the **Settings** group, select the **Hide Original**  check box, to hide the original body after the cleanup.
6. Click **OK**.

   The part is now ready to be converted to a Sheet Metal part.

2.3.2.8.4 Optimize Face

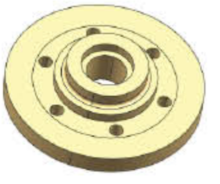
Use the **Optimize Face** command to simplify surface types, merge faces, improve edge accuracy, and recognize blends.



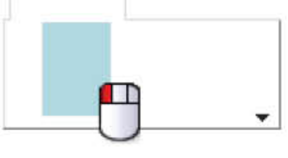
Use this command to convert B-surface faces to analytic faces on models that

Optimize faces

This example shows how to convert spline and tolerant edges to more accurate geometric edges and convert B-surfaces with analytic surfaces of an imported solid.



1. Choose **Home** tab→ **Convert** group→ **Optimize Face** .

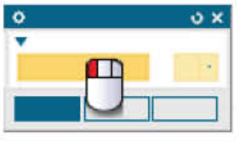


2. Select the faces that you want to optimize.

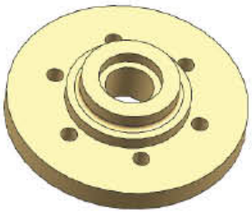
   For this example, all faces are selected.

3. In the **Settings** group, select the options that you want.

   In this example, **Clean Body before Optimize** and **Report** are selected.



4. Click **OK**.



The **Information** window displays the optimize face report.

2.3.2.8.5 Convert to Sheet Metal

You can use the **Convert to Sheet Metal** command to:

- Convert a non-sheet metal part to a valid sheet metal part so that you can perform sheet metal operations like bending, unbending, flattening, and so on. You can convert solid models and imported sheet metal-like parts to sheet metal parts.
- Remove corners of an imported or non-sheet metal geometry while converting the geometry to sheet metal.
- Convert a non-sheet metal part to a valid sheet metal part using global conversion, local conversion, or both. Using global conversion, you can convert an entire non-sheet metal part to sheet metal, whereas using local conversion, you can convert a local region such as a single face or a group of connected faces to sheet metal.

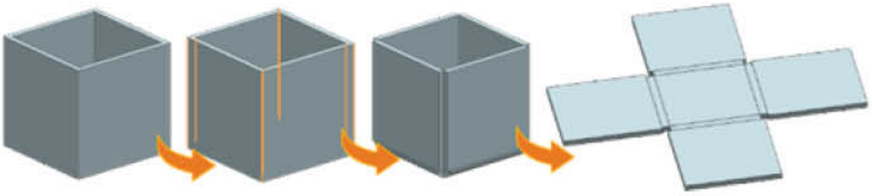
Thus, to fully convert a non-sheet metal part to sheet metal, first you use global conversion, and later use local conversion to convert small, local areas of the part that are not converted by the global conversion.

Note:

- There can only be one local conversion per feature.
- A convert feature can have a local conversion without a global conversion.

Converting solid models

The solid models must have a constant thickness. You cannot convert a sheet body to a sheet metal part. In the example, the solid model is created in the Modeling application, its edges are ripped using the **Rip** command, and then it is converted to a valid sheet metal part using the **Convert to Sheet Metal** command.



Converting non-sheet metal parts

When you convert a non-sheet metal part to a valid sheet metal part, you can also specify additional geometry for conversion.

In the example, an imported part containing non-cylindrical bends are selected as additional geometry for conversion. After conversion, you can use the **Unbend** command to flatten them.

Note:

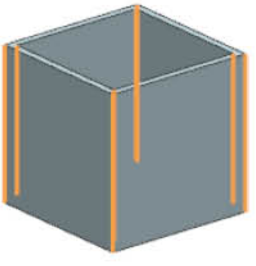
You cannot use the **Convert to Sheet Metal** command to rebend the areas that were previously unbent in the original non-sheet metal part.

a. Convert a solid model to a sheet metal part

This example shows how to convert a solid model that is created in the Modeling application, to a valid sheet metal part. The solid model is of uniform thickness.

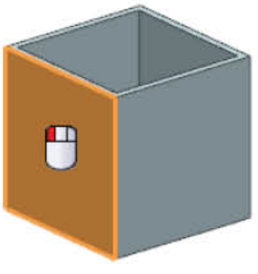
Note:

You must rip the edges of the solid model before converting it to a valid sheet metal part. Use the **Rip** command for this operation.



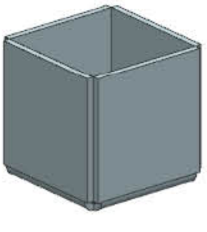
1. Choose **Home** tab→ **Convert** group→ **Convert to Sheet Metal** .
2. Select the base face.

   You must select a face that has tangent continuity with the other faces in the model.



3. Click **OK** to complete the conversion.

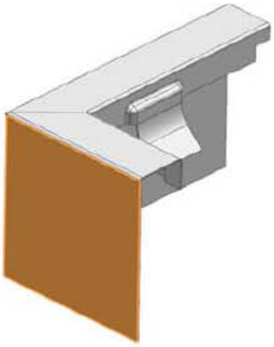
   NX creates corners wherever required.



You can now perform sheet metal operations such as bending and flattening on the part.

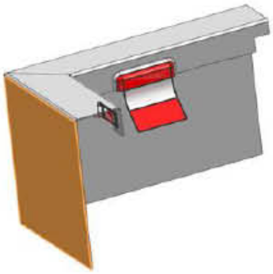
b. Convert a solid model to a sheet metal part using global and local conversions

1. Choose **Home** tab→ **Convert** group→ **Convert to Sheet Metal** .
2. In the **Global Conversion** group, select the base face.



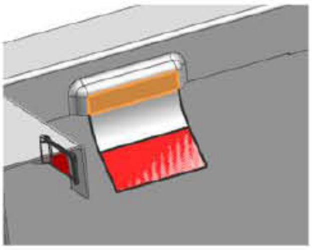
3. In the **Preview** group, select the **Preview** check box.

   In this example, the deform faces are identified and highlighted in a color specified for an alert, indicating that those faces cannot be bent or unbent.

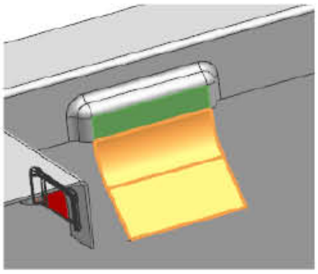


   In this case, if you try to perform flattening or unbending operations, the result may not be successful. Therefore, you need to convert the deform faces locally.

4. In the **Local Conversion** group, select the base face.



5. Select the faces to convert.



6. Click **OK** in the dialog box.

   A message appears, informing you that the body is converted successfully. However, faces that are identified and highlighted as deform faces cannot be bent or unbent.

7. Click **OK** in the message box.

   In this case, if you try to perform flattening or unbending operations, the result may not be successful, as the corner faces are intersecting. Therefore, you need to handle the corner faces.

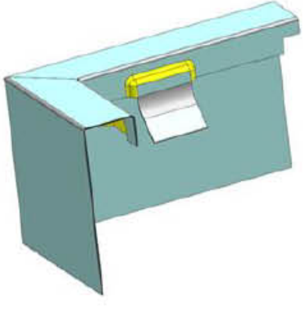
8. (Optional) On the Resource bar, select **HD3D Tools** tab→ **Visual Reporting** to view the conversion result.

   Using the **Visual Reporting**, you can check the conversion result. Also, you can see that the part under conversion is color coded with specific colors for deform faces, non-sheet metal features, or sheet metal features. This gives you clear visual feedback, highlighting the deform faces or faces that are not yet converted to sheet metal.

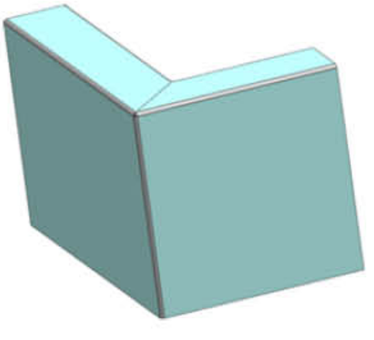
9. (Optional) In the **Report** group, from the list, select **Sheet Metal Face Type**.

10. (Optional) In the **Controls** group, click **Activate Report**.

    The part is color coded, highlighting specific colors for deform faces or the faces that are not yet converted to sheet metal. Also, the color legend is displayed in the **Results: Sheet Metal Face Type** group.



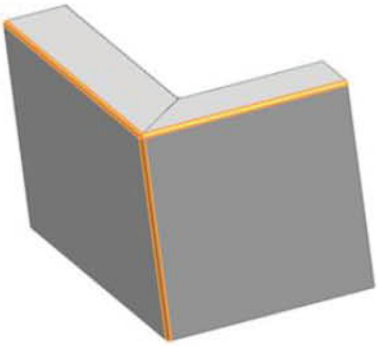
In this case, if you try to perform flattening or unbending operations, the result may not be successful, as the corner faces are intersecting.



Therefore, you need to handle the corner faces.

11. (Optional) In the **Controls** group, click **Deactivate Report**.

12. On the Resource bar, select the **Part Navigator** tab, and from the **Model History** node, double-click the **Convert to Sheet Metal** feature.
13. Choose **Corner Removal** group→ **Select Adjacent Bend Faces** .
14. Select the adjacent bend faces to remove the corner.

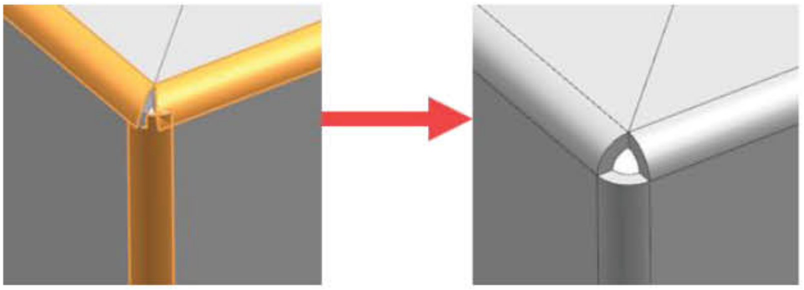


Note:

You can select the adjacent bend faces of multiple bend corners to open up the corners. To do this, you can click **Add New Set** to add a new set of faces in the **List** box. However, note that you can add only the adjacent bend faces in the same set.

15. Click **OK**.

The corners are removed and the part is fully converted to sheet metal.



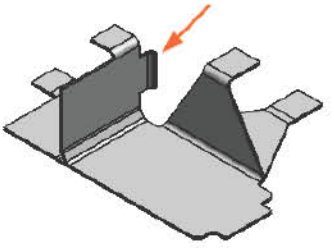
You can now add new corner treatments on the cleaned-up corners and perform sheet metal operations on the part, such as unbending or flattening.

2.3.2.9. Flat Solid And Flat Pattern
2.3.2.9.1 Flat Solid

You can use the **Flat Solid** command to:

- Create a flat representation of the formed sheet metal part. The flattened version of the part is associative to the formed version.
- Create an associative flat solid fixed at the current timestamp. This helps you to create a flat solid at an intermediate stage of the part design, rather than at the end.
- Control the orientation of the flat solid, using one of the following orientation methods:
  - **Default**: You can create a flat solid using the default orientation method, so the body will be flattened in place.
  - **Select Edge**: If you specify a linear edge from the selected stationary face, NX creates the flat solid by moving the selected edge to align with the X-axis of the absolute CSYS.
  - **Specify CSYS**: If you specify the CSYS, NX creates the flat solid by matching the specified CSYS with the absolute CSYS.

When you flatten a sheet metal part, the flat solid feature is added in the model history in the **Part Navigator**. If the part contains deformation features, they are retained in their formed condition. If you change the sheet metal model, the flat pattern automatically updates to include the new feature.



However, it is recommended not to build the bend creation (for example, **Flange**, **Bend**, **Jog**) or bend modifying (for example, resize operations) or **Convert to Sheet Metal** features on flat solid output.

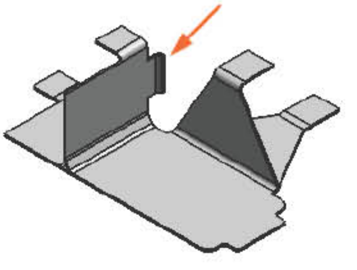
Note:

The mass properties of the flat solid are not exactly the same as the mass properties of the formed body.

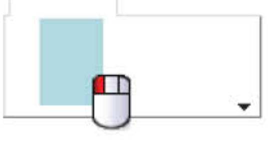
a. Create a flat solid for manufacturing

This example shows how to create a solid representation of the flat stock to which you can make dimensional changes without changing the definition of the original sheet metal part.

This example supposes that during prototyping, you found that the indicated tab is 1 mm too short after forming.

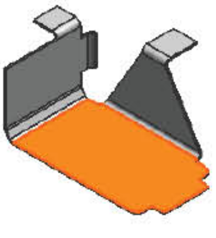


1.



Choose **Home** tab→ **Flat Pattern** group→ **Flat Solid** .

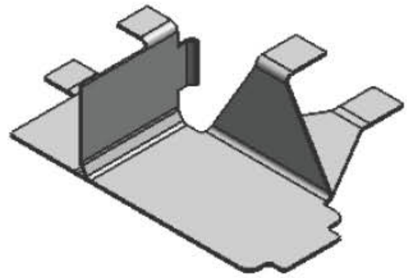
2.



Select the base tab as the stationary face.

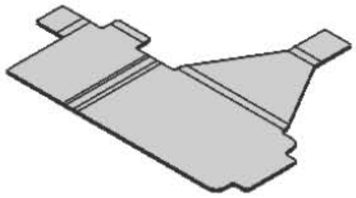
3.

Click **OK**.



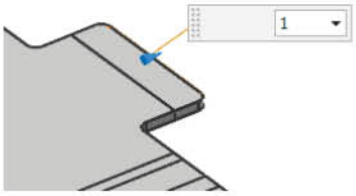
NX creates two solid bodies, one formed and one flattened.

4.



Right-click the formed solid body and choose **Hide**.

5.



Switch to the Modeling application and use the Synchronous Modeling command **Offset Region**  to offset the tab by 1 mm.

In the **Part Navigator**, notice that the flat solid feature follows all the sheet metal definition features, and is followed by the offset region feature. This means that if there is a change to the original sheet metal definition, NX will update the flat solid feature and then update the offset region feature.

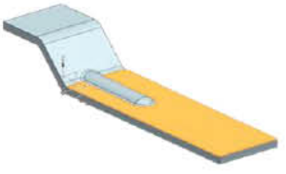
b. Create an oriented flat solid

You can create a flat solid by selecting a planar reference face and the **Orientation Method**. These options allow you to position the flat solid body for export to a machining tool.

Create a flat solid by specifying a linear edge

You can create a flat solid by specifying a linear edge from the selected stationary face.

1. Choose **Home** tab→ **Flat Pattern** group→ **Flat Solid** .
2. Select a base face to create the flat solid.



3. From the **Orientation Method** list, choose **Select Edge**.
   You can also select **Default** to create the flat solid using the default orientation method.
4. Select a linear edge from the selected base face for X-axis orientation.
   The curve of the selected edge is aligned with the X-axis of the absolute CSYS and the view is oriented accordingly.
5. In the **Preview** group, click **Show Result**.

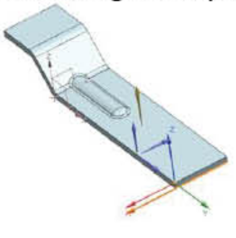
6. Click **OK** in the **Flat Solid** dialog box to create the feature.

Create a flat solid by specifying CSYS

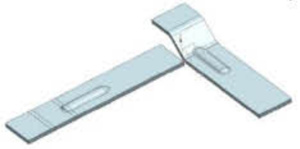
You can create and orient a flat solid by specifying a CSYS.

1. Choose **Home** tab→ **Flat Pattern** group→ **Flat Solid** .
2. Select a base face to create the flat solid.
3. From the **Orientation Method** list, choose **Specify CSYS**.
4. Specify the CSYS along which you want to align the flat solid.

   Note:
   For consistent results, select the Z-axis of the CSYS as the normal of the base face, and the CSYS origin as a point on the base face.



5. In the **Preview** group, click **Show Result**.
   The flat solid is oriented per the specified CSYS.

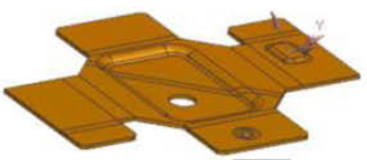


6. Click **OK** in the **Flat Solid** dialog box to create the feature.

Insert a bend table in a flat solid view

This example shows how to create a bend table using the bend information that NX displays in the flat solid view.

Before you begin, you must create a flat solid.



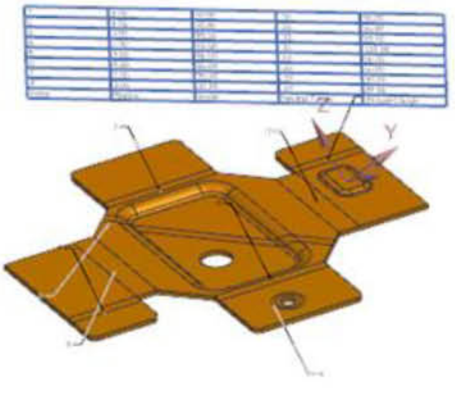
1. Choose **PMI** tab→ **Rule** group→ **Insert Instance** .
2. In the **Insert Instance** dialog box, in the **Folder View** group, expand the **MBD Logical Rules** node and select **Table**.
3. In the **Member Select** panel, select the **Create Bend Table** rule and click **OK**.

   Tip:

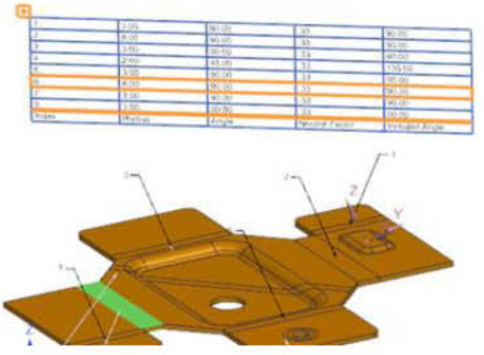
   Select the **Create Bend Table** command that creates a table from the sheet metal bends in the part.
4. In the graphics window, select the flat solid.

5. Click **OK**.

   NX inserts the bend table in the flat solid view.



When you select a row in the bend table that represents a bend, NX highlights the corresponding bend in the flat solid.



Insert a bend table in a body

This example shows how to create a bend table using the bend information that NX displays in a body.

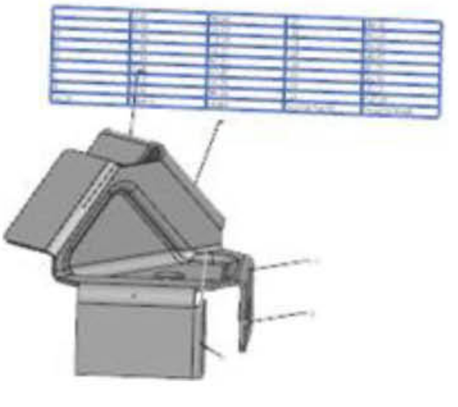
1. Choose **PMI** tab→ **Rule** group→ **Insert Instance** .
2. In the **Insert Instance** dialog box, in the **Folder View** group, expand the **MBD Logical Rules** node and select **Table**.
3. In the **Member Select** panel, select the **Create Bend Table** rule and click **OK**.

   Tip:

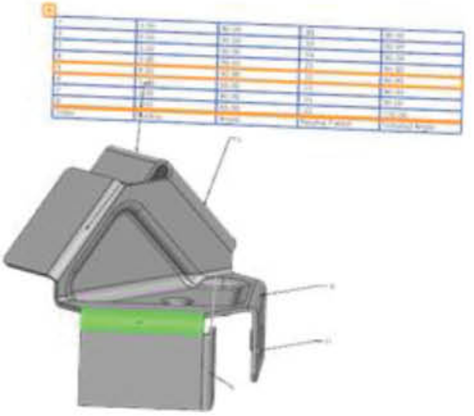
   Select the **Create Bend Table** command that creates a table from the sheet metal bends in the part.
4. In the graphics window, select the body.

5. Click **OK**.

   NX inserts the bend table and displays in the graphics window.



When you select a row in the bend table that represents a bend, NX highlights the corresponding bend in the body.



Generate the blank shape

Use **Flat Pattern** when you are making drawings of the flattened blank shape created by your sheet metal part.

Any changes made to the 3D formed model are automatically reflected in the **Flat Solid**, and also in the **Flat Pattern**.

However, it is recommended not to build the bend creation (for example, **Flange**, **Bend**, **Jog**) or bend modifying (for example, resize operations) or **Convert to Sheet Metal** features on Flat Solid output.

To create a **Flat Pattern** feature:

1. Choose **Home** tab→ **Flat Pattern** group→ **Flat Pattern** .
2. Select a planar face from the formed model to use as a basis for the flat pattern.
3. Click **OK** to create the **Flat Pattern**.

The **Flat Pattern** feature is created in a special FLAT-PATTERN view that you can add to a drawing. To see the FLAT-PATTERN, in the **Part Navigator**, click to expand **Model Views**, then double-click the FLAT-PATTERN view.

2.3.2.9.2 Flat Pattern

Use the **Flat Pattern** command to create a 2D representation of the flattened 3D sheet metal part.

Creating a flat pattern

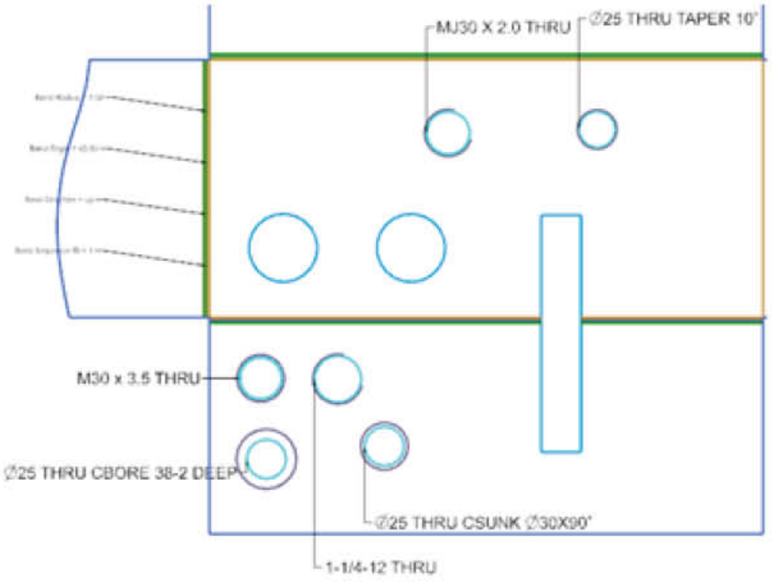
You can:

- Create a flat pattern from a formed sheet metal body or from flat solids.
- Create an associative flat pattern fixed at the current timestamp. This helps you create a flat pattern view at the intermediate stage of the part design.
- Control orientation of the Flat Pattern view for the upward face having non-linear edges. To do this, you need to specify a CSYS while defining the orientation.

Viewing hole details in a flat pattern

You can:

- Visually identify the types of holes and their parameters, such as the drill size, the screw clearance, and if it is threaded.

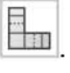


Setting Sheet Metal preferences for a flat pattern

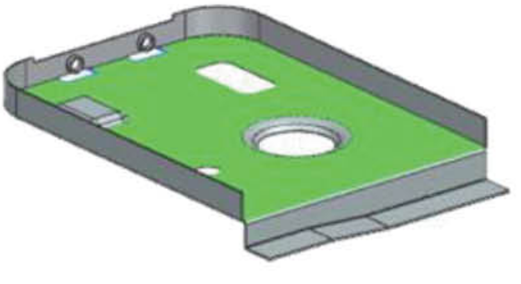
You can specify the flat pattern treatments and the default display of curves and callouts in the flat pattern view for each part individually in the **Sheet Metal Preferences** dialog box.

Create a flat pattern view by specifying a linear edge

This example shows how to create a flat pattern by specifying a linear edge from the selected upward face.

1. Choose **Home** tab→ **Flat Pattern** group→ **Flat Pattern** .
2. Select an upward face to create the flat pattern view.

   This face can either be on a formed sheet metal body, or on a flat solid body. The normal direction associated with this face is used to specify the upward direction for bend regions. For this example, select the upper tab face.



3. From the **Orientation Method** list, choose **Select Edge**.
   You can also select **Default** to create the flat pattern view using the default orientation method.
4. Select a linear edge from the selected upward face for X-axis orientation.
   The curve of the selected edge is aligned with the datum CSYS and the view is oriented accordingly.

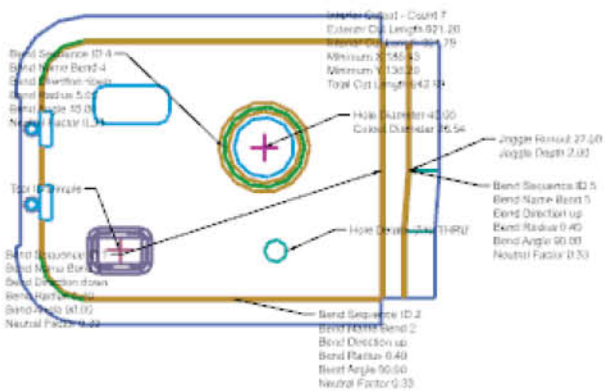
5. In the **Preview** group, click **Show Result**.
   A message appears, informing you that the flat pattern feature is created in the separate Flat Pattern view.
6. Click **OK** in the message box and in the **Flat Pattern** dialog box to create the feature.
   NX creates the following in the **Part Navigator**.

   - A flat pattern feature in the **Model History** node.
   - A flat pattern view **FLAT-PATTERN#nnn** in the **Model Views** node.

     Where nnn corresponds to the number that is incremented each time when you create a flat pattern view.

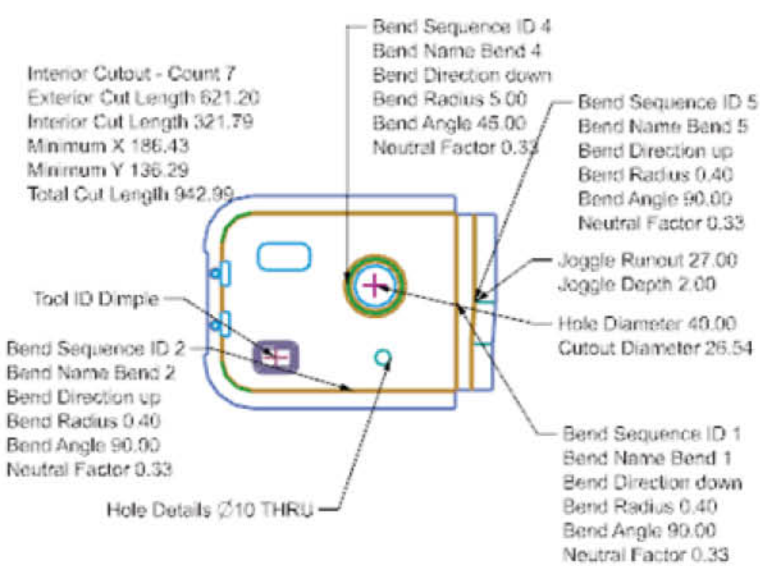
7. In the **Part Navigator**, expand the **Model Views** node, and double-click **FLAT-PATTERN#nnn**.

   NX displays flat pattern information as PMI objects in the flat pattern view.



   You can resize and reposition the generated PMI labels.

8. Choose **PMI** tab→ **Display** group→ **Resize** .
9. Click **Position** .

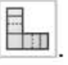


10. View the flat pattern labels that NX creates.

    a. On the Resource bar, click the **MBD Navigator** tab.
    b. In the **MBD Navigator**, expand the **PMI→ Annotation** node to view the flat pattern labels.

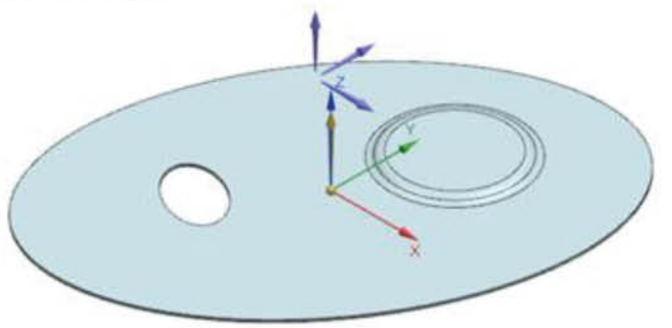
Create a flat pattern view by specifying a CSYS

You can create a flat pattern view by specifying a linear edge from the selected upward face. However, this method of orientation is not useful for upward faces that have non-linear edges. In such cases, you can specify a CSYS and create a flat pattern view that is oriented per the specified CSYS.

1. Choose **Home** tab→ **Flat Pattern** gallery→ **Flat Pattern** .
2. Select an upward face to create the flat pattern view.
3. From the **Orientation Method** list, choose **Specify CSYS**.
4. Specify the CSYS along which you want to align the flat pattern view.

   Note:

   For consistent results, select the Z-axis of the CSYS as the normal of the base face, and CSYS origin as a point on the base face.

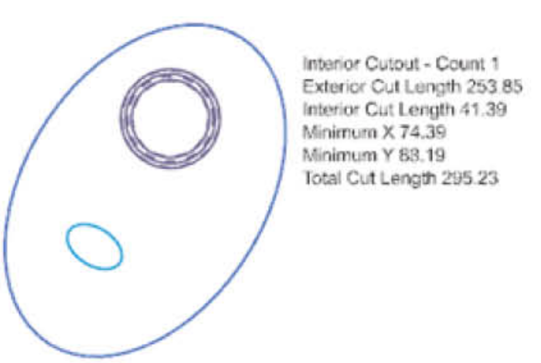


   The flat pattern view is oriented per the specified CSYS.

5. In the **Preview** group, click **Show Result**.

   A message appears, informing you that the flat pattern feature is created in the separate flat pattern view.

6. Click **OK** in the message box and in the **Flat Pattern** dialog box to create the feature.
7. To see the flat pattern view, in the **Part Navigator**, expand the **Model Views** node, and double-click **FLAT-PATTERN**.

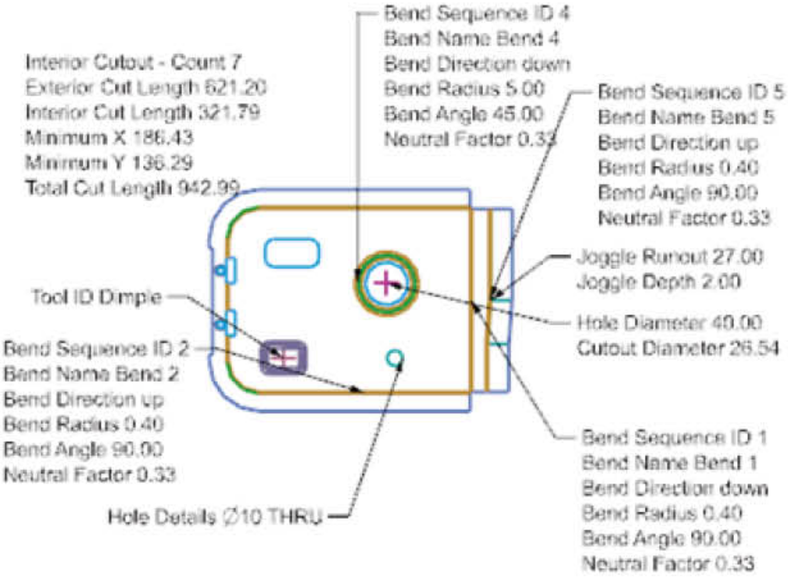


Editing a flat pattern feature

The flat pattern view consists of a set of curves and associated annotations. You can edit curves in the flat pattern. However, when you update the model, the curves associated with the flat pattern are regenerated and the changes are overwritten. To avoid this, you can create a non-associative flat pattern view.

Displaying PMI in a flat pattern

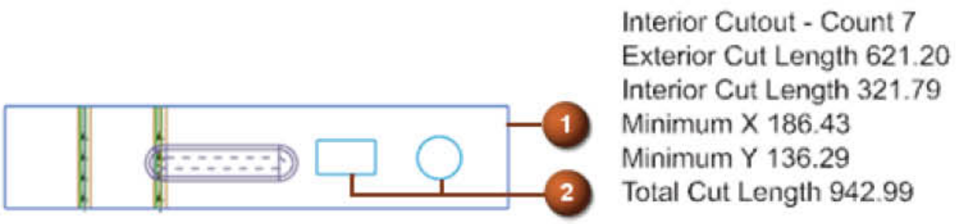
When you create a flat pattern, NX displays flat pattern information as PMI objects in the flat pattern view.



You can view the following PMI objects in the flat pattern view.

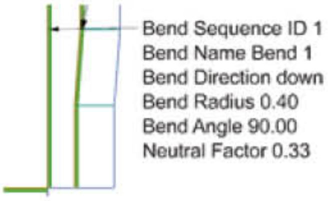
- Flat pattern attributes such as interior cutout count, minimum X, and minimum Y.

  NX also displays the total cut length including the exterior boundary (1) and interior cutouts (2) of a flat pattern.

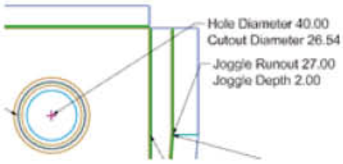


  You can view this information in the **Flat Pattern Properties** dialog box.

- Information about a bend such as bend sequence ID, bend direction, and bend angle.



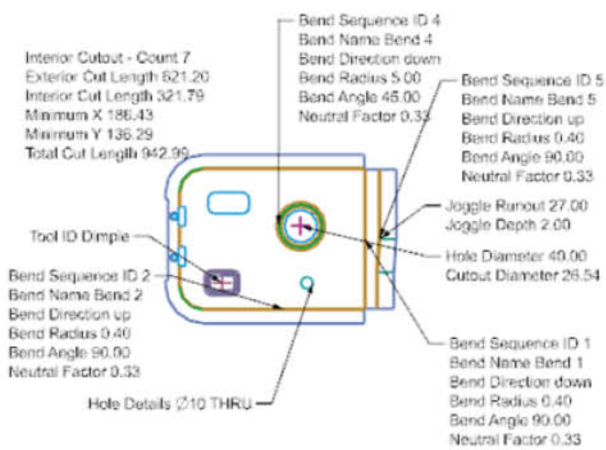
- Other information such as hole diameter, tool ID, and cutout diameter.



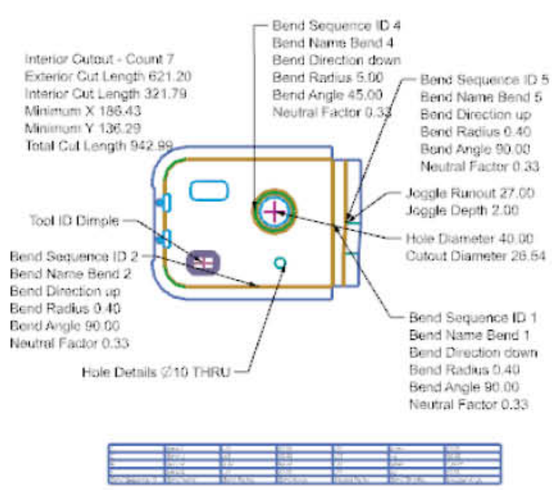
Insert a bend table in a flat pattern view

This example shows how to create a bend table using the bend information that NX displays in the flat pattern view.

1. In the **Part Navigator**, expand the **Model Views** node, and double-click the flat pattern view in which you want to create a bend table.



2. Choose **PMI** tab→ **Rule** group→ **Insert Instance** .
3. In the **Insert Instance** dialog box, in the **Folder View** group, expand the **MBD Logical Rules** node and select **Table**.
4. In the **Member Select** panel, select the **Create Bend Table** rule and click **OK**.
5. In the **Part Navigator**, expand the **Model Views** node and select the flat pattern view.
6. Click **OK**.



Now, when you create a drawing of the flat pattern, you can include this bend table in the drawing.

2.3.2.9.3 Bend List

The **Bend List** command is available only after you create a flat pattern and replace the modeling view with the flat pattern view.

After creating a flat pattern, each bend in the part is assigned a unique **Bend Sequence ID** and **Bend Name**. Using the **Bend List** command, you can edit these default bend sequence IDs and bend names per your requirement.

You can select a bend and move it up or down and change its sequence. After you move the bend, it is assigned with the new **Bend Sequence ID**. After the bend is moved to the required position, you can change its name if required.

The **Bend List** command is available for both associative and non-associative flat pattern views.

Edit bend properties

You can edit bend properties such as bend sequence ID and bend name using the **Bend List** command.

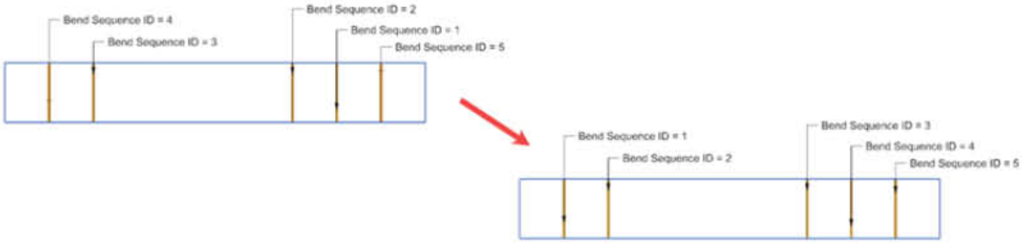
The **Bend List** command is available only after you create a flat pattern and replace the modeling view with the flat pattern view.

1. Use the **Flat Pattern** command to create a flat representation of the part.
2. To see the flat pattern view, in the **Part Navigator**, expand the **Model Views** node, and double-click **FLAT-PATTERN**.
3. Choose **Home** tab→ **Flat Pattern** group→ **Bend List** .
4. Under the **Existing Bend ID** column, select the bend and click **Move Up**  or **Move Down**  .

The existing bend ID is changed.

5. Click **Apply**.

The bend sequence ID is changed and updated in the **Flat Pattern** view.



6. (Optional) In the **Existing Bend ID** column, select a bend ID, and in the **Bend Name** box, type a different name.
7. Click **OK**.

The bend properties are edited.

Export a Flat Pattern feature
This example shows how to export Flat Pattern feature in the DXF format using the **2D Exchange** translator.

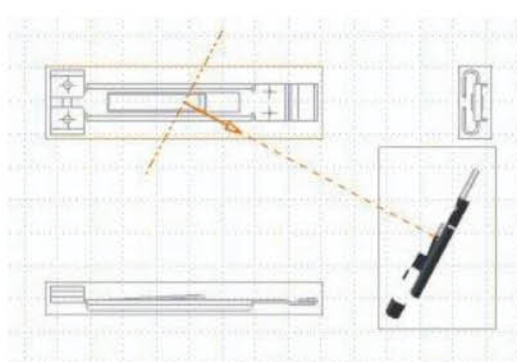
Note:
Exporting a Flat Pattern feature through a translator requires a proper translator license. This license is not included as part of Sheet Metal license.

1. Choose **Menu→File→Export→2D Exchange**.
2. In the **2D Exchange** dialog box, select the **Files** tab.
3. On the **Files** page, in the **Export to** group, select **DXF** from the **Output As** list.
4. Specify where you want to save the new file.
5. Click the **Data to Export** tab.
6. In the first **Export** list, choose **Selected Objects**.
7. In the **Model Data** group, click **Select Object** , then in the graphics window select the part.
8. In the second **Export** list, choose **Selected View**, select FLAT-PATTERN from the view list.
9. Click **OK** to export.

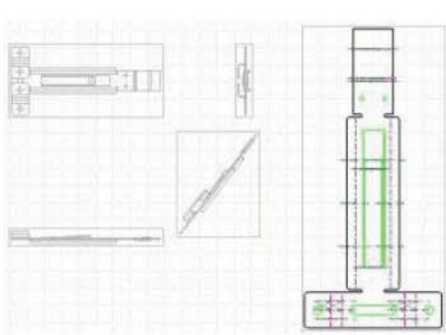
Showing both formed and flattened views on a drawing

With Sheet Metal, you can show views of your solid part in both the formed and unformed states:

1. Create a **Flat Pattern** feature.
2. Enter the Drafting application and create views of the formed solid part.



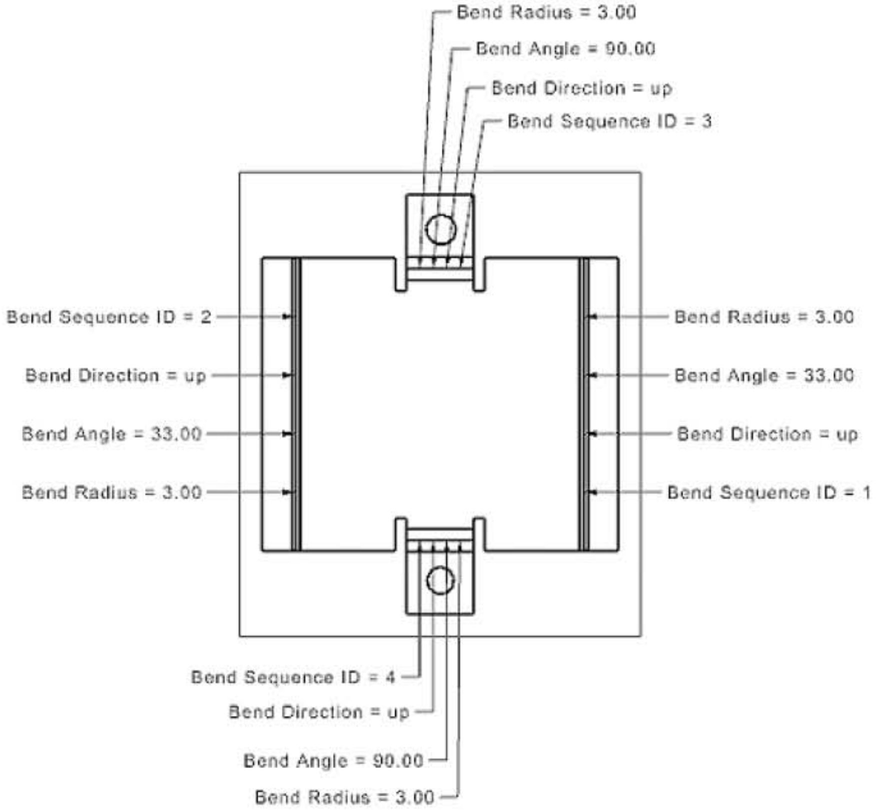
3. Click **Base View** and choose FLAT-PATTERN from the list of available views.
4. Add the flat pattern view just as you would any other part view.



Note that for the flat pattern view to be visible, the layer that the flat pattern curves were placed on (according to your **Customer Defaults**) must be a visible layer. If you are placing the flat pattern view and do not see any resulting geometry, check your layer settings.

2.3.2.9.4 Bend Table

Use the **Bend Table** command to insert a bend table in a drawing that you create using the Flat Pattern view of a Sheet Metal or FPCD part. The tabular format makes the bend information easy to read in downstream applications.



4	Bend 4	3,00	90,00	up	90,00
3	Bend 3	3,00	90,00	up	90,00
2	Bend 2	3,00	33,00	up	147,00
1	Bend 1	3,00	33,00	up	147,00
ID	Name	Radius	Angle	Direction	Included Angle

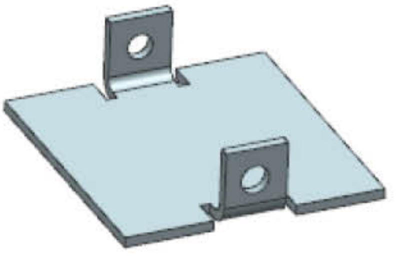
To display the bend parameters that you want in the bend table, you must set preferences in the **Drafting Preferences** dialog box. You can display any or all of the following bend parameters:

- **Bend ID**
- **Bend Name**
- **Bend Radius**
- **Bend Angle**
- **Bend Direction**
- **Included Angle**

NX derives the values of the bend parameters from the Flat Pattern view. If you delete the reference Flat Pattern view, NX deletes the bend table.

Insert a bend table in a drawing of a flat pattern view

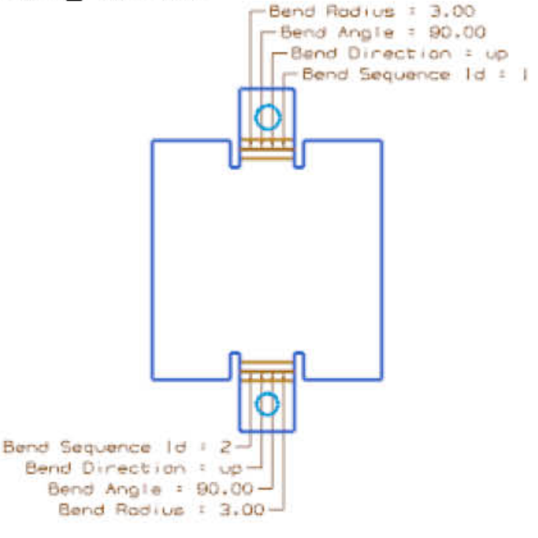
This example shows how to create a flat pattern view of the part shown, how to place it in a drawing, and how to insert the bend data in a table in the drafting view. For this example you will display only the **ID**, **Name**, **Radius**, and **Angle** columns in the bend table.



1. In the Sheet Metal or FPCD application, use the **Flat Pattern** command to create a flat representation of the part.

   Tip:

   To display the flat pattern view, in the **Part Navigator**, under the **Model Views** node, double-click FLAT_PATTERN.



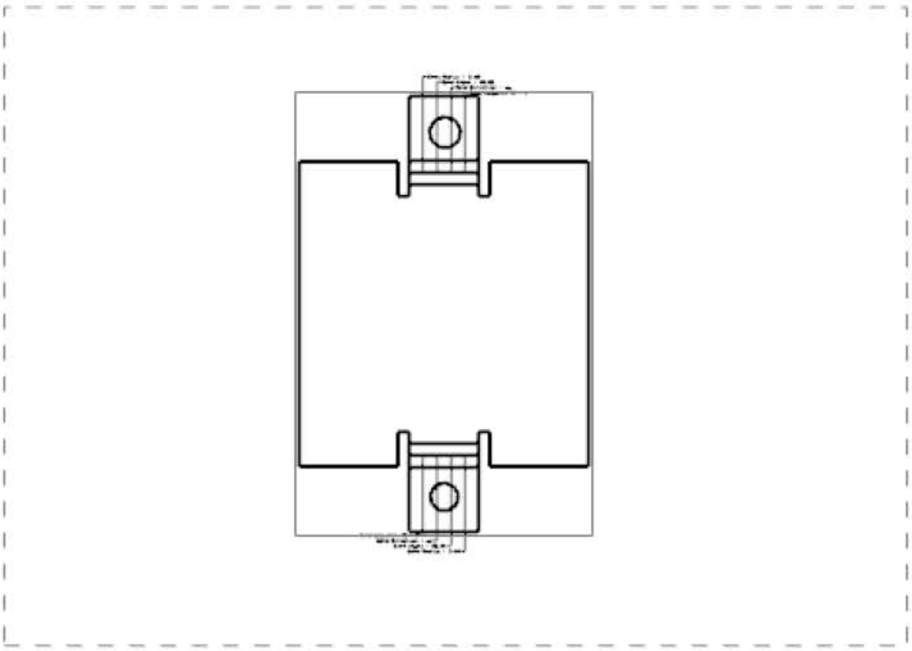
2. Switch to the Drafting application.

   The **Sheet** dialog box is displayed.

3. In the **Settings** group, make sure **View Creation Wizard** is selected.
4. Select your sheet size and click **OK**.

   The **View Creation Wizard** dialog box is displayed.

5. In the left pane, select the **Orientation** step.
6. In the **Orientation** group, in the **Model Views** list box, select the required flat pattern view.
7. Click **Finish** to add the flat pattern view to the sheet.



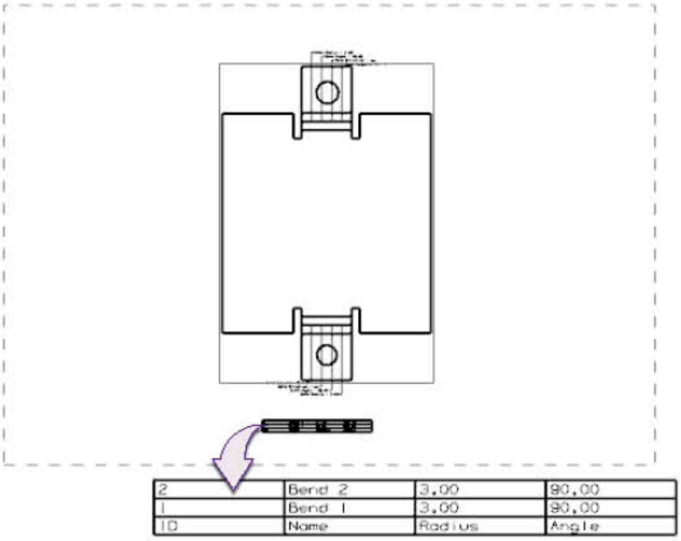
8. (Optional) To make your view smaller, do the following:
   a. Double-click your view border.
   b. In the **Settings** dialog box, in the left pane, choose **Common**→ **General**.
   c. In the **Settings** group, set your scale and click **OK**.

9. Choose **Home** tab→ **Table** group→ **Bend Table** .

10. In the **Settings** group, click **Settings** .

    The **Settings** dialog box is displayed.

11. In the **Columns** group, clear the check boxes next to the **Bend Direction** and **Included Angle** options so that these columns are not displayed in the bend table.
12. Click **Close** to return to the **Bend Table** dialog box.
13. Select the flat pattern view and then click in the sheet to place the bend table at the required location.
14. Click **Close**.



2.3.2.9.5 Sheet Metal Label

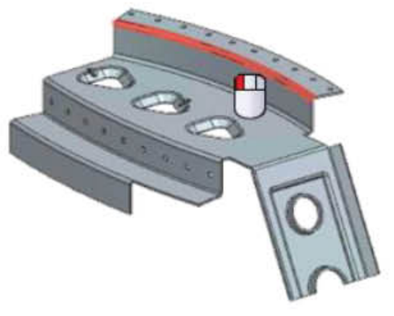
Use this command to display Product and Manufacturing Information (PMI) for a sheet metal body and a bend.

Add sheet metal labels and view them

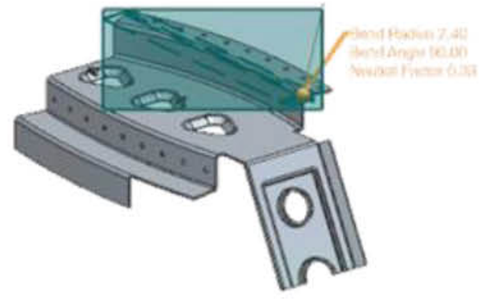
This example shows how to add PMI labels to a sheet metal body and a bend face on it.

Add a sheet metal label to a bend face

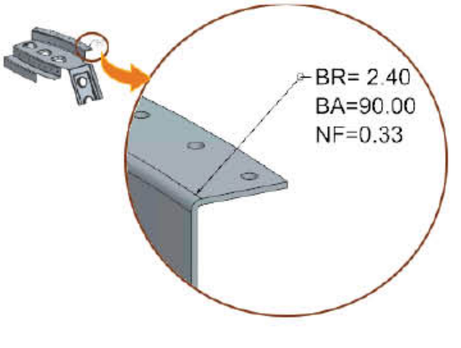
1. Choose **Home** tab→ **Flat Pattern** group→ **Sheet Metal Label** .
2. From the type list, select **Bend**.
3. Select the bend face that you want to label.



NX displays a preview of the label with the plane on which the label will appear.

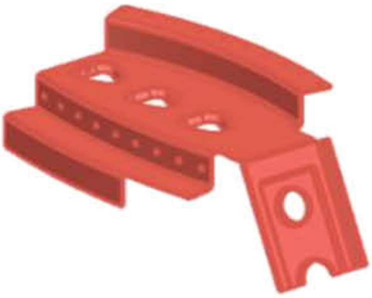


4. In the **Leader** group, from the **Type** list, select **All Around**.
5. In the **Settings** group, click **Settings** .
6. In the **Sheet Metal Label Settings** dialog box, set the following parameters.
7. Click **Close**.
8. In the graphics window, click the location at which you want to place the sheet metal label.

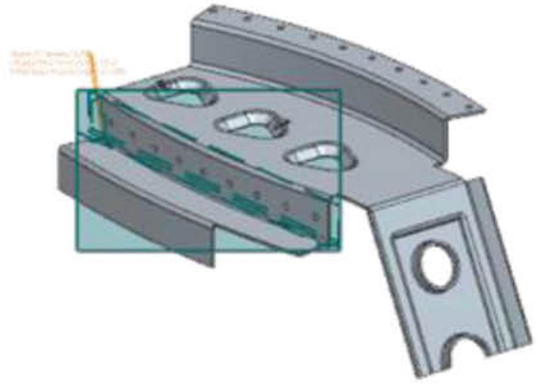


Add a sheet metal label to a body

1. Choose **Home** tab→ **Flat Pattern** group→ **Sheet Metal Label** .
2. From the type list, select **Body**.
3. Select the body that you want to label.

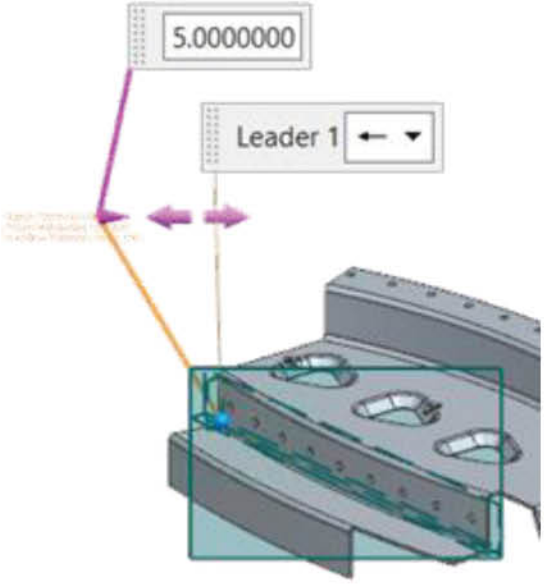


NX displays a preview of the label with the plane on which the label will appear.

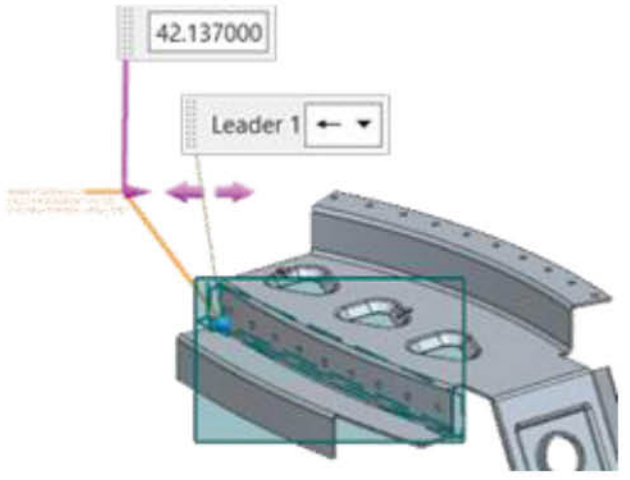


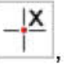
4. In the **Leader** group, click **Select Terminating Object**.
5. Point to the label.

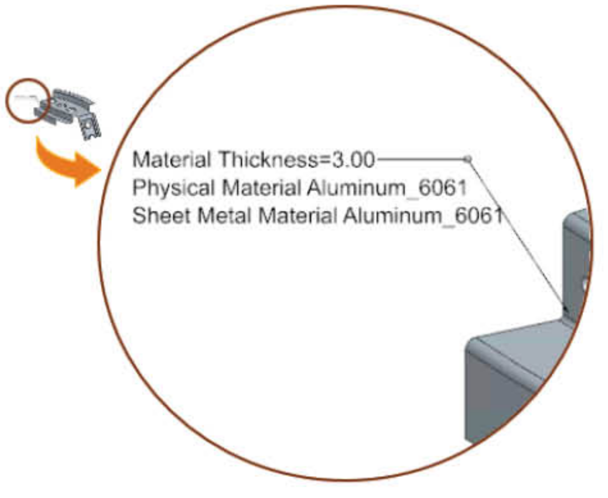
NX displays handles to adjust the stub and the **Leader** scene list.



6. Drag the handle to adjust the stub length.



7. (Optional) From the **Leader** scene list, choose a style for the display of the arrowhead.
8. Click **Close**.
9. In the **Origin** group, click **Specify Location** , and then click in the graphics window to place the sheet metal label.



2.3.2.10. Resize

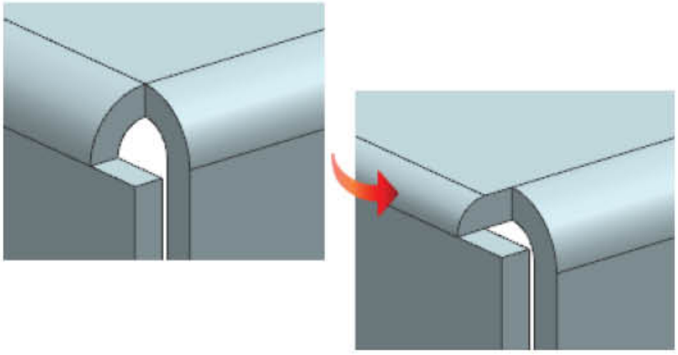
2.3.2.10.1 Resize Bend Radius

Use the **Resize Bend Radius** command to change the radius of a cylindrical bend connected to planar faces, by overriding the feature that created the bend.

You can modify the radius of individual bends to illustrate over-bending and spring back effects.

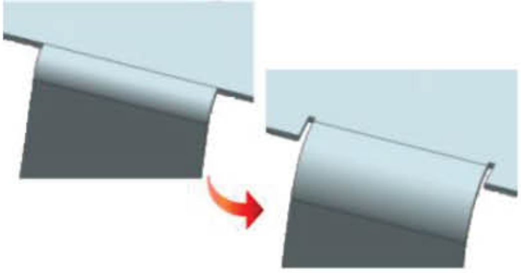
You can change the radius of the following:

- Existing bends to create bends that have zero bend radii.

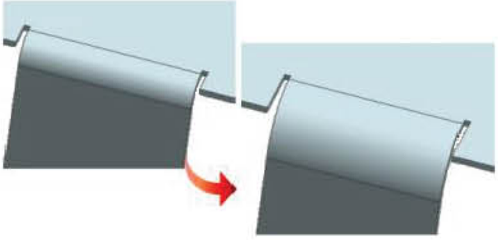


- Bends in partial flanges.

  If you resize the bends in flanges that are created with no relief, NX creates the reliefs wherever required.

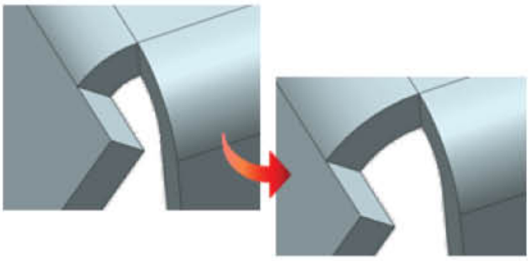


  If you resize the bends in flanges that are created with a relief, the existing reliefs on the bend are maintained.

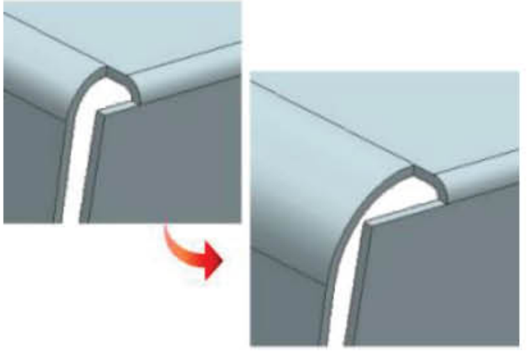


- Two adjacent bends.

  Individually change the radius of the adjacent bends. If you change the radius of one of the bends, the end caps of one bend or both bends are adjusted to keep the corner intact.

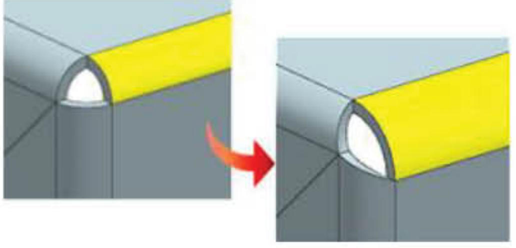


You can also individually change the bend radius of adjacent bends that are not connected, but their bends caps are, and do not form a corner. The bend end caps remain connected after you change the bend radius.



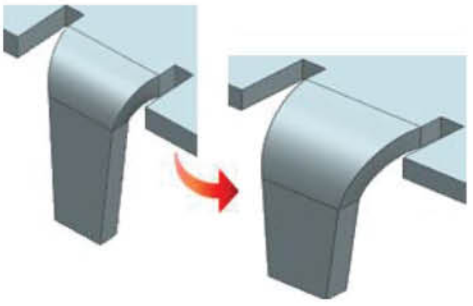
- Three adjacent bends that are connected to form a corner and which do not have any corner treatment.

  You can individually change the radius of the adjacent bends. If you change the radius of one of the bends, the end caps are adjusted to keep the corner intact.



- Bend end caps created by the **Bend Taper** command.

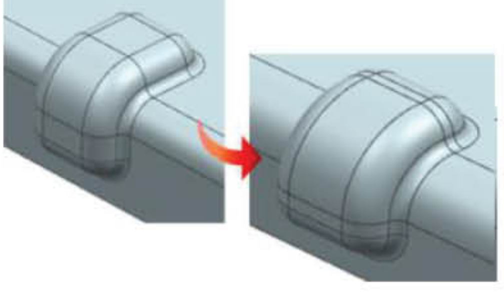
  The example shows a linear bend taper with a square relief. When you increase the bend radius, both the relief and the end caps are adjusted accordingly.



- Bends that have deform features and cutouts created across cylindrical bends.

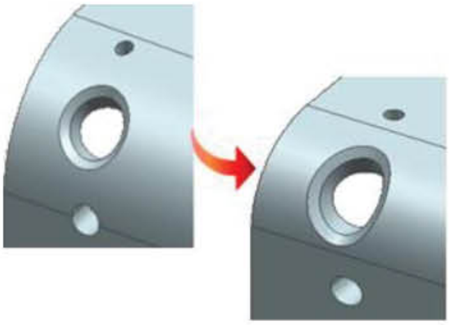
  The feature across the bend region is adjusted accordingly.

  The example shows a Dimple feature created across the bend region perpendicular to the bend centerline. When you increase the bend radius, the Dimple feature is adjusted accordingly.



- Non-uniform thickness bends created by chamfers, blends, counterbored holes, countersunk holes, tapered holes and so on.

  The example shows a countersunk hole created across the bend region. When you decrease the bend radius, the hole is adjusted accordingly.



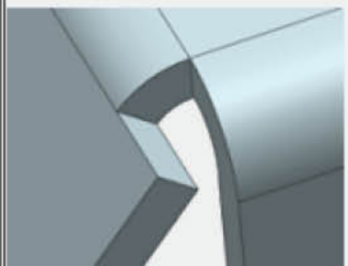
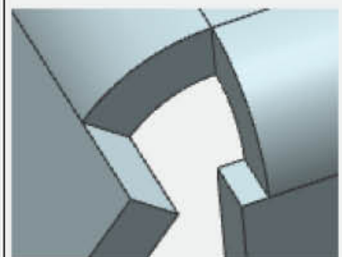
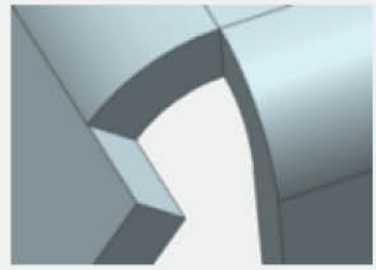
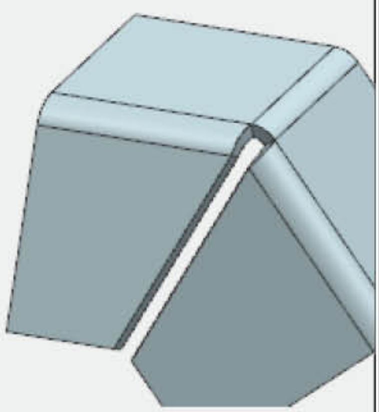
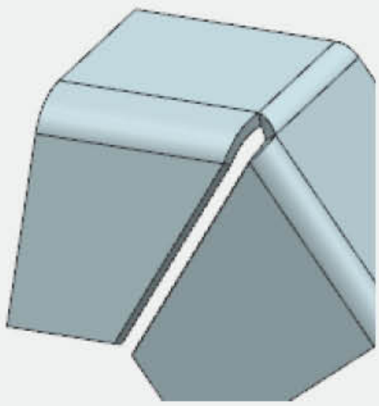
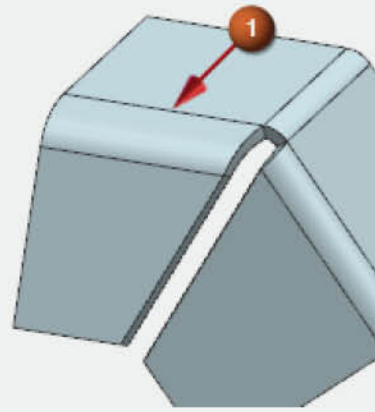
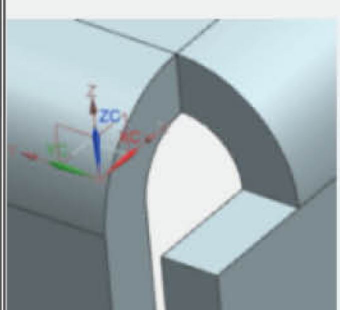
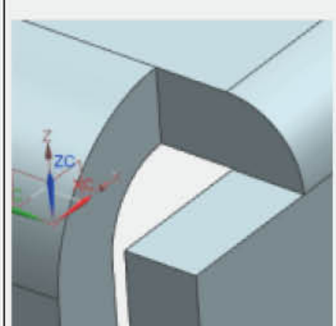
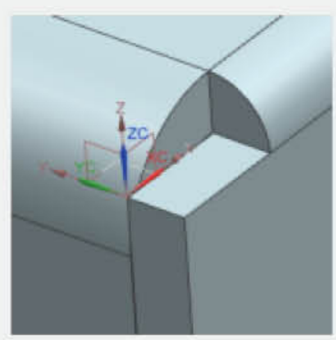
The modifications appear as a separate Resize Bend Radius feature in the Main panel of the **Part Navigator**.

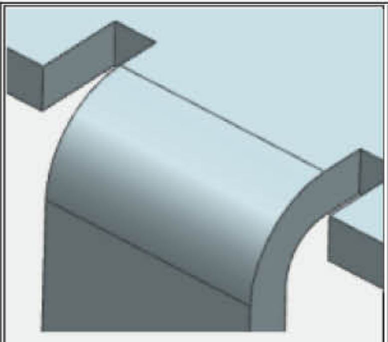
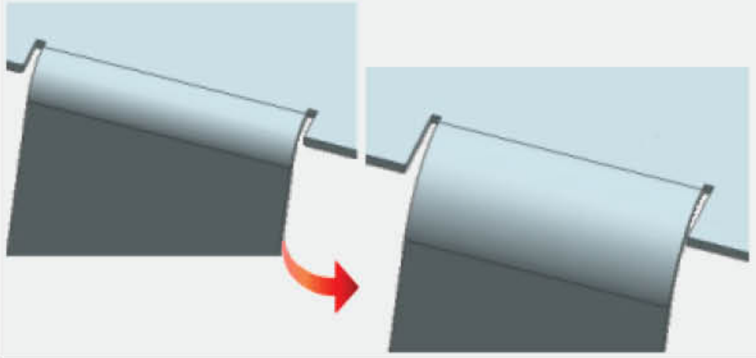
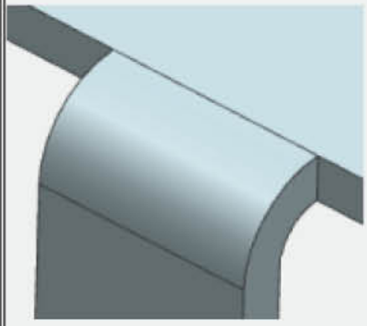
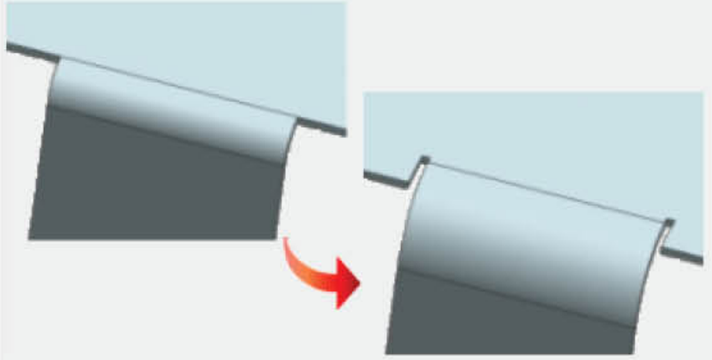
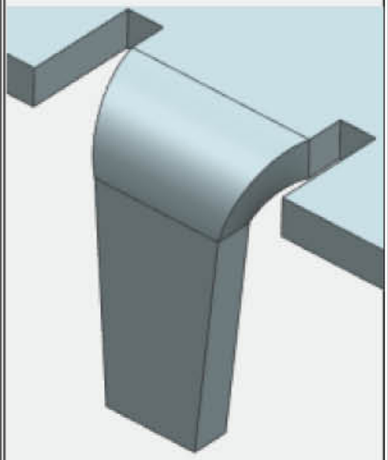
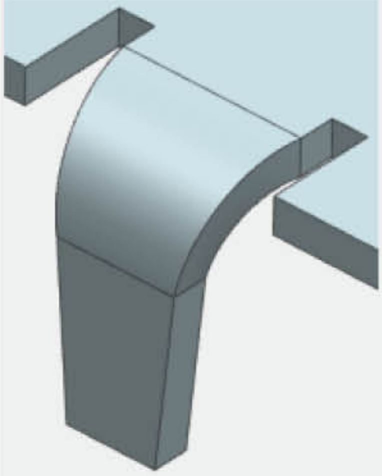
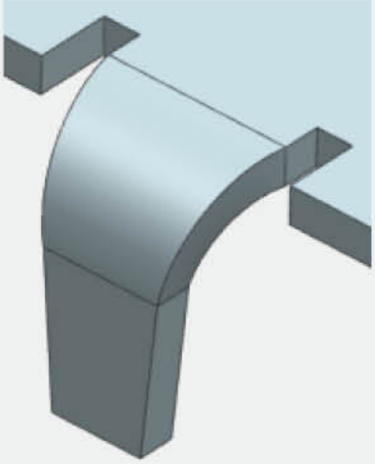
Model History
- Datum Coordinate System (0)
- SB Tab (1)
- SB Flange (2)
- SB Resize Bend Radius (3)

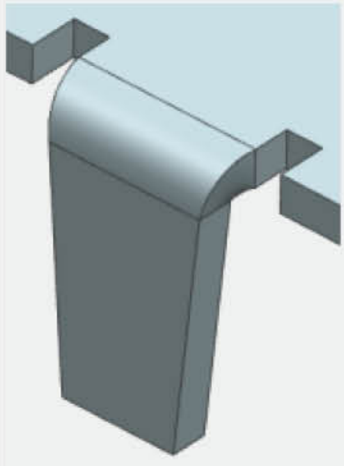
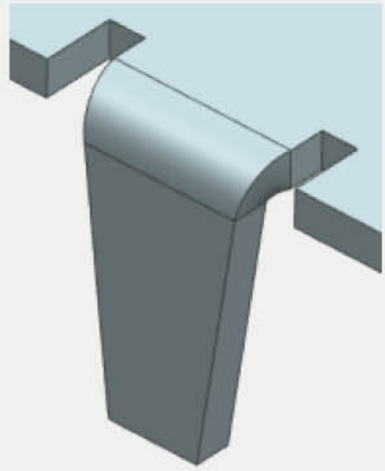
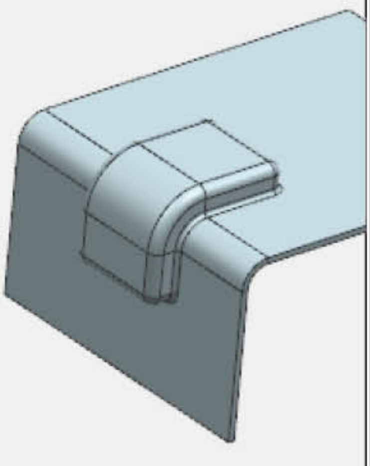
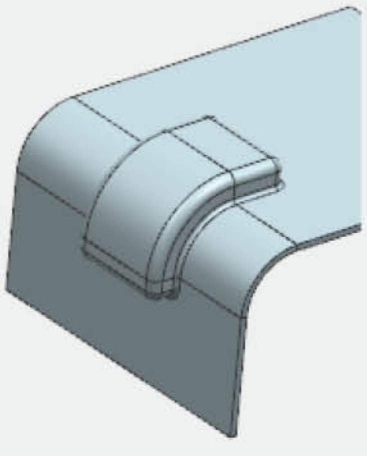
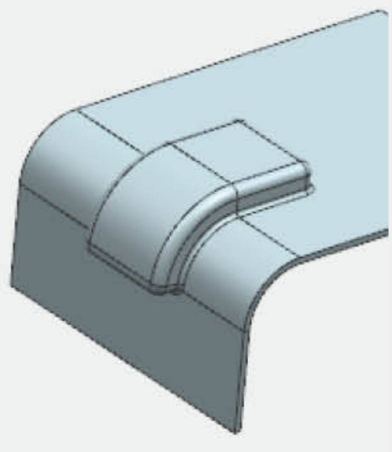
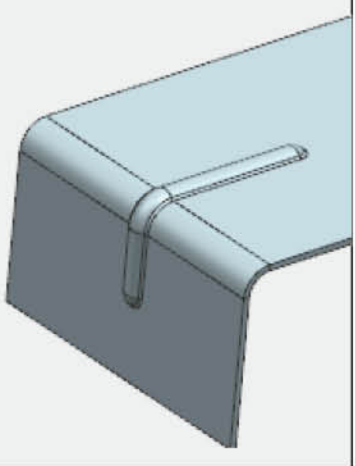
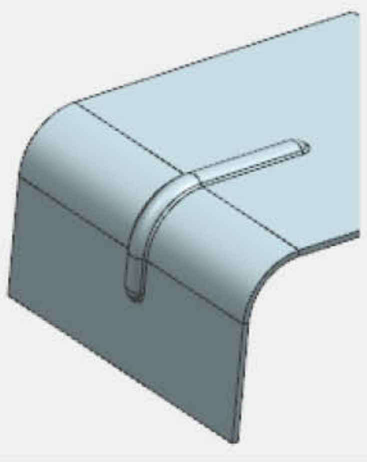
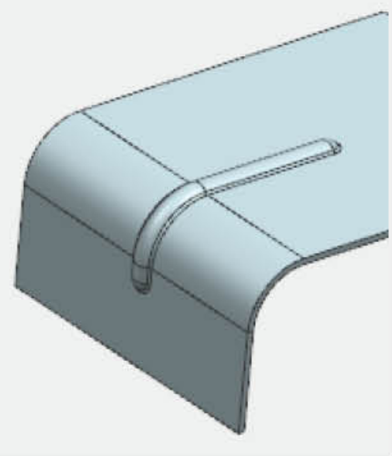
As these features are in timestamp order, you can hide, show, or edit them to display the model in various states of forming.

Depending on the type of Resize Bend Radius feature you create, the overall length, or the length of the unfolded part, remains fixed, or is changed.

Resize Bend Radius — supported geometrical conditions

Input	Result
Bends on adjacent flanges 	**Fixed Tab/ Flange Position**  **Fixed Unfolded Length** 
Three adjacent bends 	**Fixed Tab/ Flange Position**  **Fixed Unfolded Length**  1 — Stationary edge
Zero bend radius support 	**Fixed Tab/ Flange Position**  **Fixed Unfolded Length** 
Partial flanges	If you resize the bends in flanges that are created with a relief, the existing relief on the bend is maintained.

	
Partial flanges, with bend outside 	If you resize the bends in flanges that have no relief, NX creates the reliefs wherever required. 
Bend Tapers 	**Fixed Tab/ Flange Position** — bend radius increased **Fixed Unfolded Length** — bend radius increased   **Fixed Tab/ Flange Position** — bend radius decreased **Fixed Unfolded Length** — bend radius decreased

		
Dimple feature 	**Fixed Tab/ Flange Position** 	**Fixed Unfolded Length** 
Bead feature 	**Fixed Tab/ Flange Position** 	**Fixed Unfolded Length** 
Drawn cutout	**Fixed Tab/ Flange Position**	**Fixed Unfolded Length**

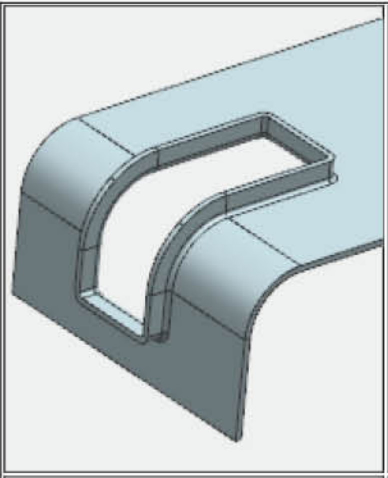
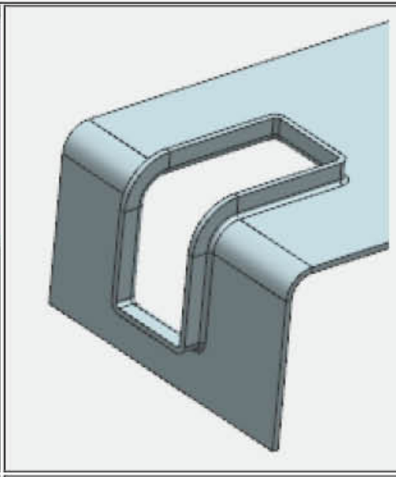
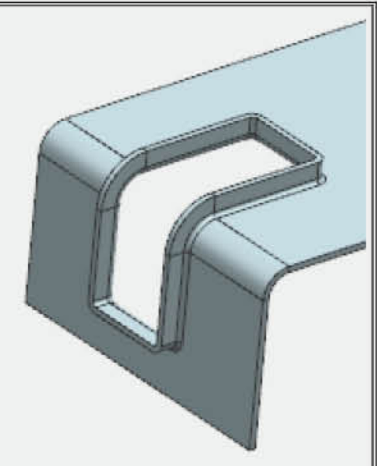
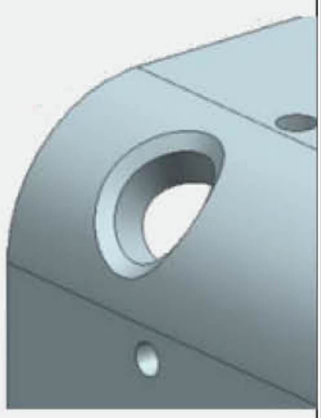
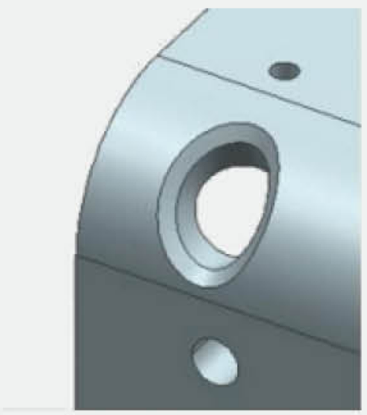
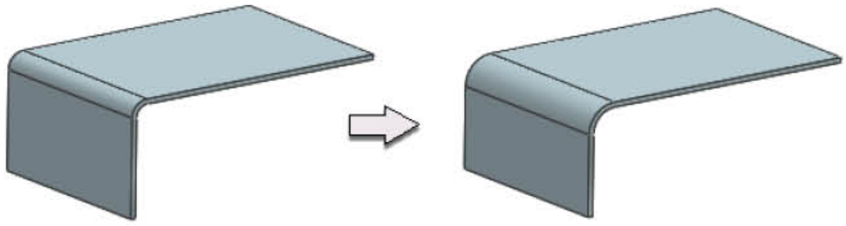
		
Non uniform thickness features	Bend radius increased	Bend radius decreased
		

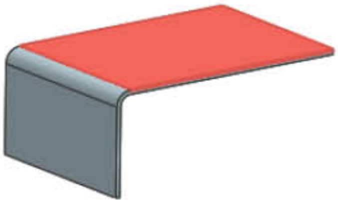
Table 2.2: Resize Bend Radius

Resize the bend radius of a flange

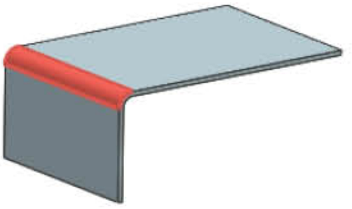
This example shows how to change the size of the bend radius of a flange while keeping the overall length of the unfolded part unchanged.



1. Choose **Home** tab→ **Resize** group→ **Resize Bend Radius** .
2. In the **Resize Bend Radius** dialog box, from the **Type** list, select **Fixed Unfolded Length**.
3. In the graphics window, select the stationary face.

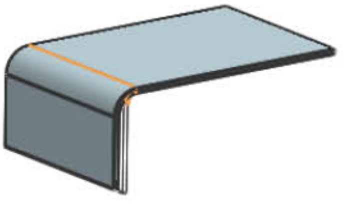


4. Select the bend whose radius you want to modify.



5. In the **Bend Parameters** group, specify a value in the **Bend Radius** box.

   For this example, **Bend Radius** = **5**.



6. Click **OK** to resize the radius of the selected bend.

   Dimensions of the formed part change but the dimensions of the unfolded part remain unchanged.

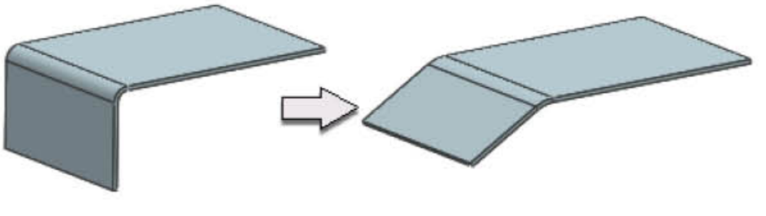
2.3.2.10.2 Resize Bend Angle

Use this command to change the angle of a bend by overriding the feature that created the bend.

You can do the following:

- Depict a bend in various intermediate stages.
- Study over-bending and spring back effects of selected bends.
- Perform forming studies on an already-designed Sheet Metal part.

  You can use the **Part Families** command to create a family of similar parts showing each of the interim forming states of bend regions in Sheet Metal parts.



The Resize Bend Angle feature is listed in the **Part Navigator** as shown in the following graphic.

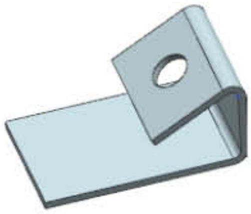
Model History
- Datum Coordinate System
- SB Tab (1)
- Extrude (2)
- SB Bend (3)
- SB Bend (4)
- Resize Bend Angle (5)

Note NX doesn't update the neutral factor value when you update the bend angle.

Resize the bend angle to show interim forming states of a bend region

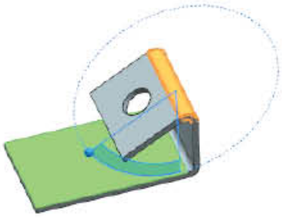
The following example shows how to create Resize Bend Angle features for bend angle values of 90, 120, 150, and 180 degrees.

1. Open the Sheet Metal part for which you want to show multiple stages of the bending operation.

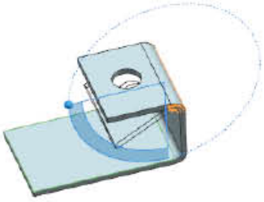


2. Choose **Home** tab→ **Resize** group→ **Resize Bend Angle** .
3. Select a non-thickness face to remain stationary when you modify the bend angle.

4. Select the bend you want to modify.

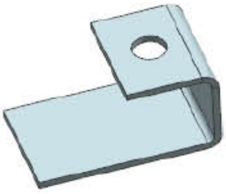


5. In the **Bend Parameters** group, in the **Angle** box, type **90**.

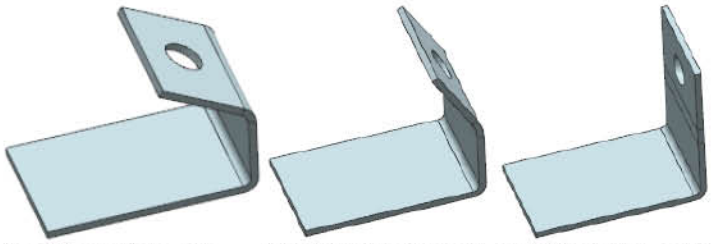


6. Click **Apply**

The bend is modified according to the new value of the bend angle.



7. Repeat steps 3 to 6 to create Resize Bend Angle features using bend angle values of 120, 150, and 180 degrees.



Bend angle = 120 **Bend angle = 150** **Bend angle = 180**

8. Click **Cancel** to close the **Resize Bend Angle** dialog box.

The Resize Bend Angle features are listed in the **Part Navigator** as shown.

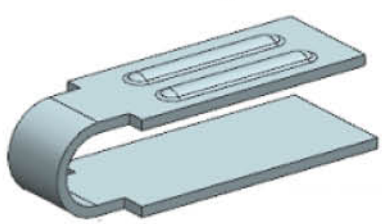
Model History
Datum Coordinate System
SB Tab (1)
Extrude (2)
SB Bend (3)
SB Bend (4)
Resize Bend Angle (5)
Resize Bend Angle (6)
Resize Bend Angle (7)
Resize Bend Angle (8)

You can then use the **Part Families** command to create a family of similar parts showing each of the interim forming states of bend regions in Sheet Metal parts.

2.3.2.11. Stiffening features

2.3.2.11.1 Bead

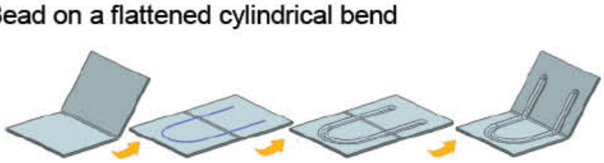
Use the **Bead** command to lift material along the contour of a sketch that simulates a stamping tool. Beads can also be created to stiffen sheet metal parts.



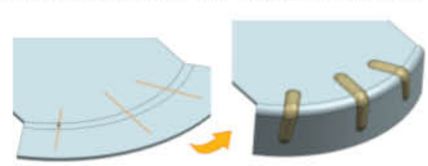
You can create beads on bends in the folded or flattened state. A bead created on a bend in the folded state does not traverse the bend.

You can also create beads on deform features in sheet metal parts.

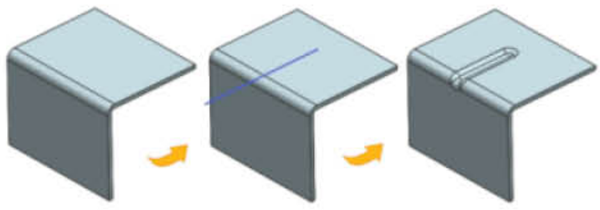
Bead on a flattened cylindrical bend



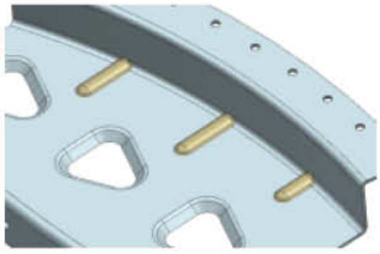
Bead on a flattened non-cylindrical bend



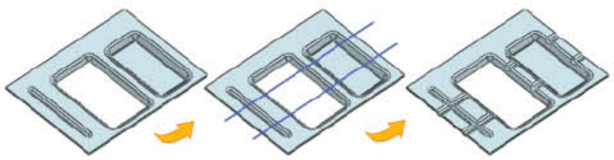
Bead on a formed cylindrical bend



Bead on a formed non-cylindrical bend

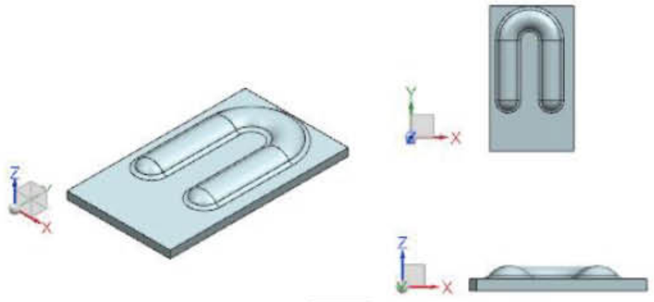


Beads across deform features

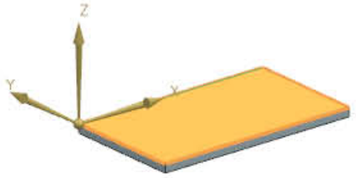


Create a bead

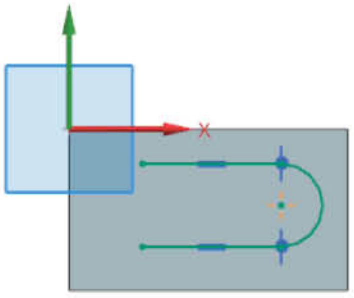
This example shows how to create a bead on a sheet metal part. The bead is created from a sketch. You can either create the sketch first and then select the **Bead** command, or you can select the **Bead** command and then create the bead.



1. Choose **Home** tab→ **Punch** group→ **Bead** .
2. In the **Section** group, click **Sketch Section** .
3. Select a planar face.



4. Sketch the contour of the bead.



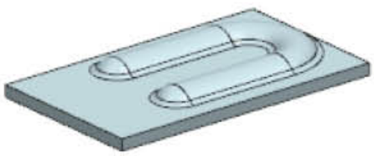
5. Specify the bead parameters.

   In the **Bead Properties** group, set the following:

   - **Cross Section** = **Circular**
   - **Depth** = **3**
   - **Radius** = 3
   - **End Condition** = **Formed**

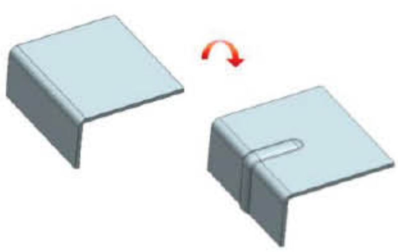
   In the **Rounding** group, set the following:

   - **Round Bead Edges** = 
   - **Die Radius** = **2**
6. Click **OK** to create the bead.



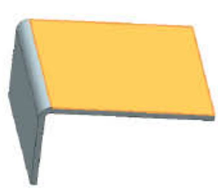
Create beads across bends

This example shows how to create a bead across a bend in a flattened state. You will flatten the bend, create the bead, and then rebend the part to its original state.

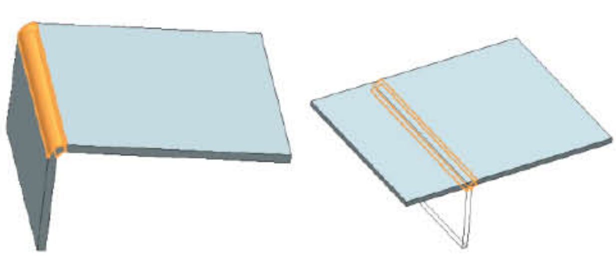


1. Flatten the sheet metal part.

   a. Choose **Home** tab→ **Bend** group→ **Unbend** .
   b. Specify the face that remains fixed during the unbend operation.



   c. Select the bend that you want to flatten.



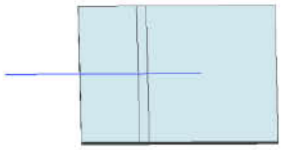
2. Create the bead.

   a. Choose **Home** tab→ **Punch** group→ **Bead** .

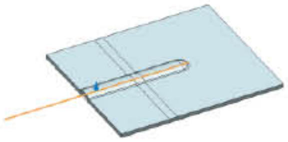
      Note:

      If the command dialog box contains an <**OK**> button, you can select the next command without clicking **OK**. NX automatically completes the current command.

   b. In the **Section** group, click **Sketch Section** .
   c. Sketch a single curve in the middle of the face as shown.



   d. Finish the sketch.

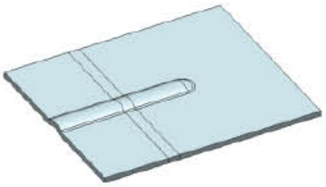


   e. Set the bead parameters.

      This example uses the following parameters:

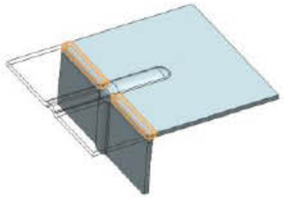
      - **Cross Section = Circular**
      - **Depth = 2**
      - **Radius = 6**
      - **End Condition = Formed**

   f. Click **OK**.



3. Rebend the sheet metal part.

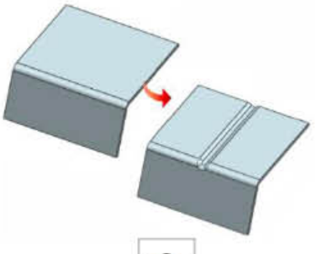
   a. Choose **Home** tab→ **Bend** group→ **Rebend** .
   b. Select the bend that you want to restore to its original state.



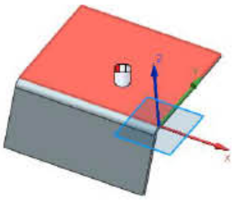
   c. Click **OK**.

Create a bead across bends in the formed state

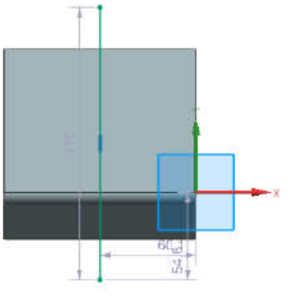
This example shows how to create a bead across a bend in the formed state. When the bends are not flattened, the bead washes out at the bend and does not wrap around the bend.



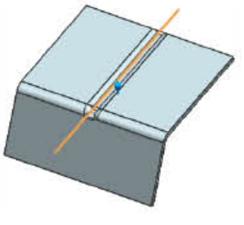
1. Choose **Home** tab→ **Punch** group→ **Bead** .
2. In the **Bead** dialog box, in the **Section** group, click **Sketch Section** .
3. To specify the sketch plane, select the face on which you want to create the bead.



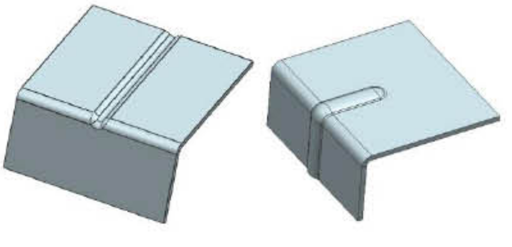
4. Sketch a single curve across the bends.



5. Finish the sketch.



6. Click **OK**.

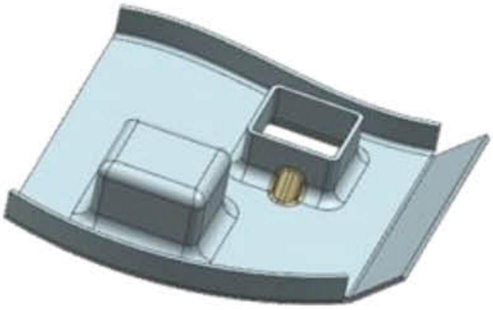


2.3.2.11.2 Gusset

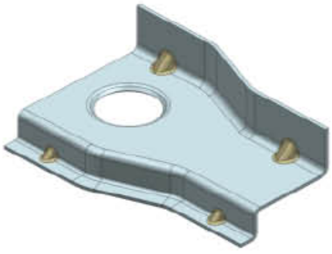
Use the **Gusset** command to create a stiffening feature on a part.

You can:

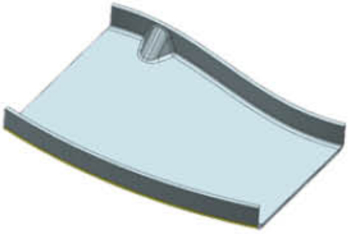
- Create strengthening features where extra strength is required in the bend region.
- Create punched or embossed shapes in bend regions of sheet metal parts.
- Specify default dimensions for the Gusset feature using customer defaults.
- Create gussets on cylindrical blends that are created by deform features such as a dimple, a drawn cutout, a solid punch, and so on.



- Create gussets on cylindrical or non-cylindrical bends that are created by different flanges.



Gussets on cylindrical segments of chained bends

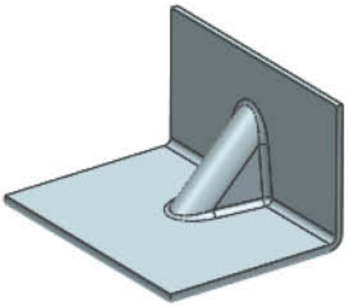


A gusset on a non-cylindrical bend segment

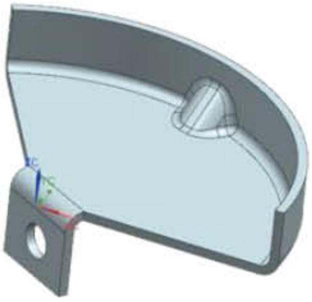
Gusset feature representation in the flat pattern view

Gusset features on cylindrical or non-cylindrical bends are represented as centerline curves in the flat pattern view.

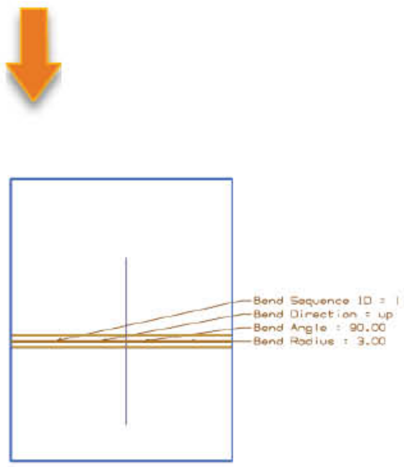
Gusset on a cylindrical bend

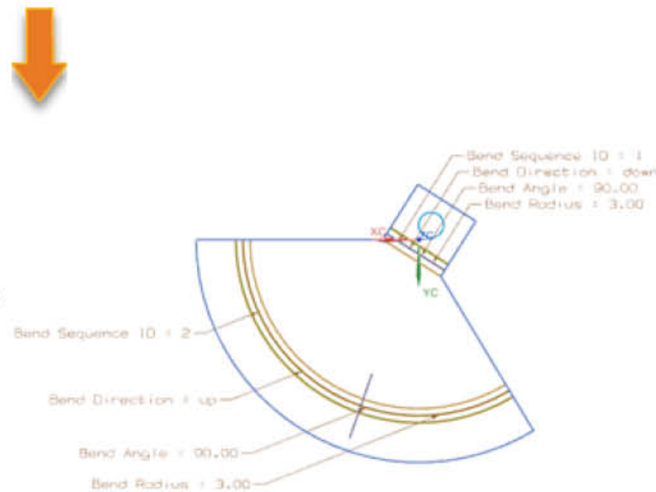


Gusset on a non-cylindrical bend



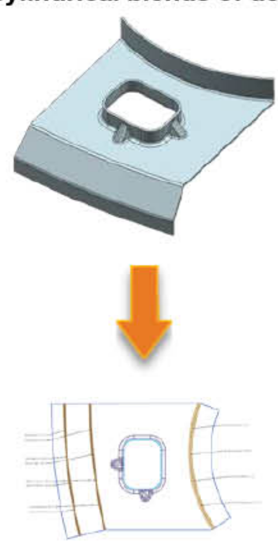
Gusset on a cylindrical bend **Gusset on a non-cylindrical bend**





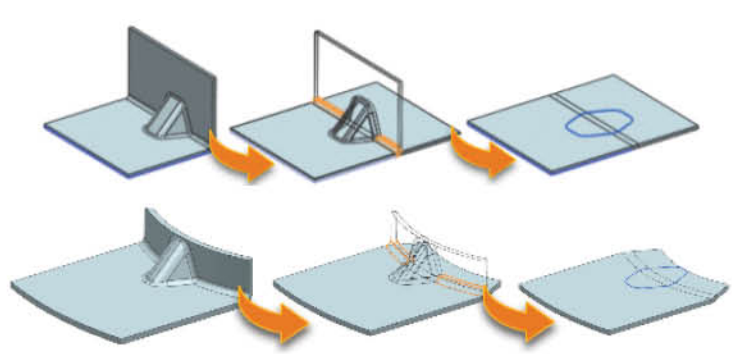
Gusset features on cylindrical blends of deform features are represented as interior feature curves in the flat pattern view.

Gussets on cylindrical blends of deform features



Gusset interoperability

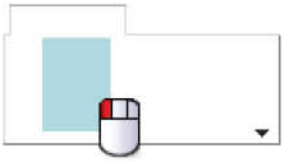
You can view the boundary edges of the gusset as curves on both layers when you flatten the bend using the **Unbend** command.



a. Create a round gusset

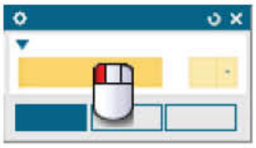
This example shows how to create a round gusset using the automatic profile.

1.



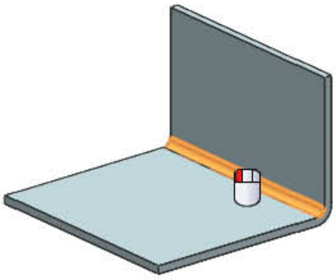
Choose **Home** tab→ **Punch** group→ **Gusset** .

2.



Set **Type** to **Automatic Profile**.

3.

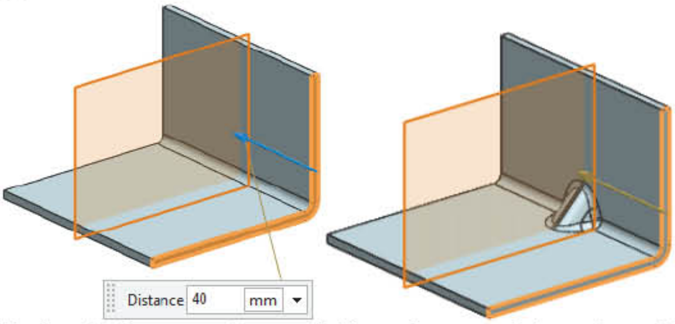


Select the bend face.

4.

Middle-click to advance to the next step.

5.



To locate the gusset, specify the reference datum plane. You can select a datum plane, or create one using the **Specify Plane** options.

NX displays a preview of the gusset.

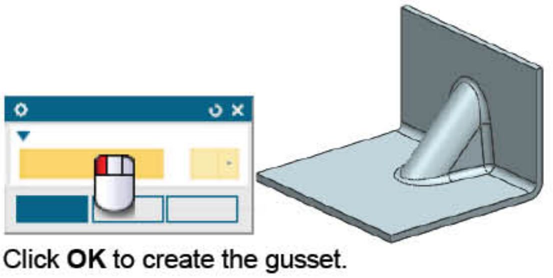
Note:
The datum plane must be perpendicular to the axis of the bend.

6.
In the **Shape** group, set the gusset parameters.

For this example:

- **Depth = 25**
- **Form = Round**
- **Dimensions** subgroup:
    - **Width = 20**
    - **Side Angle = 10**
    - **Die Radius = 3**

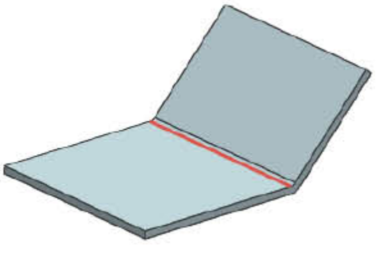
7.



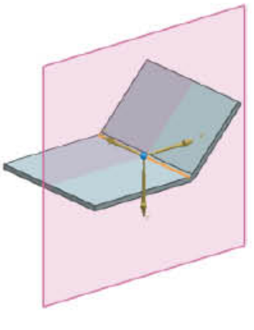
Click **OK** to create the gusset.

b. Create a round gusset with a user defined profile

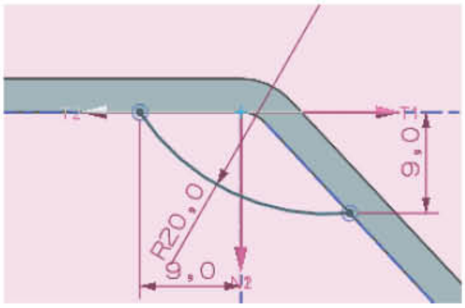
1. Choose **Home** tab→ **Punch** group→ **Gusset** .
2. From the **Type** list, select **User Defined Profile**.
3. In the **Section** group, click **Sketch Section**  and sketch the gusset profile.
4. In the **Create Sketch** dialog box, specify the tangent path along which you want to define the section curve for the gusset.



5. In the **Plane Location** group, from the **Location** list, select **% Arc Length**.
6. In the **% Arc Length** box, type **50** and click **OK**.



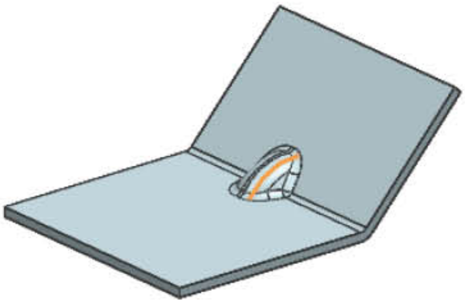
7. Sketch the profile for the gusset.



Note:

The sketch dimensions shown here are approximate.

8. Right-click in the graphics window, and choose **Finish** to exit the Sketch.



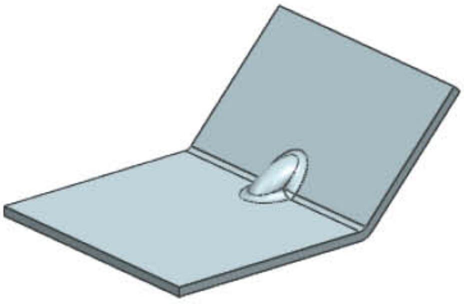
9. In the **Section** group, from the **Width Side** list, select **Symmetric**.
10. In the **Shape** group, set the gusset parameters.

    For this example:

    - **Form** = **Round**
    - **Dimensions** subgroup:
        - **Width** = **10**
        - **Side Angle** = **0**
        - **Die Radius** = **3**

    Note:

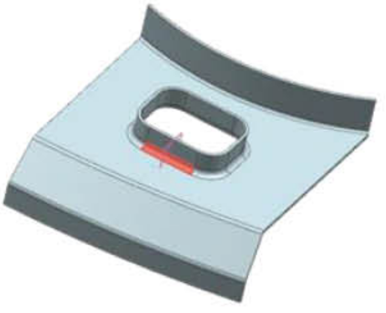
    If the user defined profile has more than one curve, the side angle must be **0**.
11. Click **OK** to create the gusset.



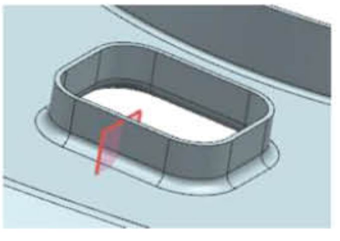
c. Create gussets on cylindrical blends

This example shows how to create gussets on cylindrical blends that are created by deform features such as a dimple, a drawn cutout, or a solid punch, and so on.

1. Choose **Home** tab→ **Punch** group→ **Gusset** .
2. From the **Type** list, select **Automatic Profile**.
3. In the **Bend** group, select the blend face.



4. In the **Location** group, select the plane which you want to use to locate the gusset.



The gusset is created symmetrically on both sides of this plane.

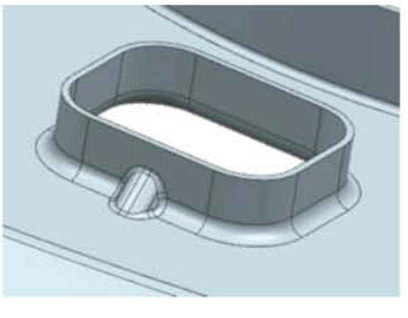
Note:

The plane must be perpendicular to the selected bend segment.

5. In the **Shape** group, set the gusset parameters.

For this example:

 - **Form** = **Square**
 - **Dimensions** subgroup:
   - **Width** = **10**
   - **Side Angle** = **30**
   - **Punch Radius** = **3**
   - **Die Radius** = **3**
6. Click **OK** to create the gusset.

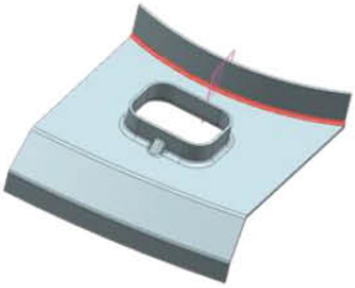


d. Create gussets on non-cylindrical bends

This example shows how to create gussets on non-cylindrical bends that are created by an advanced flange, a chained contour flange, or a joggle.

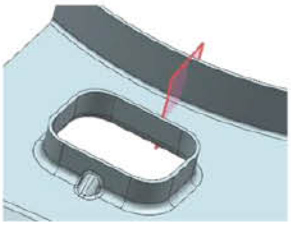
1. Choose **Home** tab→ **Punch** group→ **Gusset** .
2. From the **Type** list, select **Automatic Profile**.
3. In the **Bend** group, select the bend face.

   For this example, we select a bend face created by an advanced flange.

4. In the **Location** group, select the plane which you want to use to locate the gusset.

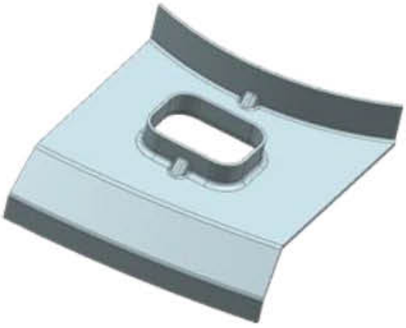
   The gusset is created symmetrically on both sides of this plane.

5. In the **Shape** group, set the gusset parameters.

   For this example:

   - **Form** = **Square**
   - **Dimensions** subgroup:
     - **Width = 10**
     - **Side Angle = 30**
     - **Punch Radius = 3**
     - **Die Radius = 3**
6. Click **OK** to create the gusset.

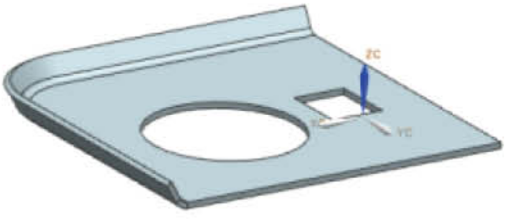
2.4. Advanced sheet metal commands and modify commands

2.4.1 Advanced sheet metal commands and modify commands

2.4.1.1 Advanced Flange

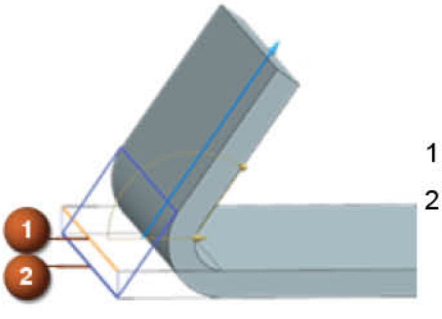
Use this command to add a flange along an edge using a bend angle or a reference face. The edge and reference face can be curved.

You can select a single edge or a chain of tangent edges.



When you create a flange, you can:

- Specify that the flange body is on the inside of the bend and indicate that the material is from the edge opposite the selected reference edge using the **Material Inside OML** option.

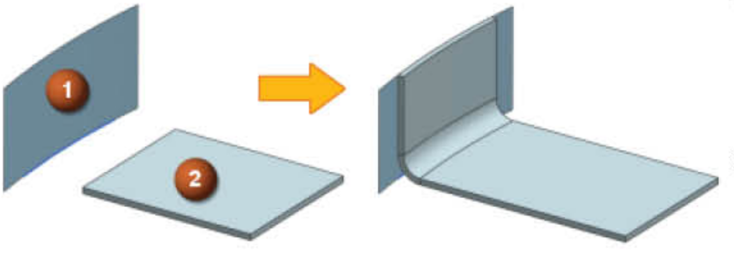


1 Selected reference edge
2 Material inside opposite the selected reference edge

Inset = Material Inside OML

- Define the developed flange length according to the DIN6935 standard using the **Tangent** option.

Flange using a reference face that does not intersect the body



1 Reference face

2 Sheet Metal body

This command is useful when you want to create non-linear flanges, or when you work with sheet metal support structures in complex assemblies.

Note:

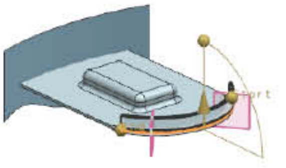
- The advanced flanges that you create on linear edges of a body work like regular flanges.

- To create more accurate flat representations, apply compensation for material deformation to the start and end of the advanced flange edge chain.

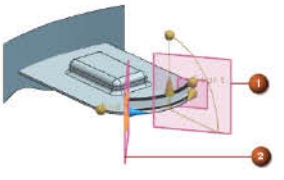
Create an Advanced Flange feature

This example shows how to create an Advanced Flange feature along a non-linear edge. The direction of the flange is along the -ZC axis.

1. Choose **Home** tab→ **Bend** group→ **Advanced Flange** .
2. In the **Advanced Flange** dialog box, from the **Type** list, select **By Value**.
3. In the graphics window, select the base edge for the flange.

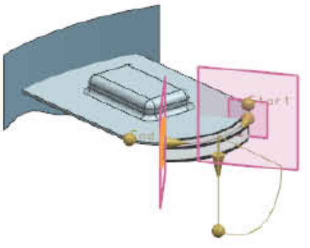


4. In the **End Limits** group, set the following:
   a. Click **Specify Plane 1** and select a plane to specify the start limit of the flange.
   b. Click **Specify Plane 2** and select a plane to define the end limit of the flange.



(1) End plane
(2) Start plane

5. In the **Relief** group, specify the bend relief parameters.
   a. **Bend Relief** = **Square**
   b. **Depth** = **3**
   c. **Width** = **3**
   d. **Corner Relief** = **None**
6. In the **Flange Properties** group, set the following:
   a. Click **Reverse Direction** to specify the flange direction along the -ZC axis.
   b. Specify the flange properties.
      - **Length = 20**
      - **Angle = 90**
      - **Inset = Material Inside**

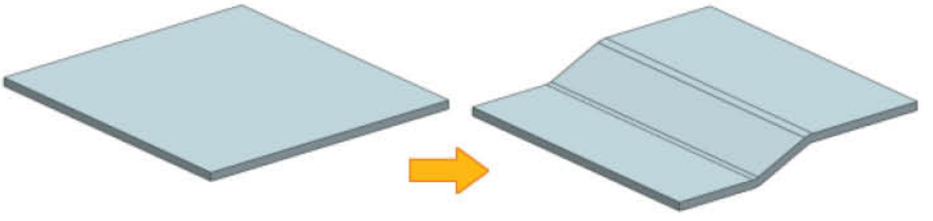


7. Click **OK** or **Apply** to create the Advanced Flange feature.

2.4.1.2 Joggle

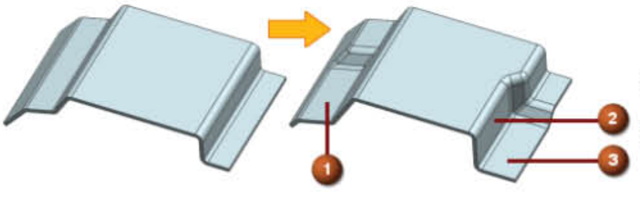
Use this command to offset a portion of a flange or a tab defined by one or two datum planes. You can create a single joggle or twin joggles. You can also apply partial or full compensation to the edges of a joggle to create more accurate flat representations.

Joggle on a tab face



Joggle on a web face

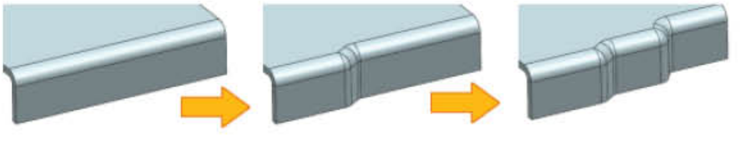
You can select multiple adjacent or non-adjacent web faces simultaneously to create the joggle. You can also specify a different depth value and direction for each web face.



1, 2 Intermediate web Faces

3 End web face

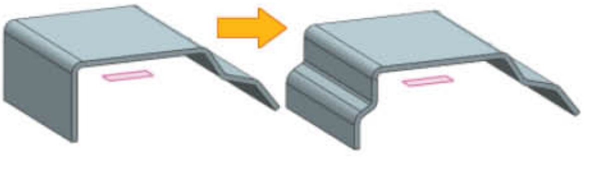
You can also create nested joggles on the same web face.



Note:

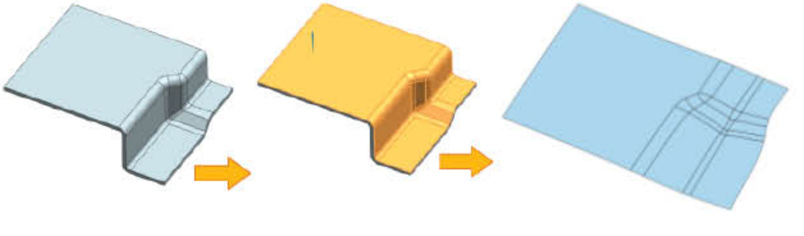
Make sure the joggle transition does not overlap when you create nested joggles.

Joggle across a datum plane that does not intersect adjacent bend faces



Flattening a complex joggle

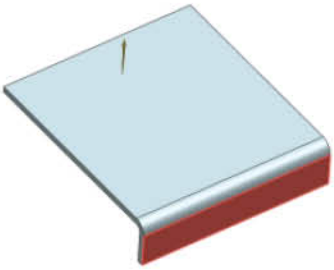
To flatten a joggle created on intermediate or multiple adjacent web faces, use the **Flattening and Forming** command in the Modeling application.



Complex joggle One layer selected Flattened output

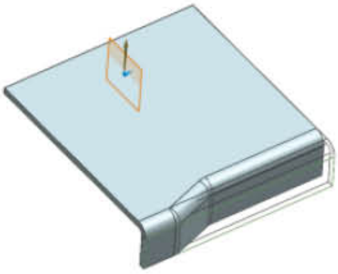
Create a joggle on the web face of a flange

1. Choose **Home** tab→ **Bend** group→ **Joggle** .
2. Select the face of a flange on which you want to create a joggle.



3. In the **Face** group, specify the offset value for the joggle.
   For this example:
   Value = 8
4. In the **Limits** group, specify the joggle type and the start plane.
   For this example:
   Limit Type = Single

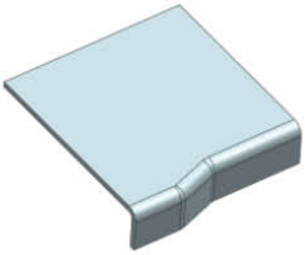
   Specify Start Plane = XC-ZC Plane 



5. In the **Transition** group, in the **Side 1** subgroup, set the joggle properties.

   For this example:

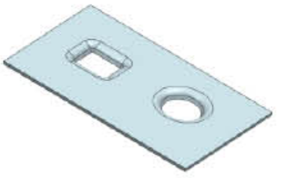
   - **Runout = 25**
   - **Clearance = 0**
   - **Stationary Radius = 5**
   - **Offset Radius = 5**
6. Click **OK** to create the joggle.



2.4.1.3 Lightening Cutout

Use the **Lightening Cutout** command to cut an area of the model inside a sketch that simulates a stamping tool.

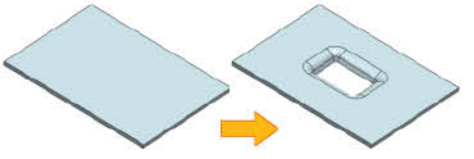
You can create flanged cutouts. You need to define a cutout diameter or create a sketch of a cutout.



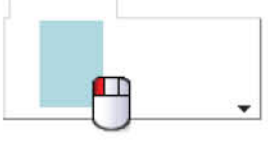
You can also create lightening holes in standard sizes. To specify the standard sizes, use the **Sheet Metal Materials Standards File** customer default.

Create a lightening cutout

This example shows how to create a lightening cutout on a tab using a sketch.



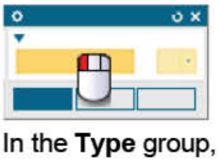
1.



Choose **Home** tab→ **Bend** group→ **Lightening Cutout** .

NX creates a preview as soon as you finish the sketch. If you set the properties first, and then sketch, NX creates the lightening cutout to your specified values.

2.



In the **Type** group, set the following:

- **Type = User Defined** 

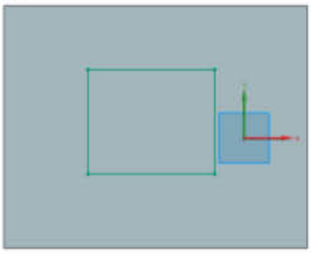
Cutout Properties group:

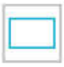
- **Length = 8**
- **Angle = 60**
- **Clearance = 5**
- **Section Corner Radius = 10**

Position group:

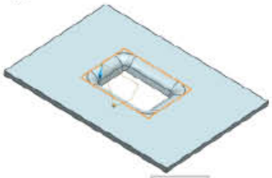
Click **Sketch Section** 

3.



Sketch the cutout section as shown using the **Rectangle**  command.

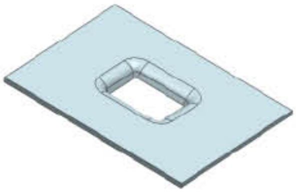
4.



Click **Finish** .

5.

Examine the preview and then middle-click to create the lightening cutout.



2.4.1.4 Meta Form

Use this command to unform complex geometry into an alternate shape. This command also works with non-Sheet Metal features and takes material characteristics into account.

Unform complex geometry

Choose **Home** tab→ **Bend** group→ **Meta Form** .

In the graphics window, select the region to unform.

In the **End Region** group, click **Select Face** and select the target faces to unform.

In the **Transform Geometry** group, select the edges from the unform region.

Specify the layer where you want to place the resulting Meta Form feature.

For this example, **Layer** = **161**.

Specify the boundary conditions.

In the **Boundary Conditions** group, from the **Constraint Type** list, select **Curve-to-Curve**.

In the graphics window, select a curve in the target region to specify the start region curves to be used for mapping.

In the **Boundary Conditions** group, click **Add Boundary Condition** .

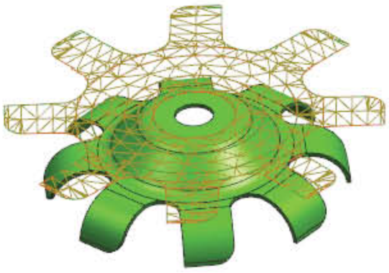
From the **Constraint Type** list, select **Curve-along-Curve**.

Select a curve in the target region to specify the start region curves to be used for mapping.

In the **Thickness** group, select the **Infer Thickness** check box.

In the **Settings** group, accept the default values for the **Material Properties**, **Remove Holes**, and **Tolerances** options.

Click **OK** or **Apply** to create the Meta Form feature.



Validate sheet metal parts

Run validation checks on Sheet Metal parts

This procedure shows how to run the minimum web length validation check on a Sheet Metal part.

Open the Sheet Metal part.

On the Resource bar, click the **HD3D Tools** tab.

On the **HD3D Tools** tab, double-click **Check-Mate**.

In the **Check-Mate** dialog box, in the **Settings** group, click **Set Up Tests**.

In the **Set Up Tests** dialog box, in the **Parts** tab, select **Current Part**.

Click the **Tests** tab and from the **Categories** list, select the tests you want to run.

For this example, expand the Sheet Metal node, click **Check Web Length** and then click **Add to Selected**.

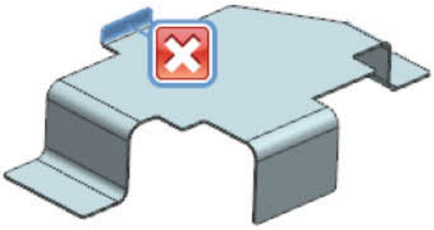
The selected test is added to the **Chosen Tests** list.

Click **Execute Check-Mate** to execute the test.

Click **Close** to close the **Set Up Tests** dialog box.

The validation results are displayed in the **Check-Mate** dialog box, in the **Results** group, and in the graphics window.

The following graphic shows a failed test.

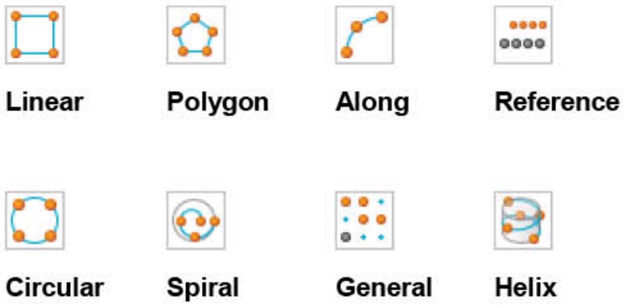


2.4.3 Patterning in sheet metal

2.4.3.1 Pattern Face

Use the **Pattern Face** command to create patterns of faces in various pattern layouts and define pattern boundaries, reference points, orientation, and clocking.

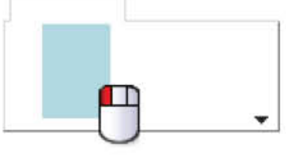
You can create a pattern of faces using a variety of pattern layouts.



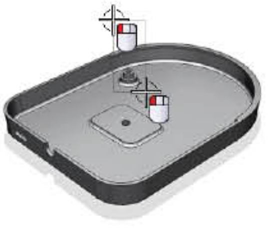
a. Create a rectangular pattern of boss faces

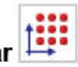
This example shows how to create a linear pattern of the faces of a boss in two directions to create a rectangular pattern of connection bosses.

Choose **Home** tab→ **Feature** group→ **Pattern Face** .



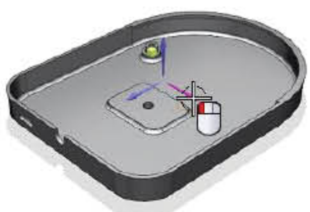
Select the faces as to pattern.



In the **Pattern Definition** group, from the **Layout** list, select or confirm **Linear** .

Middle-click to advance.

In the **Direction 1** subgroup, with **Specify Vector** active, specify an axis for the first direction of the pattern.



Set the pattern spacing parameters.

For this example they are set as shown:

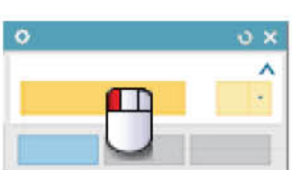
Spacing = Count and Pitch

Count = 3

Pitch Distance =20

Press Enter after specifying a parameter to display the effect of that parameter on the pattern.

In the **Direction 2** subgroup, select the **Use Direction 2** check box .



With **Specify Vector** active, specify an axis for the second direction of the pattern.

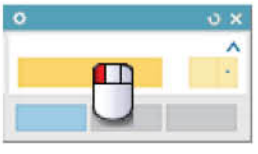
Set the pattern spacing parameters.

For this example they are set as shown:

Spacing = Count and Pitch

Count = 3

Pitch Distance =20

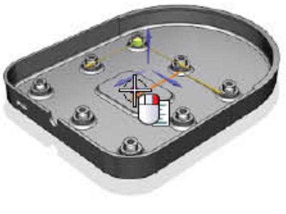


An **Alert** message displays because the reference point of the center instance is located in hole, or void area, that specific instance could not be patterned.

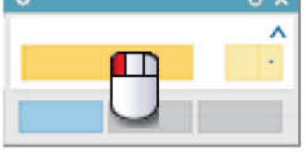
The **Pattern Face** operation can be completed now, but there will be an **Alert** graphic  on the respective **Part Navigator** node.

The instance reference point can also be deleted if it is not needed.

Right-click on the center reference point and select **Delete**.



Click **OK**.

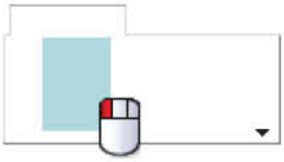


In the **Part Navigator**, the **Pattern Face** node graphic is updated .

b. Create a grip pattern on a knob sleeve

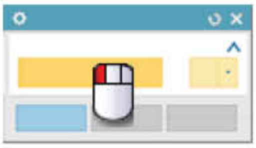
This example shows how to pattern the faces of protrusions and recesses to create a grip pattern on a cylindrical sleeve for a knob.

Choose **Home** tab→ **Feature** group→ **Pattern Face** .



Select the faces as shown.

In the **Pattern Definition** group, from the **Layout** list, select or confirm **Circular** .



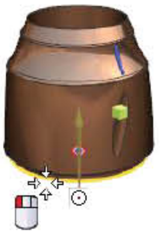
Middle-click to advance.

In the **Rotation Axis** subgroup, with **Specify Vector** active, specify an axis of rotation for the circular pattern.

For this example, the Z axis of the Orient Xpress tool is selected.

With **Specify Point** active, specify a rotation point.

For this example, the arc center of the bottom edge is selected to specify a rotation point.



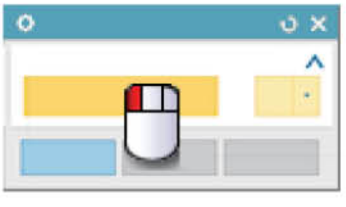
In the **Angular Direction** subgroup, set the pattern spacing parameters.

For this example they are set as shown:

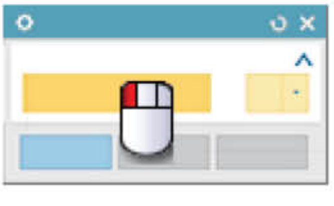
Spacing = Count and Span

Count = 10

Span Angle =360




Click **OK**.

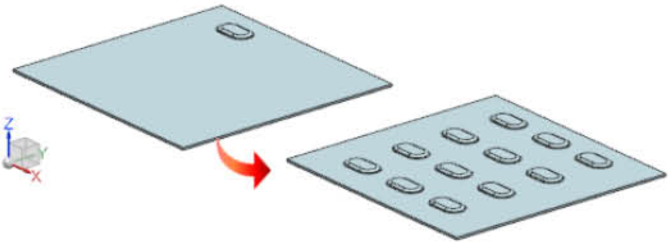



2.4.3.2 Pattern Feature

Use the **Pattern Feature** command to create a pattern of features in layouts such as linear, circular, or polygon, with options for pattern boundary, instance orientation, clocking, and variance.

Create a pattern of Sheet Metal features in a linear layout

This example shows how to create a pattern of a Bead feature in two linear directions.

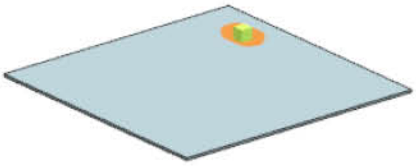


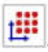
Choose **Insert→ Associative Copy→ Pattern Feature** .

In the **Feature to Pattern** group, **Select Feature**  is active.

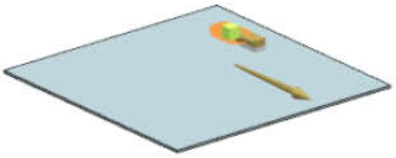
Select the feature to pattern.

For this example, select the Bead feature.



In the **Pattern Definition** group, from the **Layout** list, select **Linear** .

In the **Direction 1** subgroup, from the **Vector** list, select **XC** .



Specify the parameters to create the pattern in direction 1.

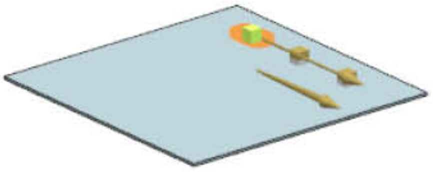
For this example the following parameters are used.

In the **Direction 1** subgroup, set the following:

Spacing = Count and Span

Count = 3

Span Distance = 150



To specify that you want to create the pattern in two directions, in the **Direction 2** subgroup, select the **Use Direction 2** check box.

To specify the second direction in which to create the pattern, from the Vector list, select **-YC**.

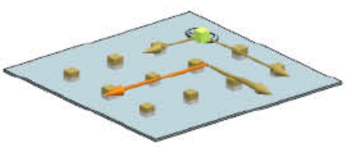
Specify the parameters to create the pattern in direction 2.

For this example the following parameters are used.

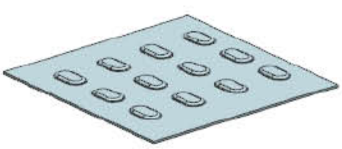
Spacing = Count and Pitch

Count = 4

Pitch Distance = 70



Click **OK** to create the pattern.

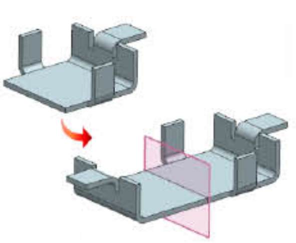


2.4.4 Mirroring in Sheet Metal

2.4.4 .1 Mirror features

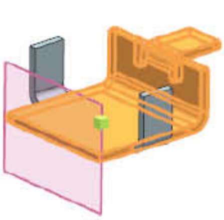
a. Mirror selected features of a part

This example shows how to mirror features shown about an existing datum plane.



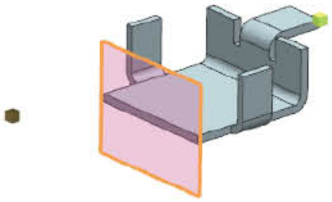
Choose **Home** tab→ **Modeling** group→ **Mirror Feature**.

Select the features that you want to mirror.



In the **Mirror Plane** group, from the **Plane** list, select **Existing Plane**, and then click **Select Plane** 
.

Select the datum plane.

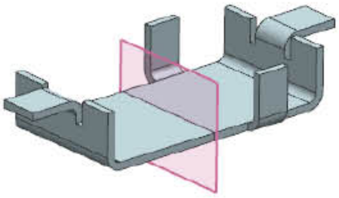


NX displays the mirrored reference point.

If no suitable mirror plane exists, you can select a planar face or set the **Plane** list to **New Plane** and use the **Specify Plane** options to create the mirror plane.

Click **OK**.

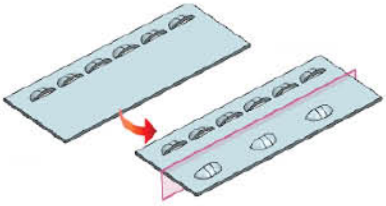
The selected features are mirrored about the specified plane.



NX creates a single Mirror Feature. In the **Part Navigator**, the feature is listed as **Mirror Feature** under the **Model History** node

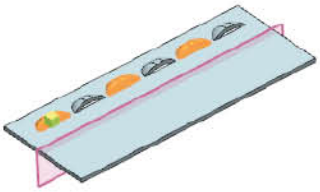
b. Mirror selected instances of a pattern

This example shows how to mirror selected instances of a pattern of Louvre features.



Choose **Home** tab→ **Modeling** group→ **Mirror Feature** .

Select the instances of the pattern you want to mirror.



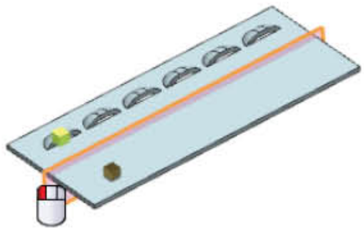
Tip:

To ensure that you select an instance instead of the entire pattern, use the **Quick Pick** dialog box, or select the instance in the **Part Navigator**.

In the **Mirror Plane** group, from the **Plane** list, select **Existing Plane**, and then click **Select Plane** 
.

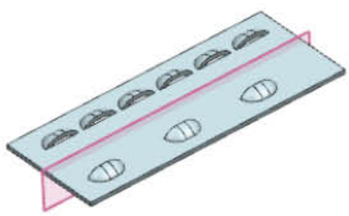
Select the datum plane.

You can also select a planar face.



Click **OK**.

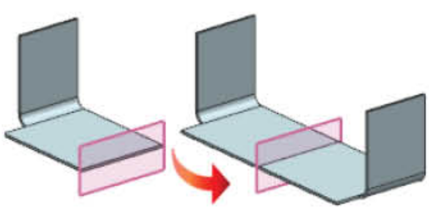
The selected instances are mirrored about the specified plane.



NX creates a single Mirror Feature. In the **Part Navigator**, the feature is listed as **Mirror Feature** under the **Model History** node.

2.4.4.2 Mirror body

Use the **Mirror Body** command to model symmetrical parts by mirroring a body across a datum plane.

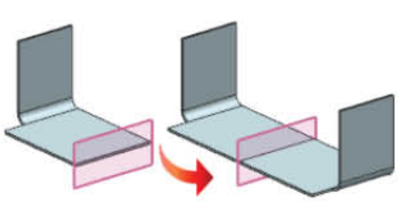


You can unbend and rebend the mirrored body, and perform other sheet metal operations on it such as creating a flat solid or flat pattern.

Because NX retains the sheet metal characteristics during a mirroring operation, you can perform sheet metal operations on the mirrored body without using the **Convert to Sheet Metal** command.

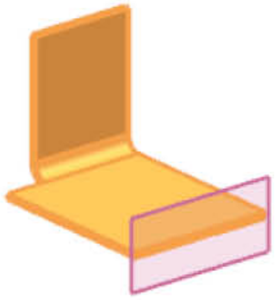
c. Mirror a body

This example shows how to mirror a sheet metal body about an existing datum plane. If a suitable datum plane does not exist, you must first create one using the **Datum Plane** command.



Choose **Home** tab→ **Modeling** group→ **Mirror Body** .

Select the sheet metal body you want to mirror.



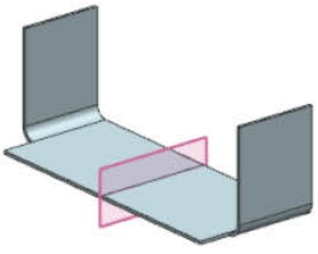
In the **Mirror Plane** group, click **Select Plane**  and then select the mirror plane.

In the **Settings** group, set the following parameters to make the Mirror Body feature associative to the original body and fix it at a current timestamp.

Associative = ☑

Fix at Current Timestamp = ☑

Click **OK** to mirror the selected body.

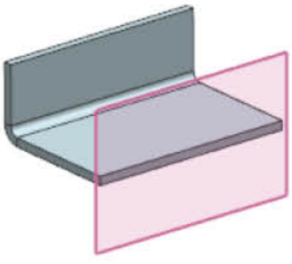


2.4.4.3 Unite

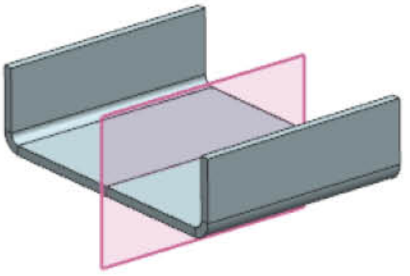
Use the **Unite** command to unite sheet metal bodies.

As the sheet metal characteristics are retained during the unite operation, you can perform sheet metal operations on the resulting body without using the **Convert to Sheet Metal** command.

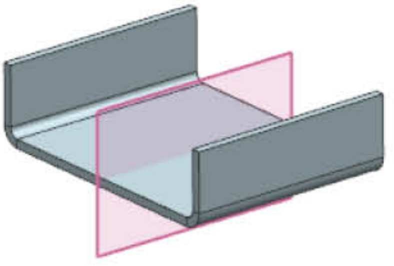
The example shows a typical workflow of mirroring and uniting the mirrored sheet metal bodies.



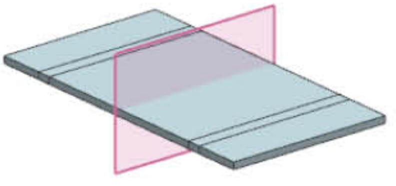
Original sheet metal body



Mirror the sheet metal body



United sheet metal body and the mirrored body



Flatten the united body

2.4.5 Renew Feature

Use the **Renew Feature** command to recompute features created in earlier versions of NX to the current feature version.

Unlike **Update**, which recreates a feature in its original feature version, **Renew Feature** recreates a feature using code from the current NX version.

Renew selected features

Open the part file that contains the features you want to renew.

Choose **Menu**→ **Edit**→ **Feature**→ **Renew Feature** .

From the **Sort** list, select **Feature Type**.

In the **Feature List** column, expand the Feature Type for the features you want to renew.

In the expanded feature list, select a feature to renew.

Click **Apply** to change the selected feature to the current feature version.

The feature version of the selected feature is changed to the current feature version and the item drops off the feature list.

If dependent features do not update correctly or as expected, you can **Undo**  the last operation.

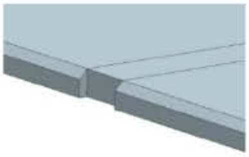
Continue this procedure for other features in the **Feature List** that you want to renew.

When you have finished all renew operations, exit the **Renew Feature** dialog box by clicking **OK**.

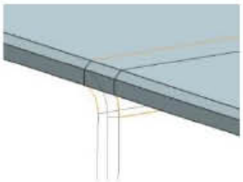
2.4.6 Weld preparations on Sheet Metal parts

Weld preparations on Sheet Metal parts are usually created by applying a Chamfer feature along the edge between the thickness face and the top or bottom face of a part as shown in the following graphic.

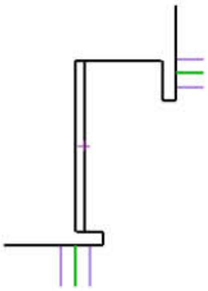
In NX 5 and previous versions of NX, when you apply a chamfer on these edges and then unbend the part, the **Unbend**  command would subtract the chamfered area, as shown in the following graphic.



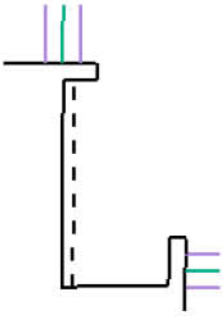
Since NX 6, these areas are now handled properly, and the result is what you would expect for a sheet metal part, as shown in the following graphic.



These new weld preparation edges are also reflected properly in the Flat Pattern feature. If the Chamfer feature is on the same side of the part as the upward reference face you selected for the Flat Pattern feature, the chamfered edge is displayed as an edge, as shown in the following graphic.

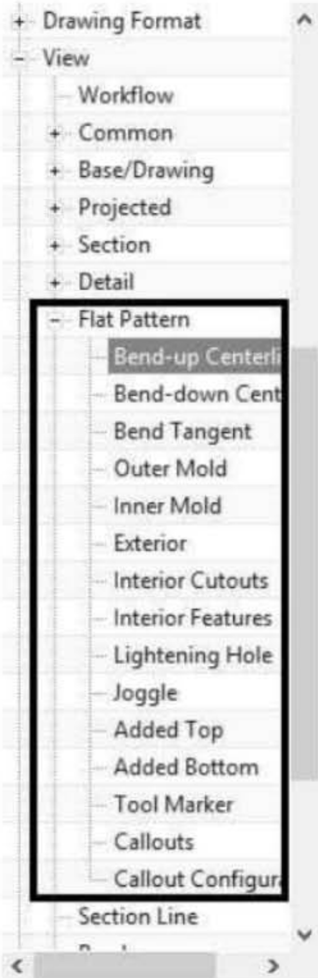


If the chamfered edge is on the side of the part opposite to the reference face, it is displayed as a hidden edge, as shown in the following graphic.

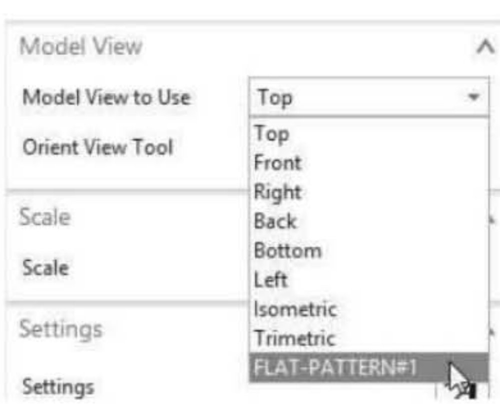


2.5 Sheet Metal Drawings

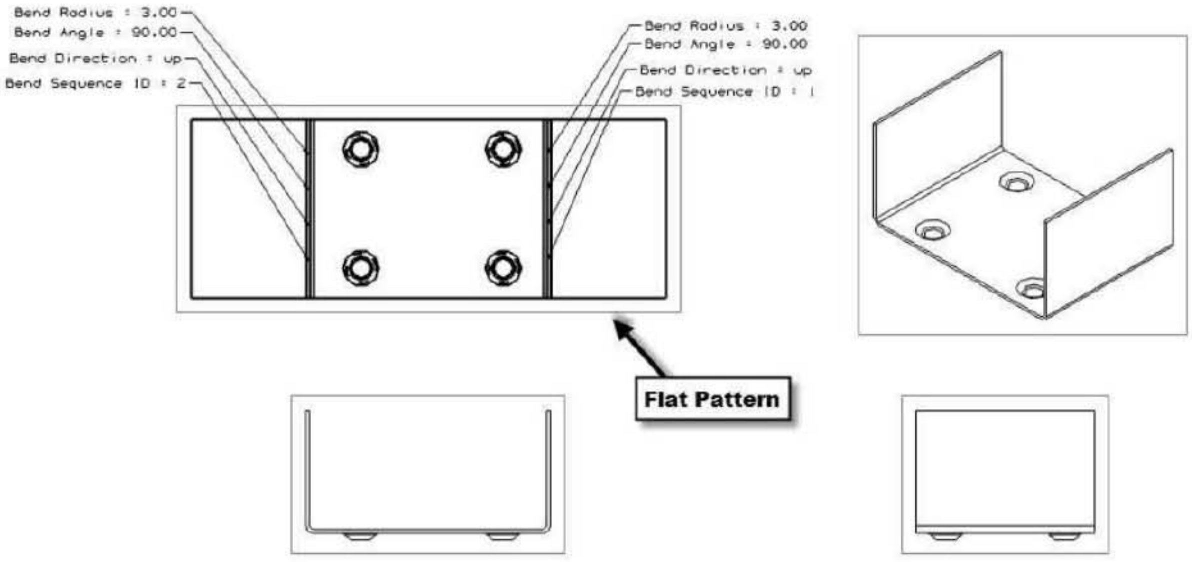
Creating drawings of a sheet metal part is same as creating any other drawing. However, there are some options specific to sheet metal flat pattern. You can access these settings under **View > Flat Pattern** section of the **Drafting Preferences** dialog.



To create a flat pattern view, activate the **Base View** command and select the sheet metal part. On the **Base View** dialog, under the **Model View** section, select **Model View to Use > FLAT-PATTERN#1**

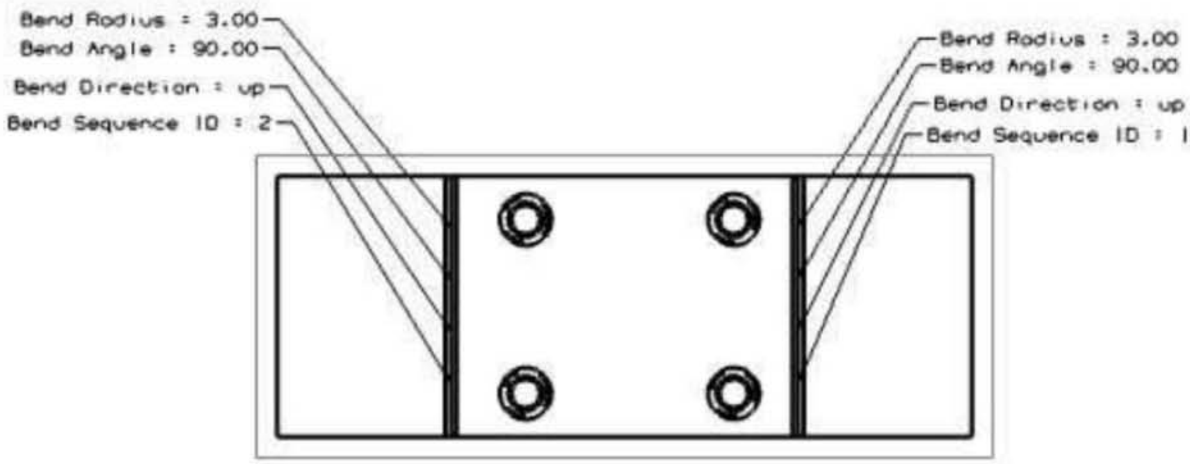


On the dialog, set the **Scale** value and click to place the view. You will notice that lines represent the bends.



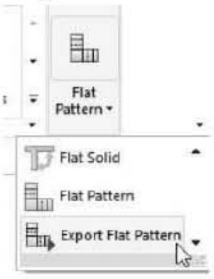
To add a bend table, click **Home > Table > Bend Table** on the ribbon, and then click on the flat pattern view.
Click on the sheet to position the bend table.

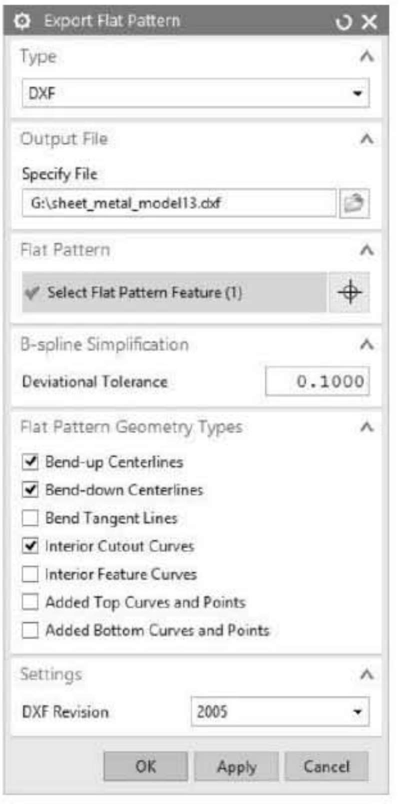
2	Bend 2	3.00	90.00	up	90.00
1	Bend 1	3.00	90.00	up	90.00
ID	Name	Radius	Angle	Direction	Included Angle



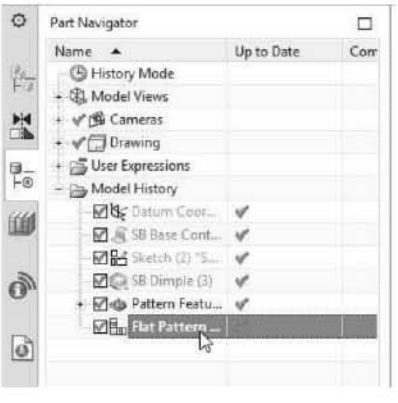
Export Flat Pattern

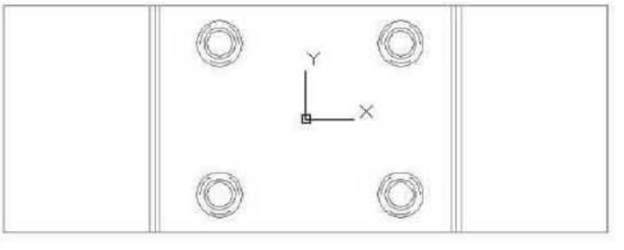
In addition to creating drawings, you can directly export a sheet metal to DWF or Trumpf GEO formats. All you have to do is click **Flat Pattern > Export Flat Pattern**. On the **Export Flat Pattern** dialog, select **Type > DWF** and specify the output file location. Under the **Flat Pattern Geometry Types** section, check the geometry types to export. Under the **Settings** section, select the **DWF Revision** type.





Under **Part Navigator**, click on the **Flat Pattern** feature to export, and the click **OK**.

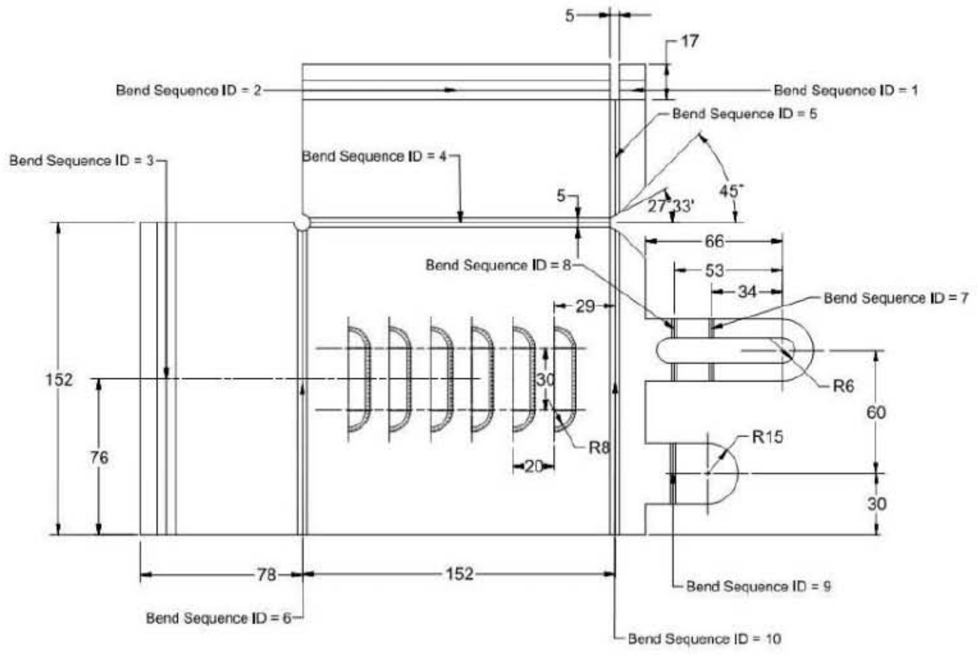


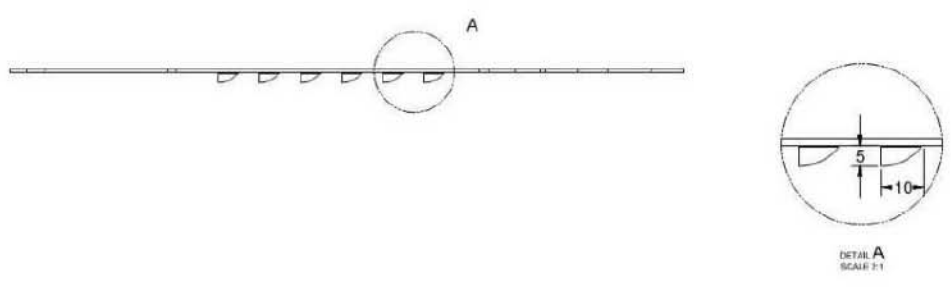


2.6. Practical exercises

2.6.1 Example

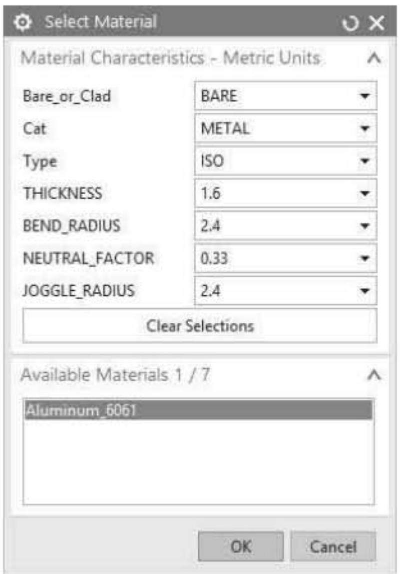
In this example, you will construct the sheet metal part shown below.



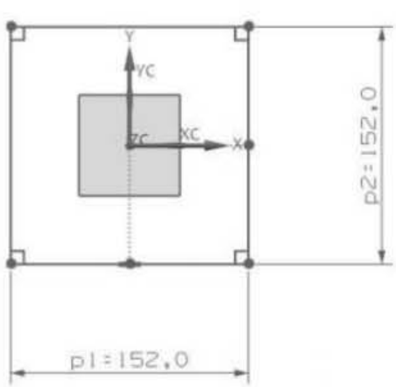


10	Bend 10	2.40	90.00	up	90.00
9	Bend 9	2.40	45.00	down	135.00
8	Bend 8	2.40	45.00	down	135.00
7	Bend 7	2.40	45.00	up	135.00
6	Bend 6	2.40	90.00	up	90.00
5	Bend 5	2.40	90.00	up	90.00
4	Bend 4	2.40	90.00	up	90.00
3	Bend 3	2.00	208.05	down	-28.05
2	Bend 2	2.00	208.05	down	-28.05
1	Bend 1	2.00	208.05	down	-28.05
ID	Name	Radius	Angle	Direction	Included Angle

1.Start **NX 11**.
2. On the ribbon, click the **New** button to open the **New** dialog.
3. On the **New** dialog, select **Units > Millimeters** and click **Sheet Metal**. Click **OK**.
4. On the **Top Border Bar**, click **Menu > Preferences > Sheet Met**

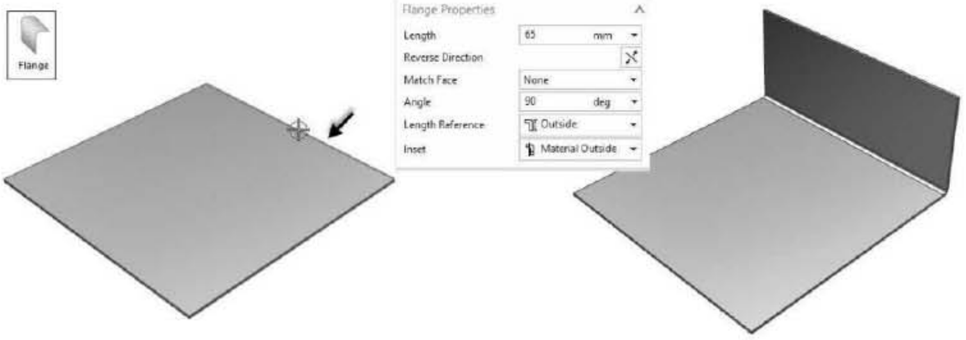


5. On the **Sheet Metal Preferences** dialog, under **Part Properties**, select **Parameter Entry > Material Selection**.
Click **Select Material**.
6. On the **Select Material** dialog, select **Aluminiun_6061** from the **Available Materials** section and click **OK**.
7. On the **Sheet Metal Preferences** dialog, type-in **2.4** in the **Relief Depth** and **Relief Width** boxes. Click **OK**.
8. On the ribbon, click **Home > Basic > Tab** and click on the XY plane.
9. Create a sketch and click **Finish** on the ribbon. On the **Tab** dialog, click **OK** to create the tab feature.

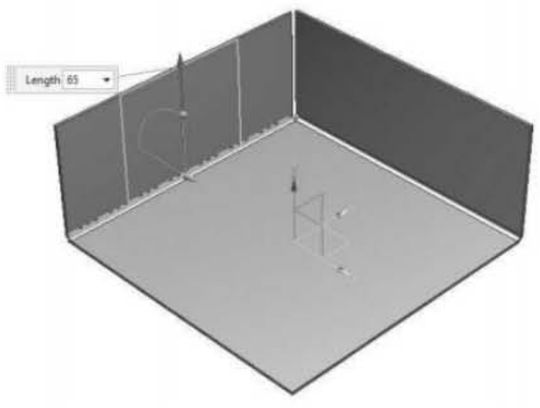


10. On the ribbon, click **Home > Bend > Flange** and click on the back edge.

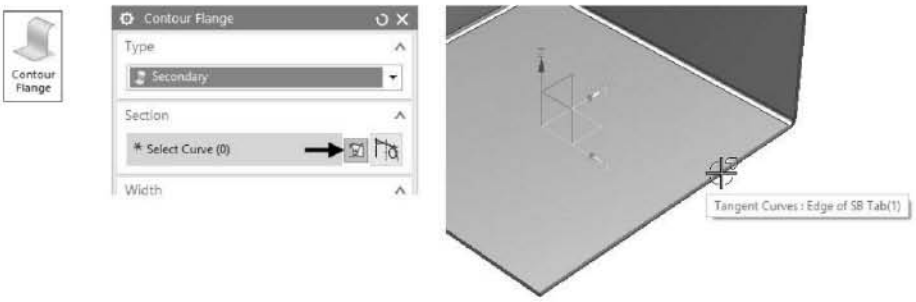
11. On the **Flange** dialog, under **Flange Properties**, select **Length Reference > Outside**. Select **Inset > Material Outside**.
12. Type-in **65** in the **Length** box. Click **OK** to create the flange.



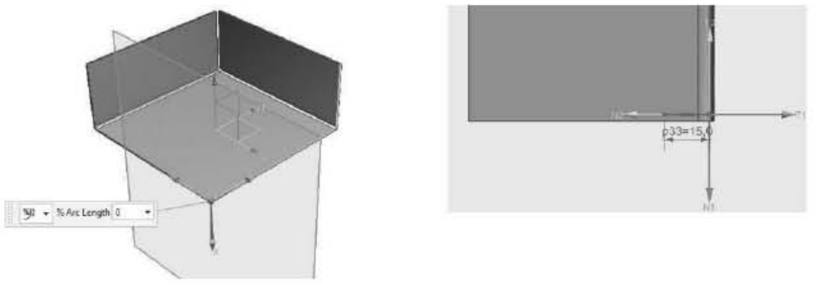
13. Create another flange on the left side. The flange length is 65 mm.



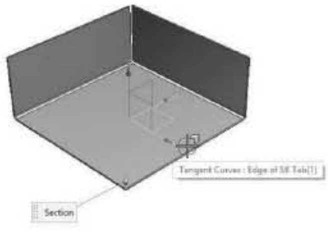
14. On the ribbon, click **Home > Bend > Contour Flange**.
15. On the **Contour Flange** dialog, select **Section > Sketch Section** and click on the right edge of the tab feature.



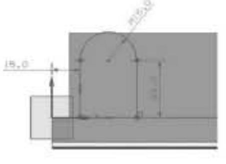
16. In the **%Arc Length** box, type-in **0** and click **OK**.
17. Draw a line of **15** mm length and click **Finish** on the ribbon.



18. On the **Contour Flange** dialog, select **Width Option > Chain**.
19. Click **Select Edge** from the **Width** section and click on the edge of the tab, as shown in figure. Click **OK** to
complete the flange.

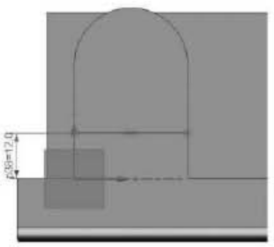
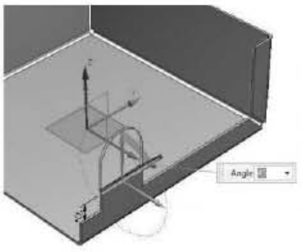
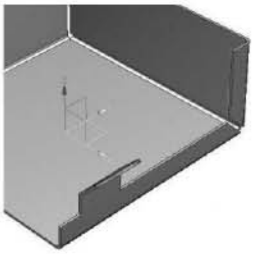


20. On the ribbon, click **Home > Basic > Tab**.
21. Click on the outer face of the contour flange and draw the sketch shown below.
22. Click **Finish** on the ribbon.
23. On the **Tab** dialog, select **Type > Secondary**, and then click **OK** to create a tab feature.

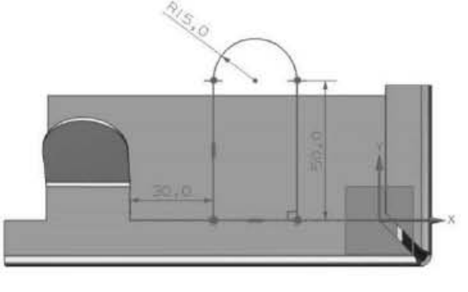



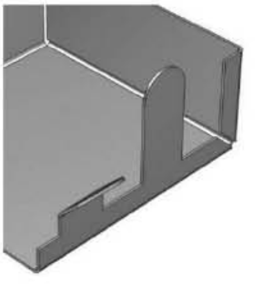

24. Activate the **Sketch** command, and then select **Sketch Type > On Plane**. Next, click on the outer face of the
tab, and then click **OK**.
25. Draw a horizontal line at 12 mm distance above the top edge of the contour flange. Next, click **Finish Sketch**.
26. Activate the **Bend** command (On the ribbon, click **Home > Bend > More > Bend**) and click on the line.
27. On the **Bend** dialog, type-in **45** in the **Angle** box.

28. Use the Reverse Side button to make sure that the output matches that shown below. Next, click **OK** to bend the tab feature.

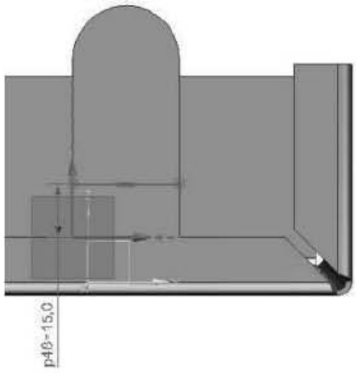




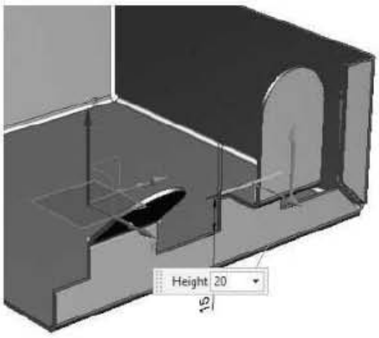
29. Draw another sketch on the outer face of the contour flange.
30. Activate the **Tab** command and create a tab feature using the sketch.

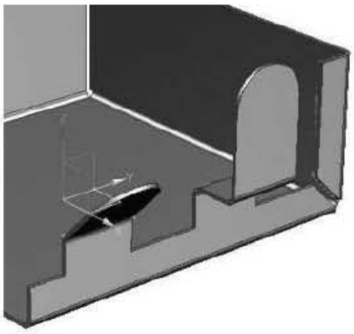




31. Draw a line on the outer face of the tab feature. Next, 15mm dimension between the line and the top edge of
the contour flange feature. Click **Finish Sketch** to exit the sketch.
32. Activate the **Jog** command (click **Home > Bend > More > Jog** on the ribbon) and click on the sketched line.
33. On the **Jog** dialog, select **Height Reference > Outside** and **Inset > Material Outside**.
34. Make sure that the **Extend Section** option is selected.
35. Type-in **20** in the **Height** box and click **OK** to add jog to the tab feature.

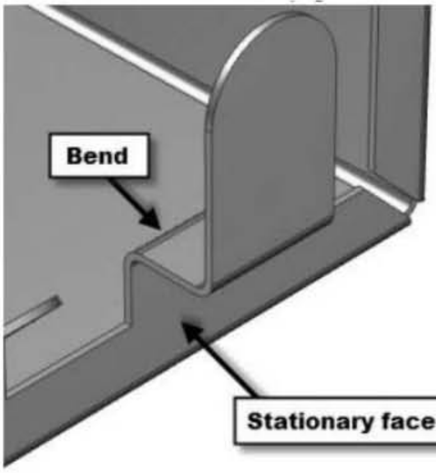


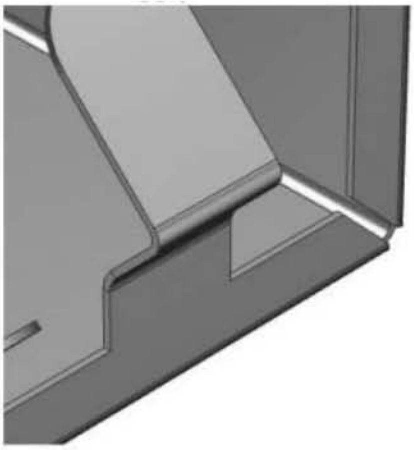




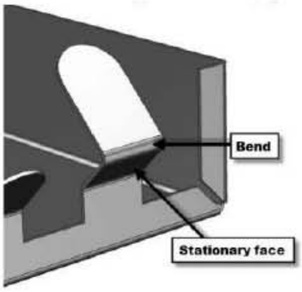
36. On the ribbon, click **Home > Form > Resize Bend Angle**  .

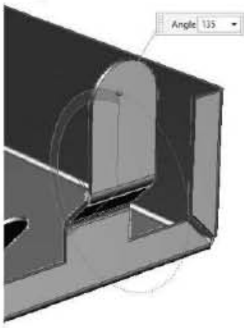
37. Click on the outer face of the contour flange feature to define the stationary face.
38. Click on the lower bend of the jog feature and type-in **135** in the **Angle** box. Click **Apply**.



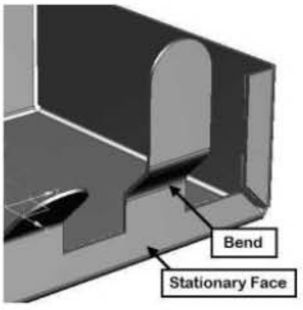


39. Click on the inclined face of the jog feature to define the stationary face.
40. Click on the upper bend of the jog and type-in 135 in the **Angle** box. Click **OK**



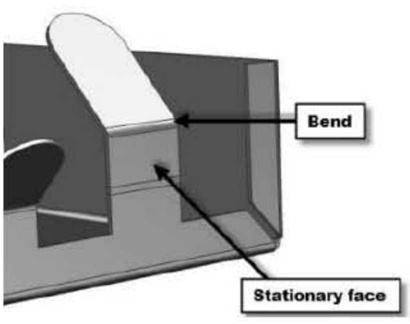


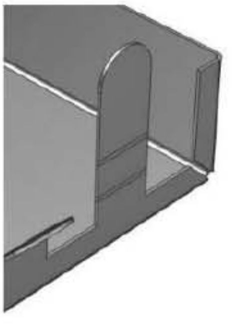
41. On the ribbon, click **Home > Form > Unbend** .
42. Click on the outer face of the contour flange feature to define the stationary face.
43. Click on the lower bend of the jog feature. Click **Apply** to unbend the jog



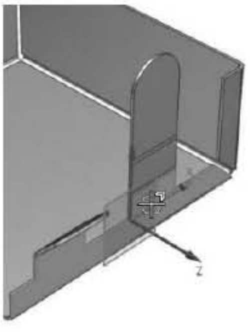
44. Click on the face located between two bends of the jog to define the stationary face.

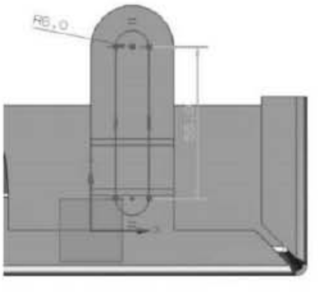
45. Click on the upper bend and click **OK**.



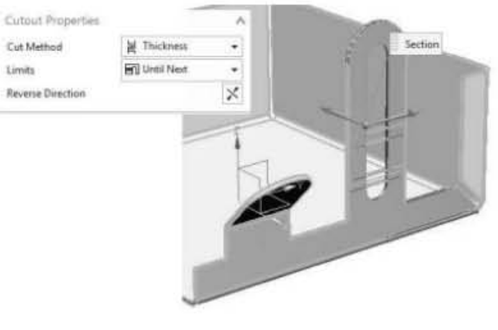


46. On ribbon, click **Home > Feature > Normal Cutout** .
47. Click on the outer face of the unbent jog feature. Create the sketch and click **Finish** on the ribbon.

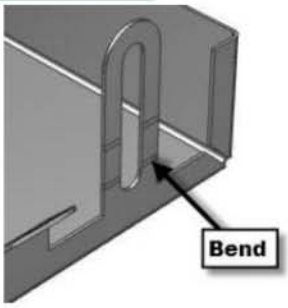


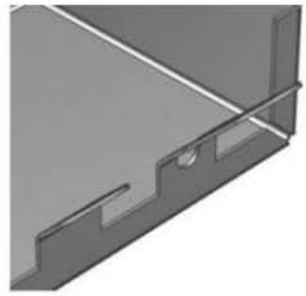


48. On the **Normal Cutout** dialog, select **Limits > Until Next**. Click **OK** to complete the normal cutout feature.

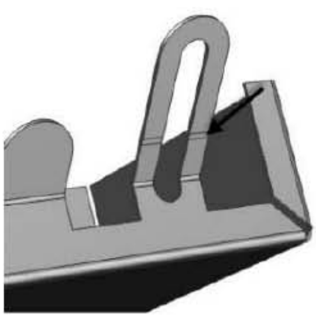


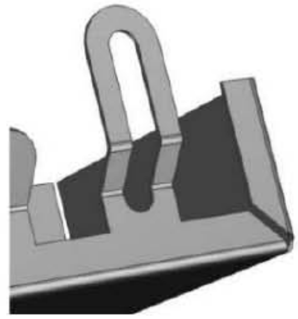
49. On the ribbon, click **Home > Form > Rebend** and click on the lower bend face of the jog feature. Click
Apply to rebend the feature



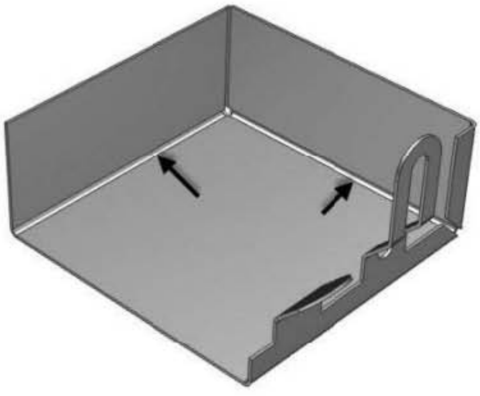


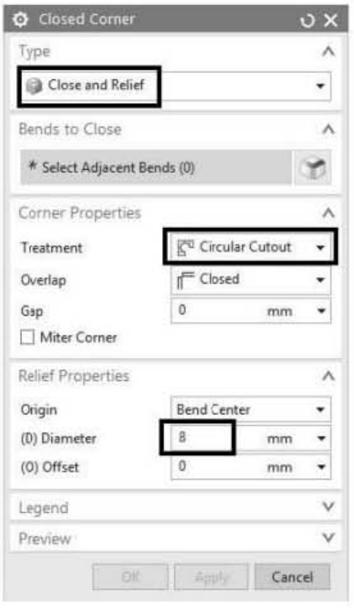
50. Click on the upper bend face, and click **OK**.



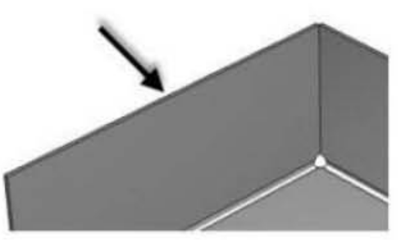


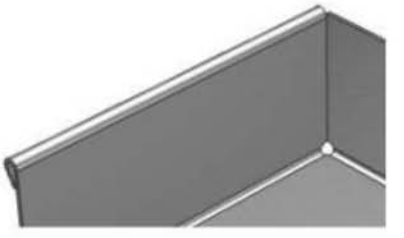
51. Activate the **Closed Corner** command (click **Home > Corner > Closed Corner** on the ribbon) and click on
the bends of the flange features.
52. On the **Closed Corner** dialog, select **Type > Close and Relief**. Under **Corner Properties** section, select
Treatment > **Circular Cutout**.
53. Under the **Relief Properties** section, set the **(D) Diameter** value to 8 mm. Click **OK** to close the bends.



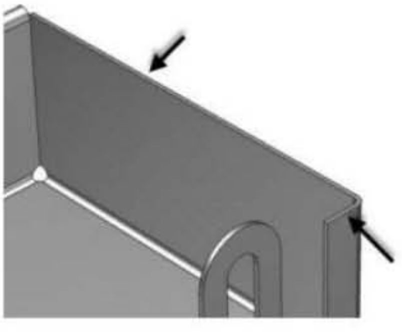


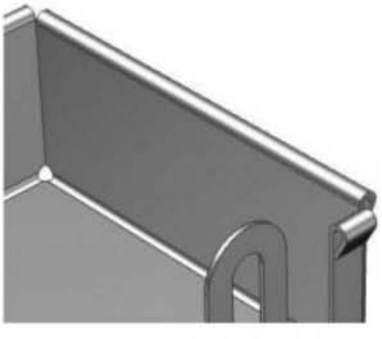
54. Activate the **Hem Flange** command (On the ribbon, click **Home > Bend > More > Hem Flange** )
55. On the **Hem Flange** dialog, select **Type > Closed Loop**.
56. Under **Inset Options**, select **Inset > Material Outside**.
57. Set **Bend Radius** to 2 and **Flange length** to 8.
58. Click on the outer edge of the left-side flange, and click **Apply**



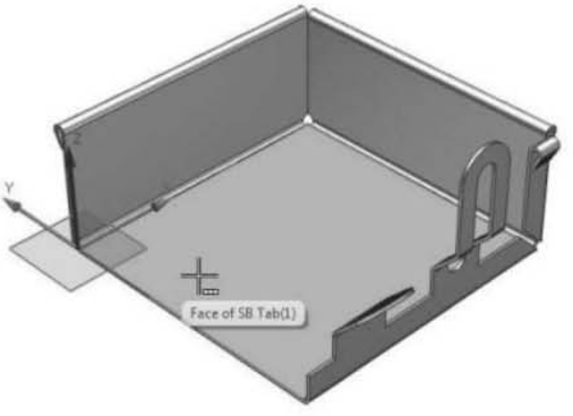


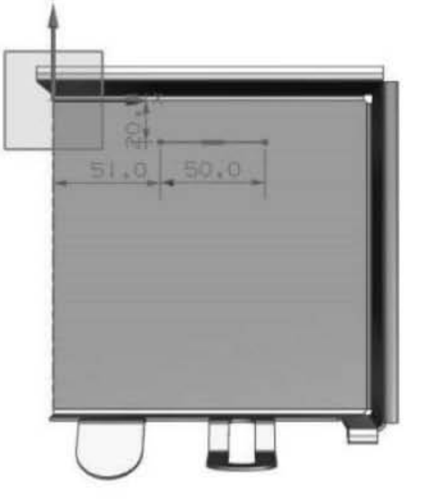
59. Click on the outer edges of the backside flange and contour flange. Click **OK** to create the hem features.





60. Activate the **Louver** command (click **Home > Punch > Louver** on the ribbon) and click the **Reset** icon on
the Louver dialog.
61. Click on the top face of the tab feature.
62. Draw the sketch shown in figure and click **Finish** on the ribbon.

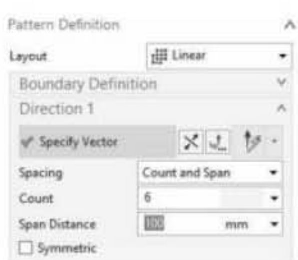


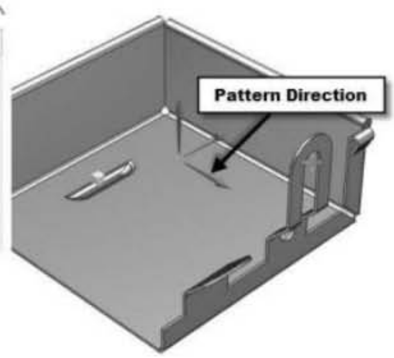


63. On the **Louver** dialog, type-in **5** in the **Depth** box and click the **Reverse Direction** button next to it.
64. Type-in **10** in the **Width** box.

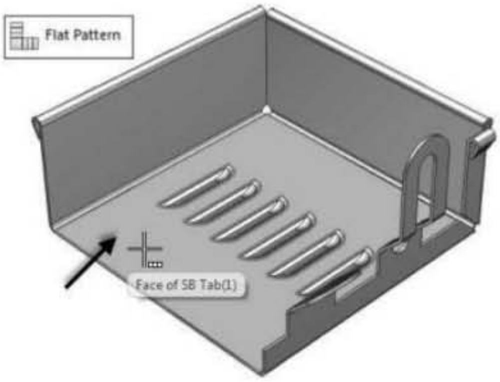
65. Under the **Louver Properties** section, select **Louver Shape > Formed**.
66. Expand the **Louver** dialog and check the **Round Louver Edges** option under the **Rounding** section. Type-in **1** in the **Die Radius** box and click **OK** to create the louver.

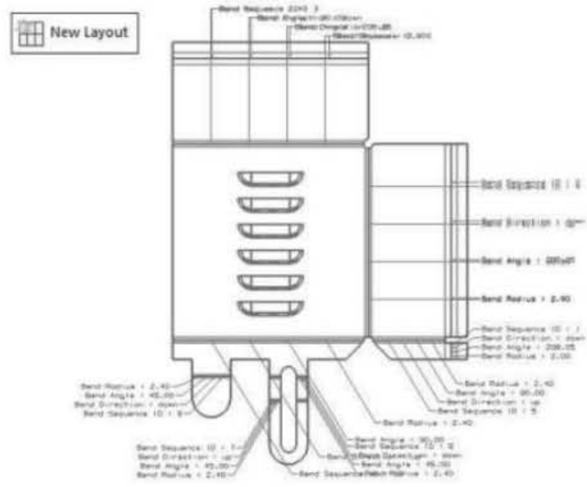
67. Activate the **Pattern Feature** command (On the ribbon, click **Home > Feature > Pattern Feature**) and click louver feature.
68. On the **Pattern Feature** dialog, select **Layout > Linear.**
69. Under the **Direction 1** section, click **Specify Vector** and click on the X-axis vector.
70. Under the **Direction 1** section, select **Spacing > Count and Span**. Type-in 6 and 100 in the **Count** and **Span Distance** boxes.
71. Click **OK** to create the linear pattern of the louver.





72. On the ribbon, click **Home > Flat Pattern > Flat Pattern**.
73. Click on the top face of the tab feature. Uncheck the **Move to Absolute CSYS** option and click **OK** on the **Flat**
Pattern dialog to create the flat pattern. Next, click **OK** on the **Sheet Metal** message.
74. On the ribbon, click **View > Orientation > More > New Layout**.
75. On the **New Layout** dialog, click FLAT-PATTERN#1 and click **OK** to view the flat pattern.

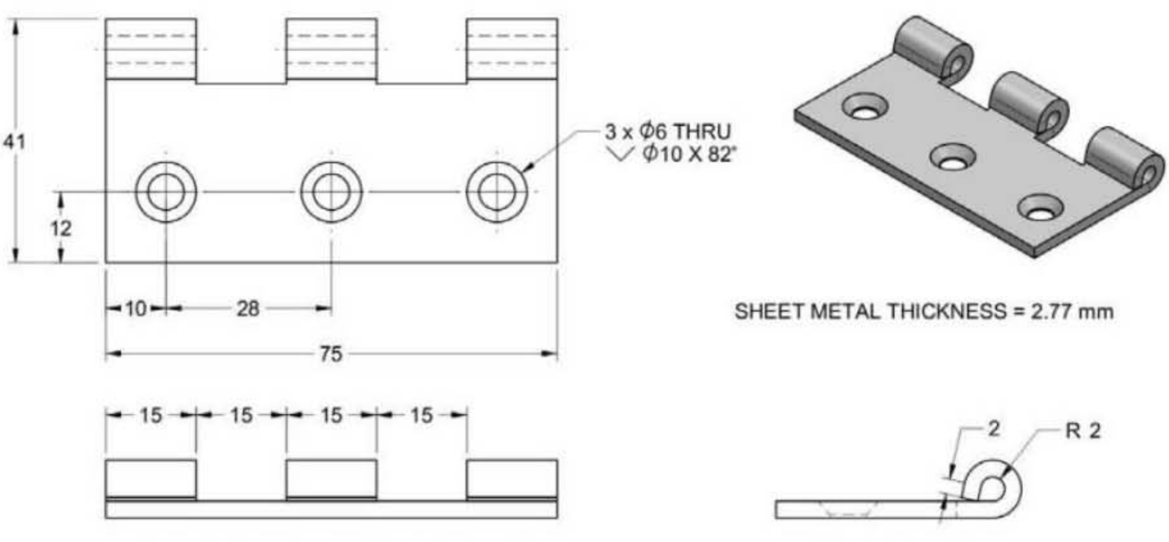




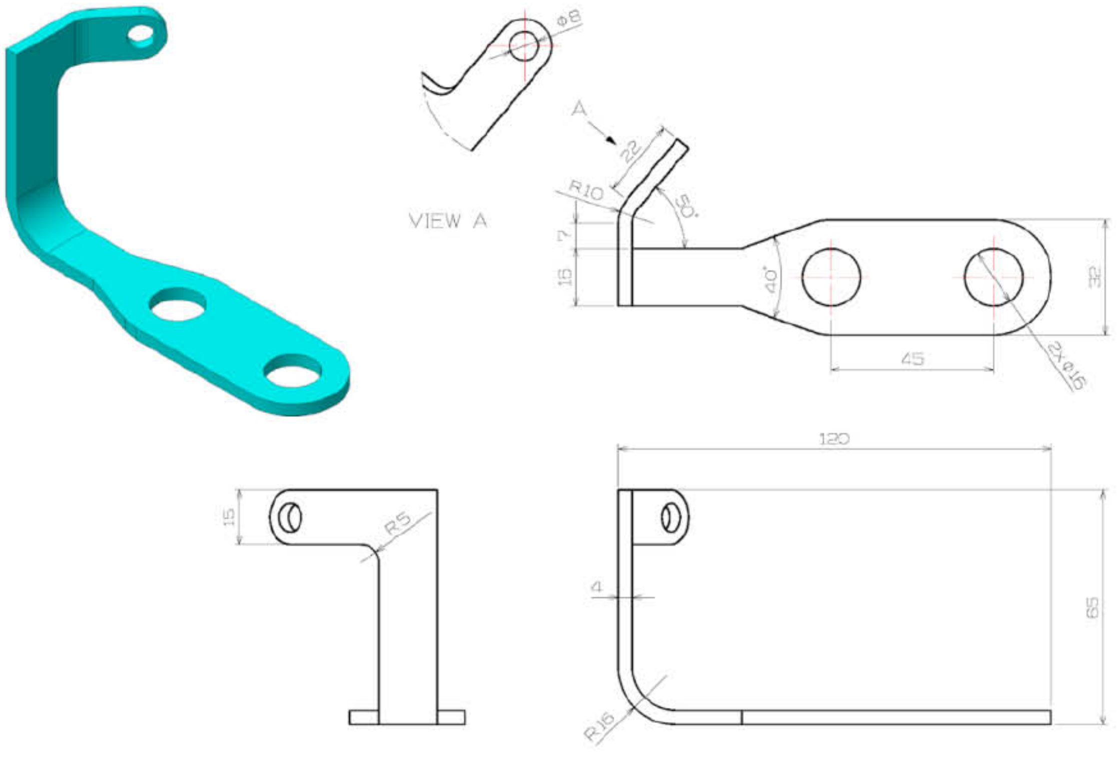
76. On the ribbon, click **View > Orientation > More > Replace View**. On the **Replace View with** dialog, click
Isometric, and then click **OK**.
77. Save and close the sheet metal part.

2.6.2 Exercises

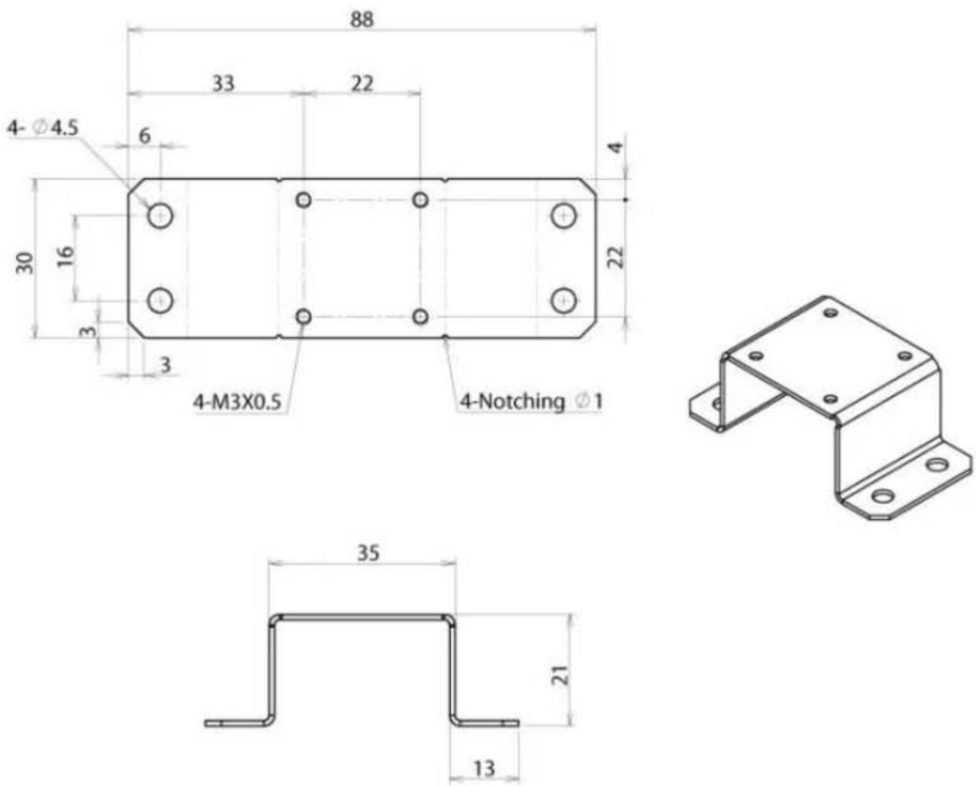
Exercise 1



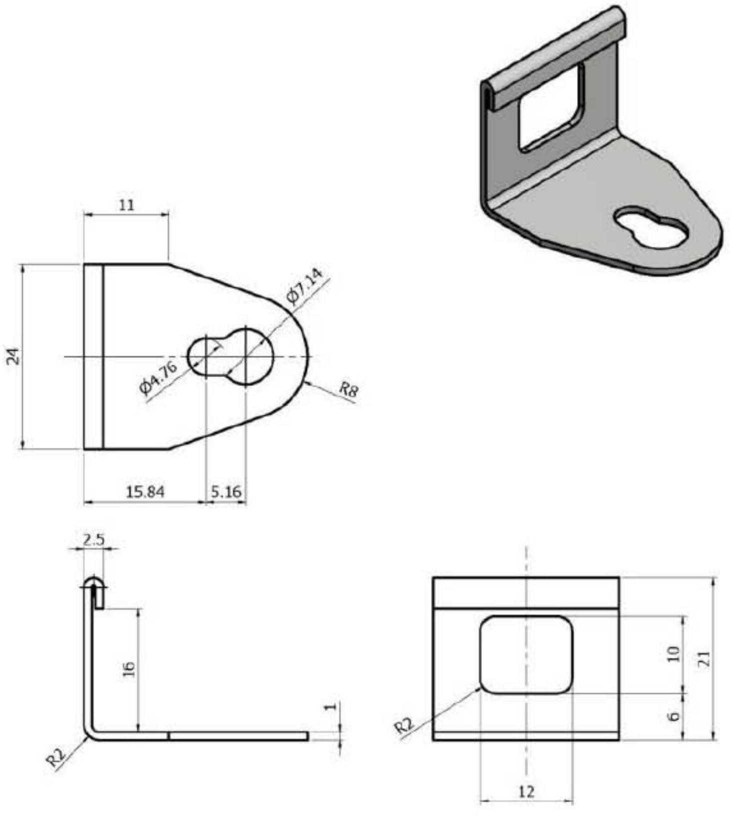
Exercise 2


VIEW A
A
Φ8
22
R10
50°
7
15
40°
32
45
2XΦ16
120
15
R5
4
65
R16

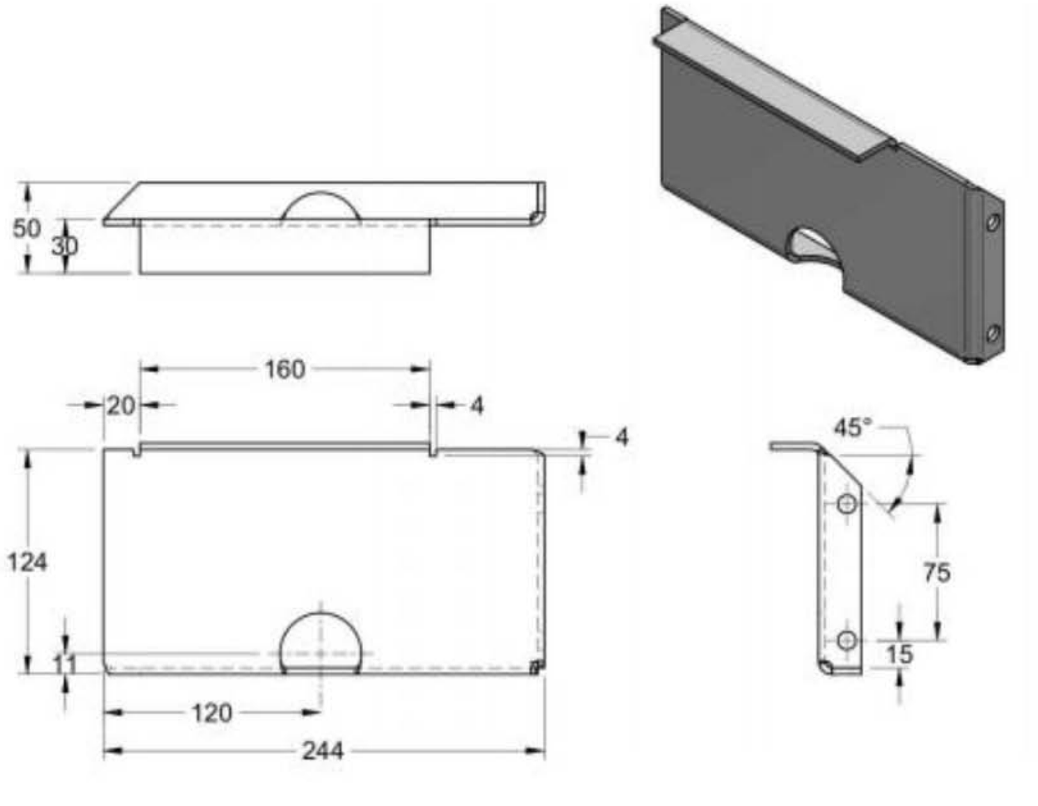
Exercise 3

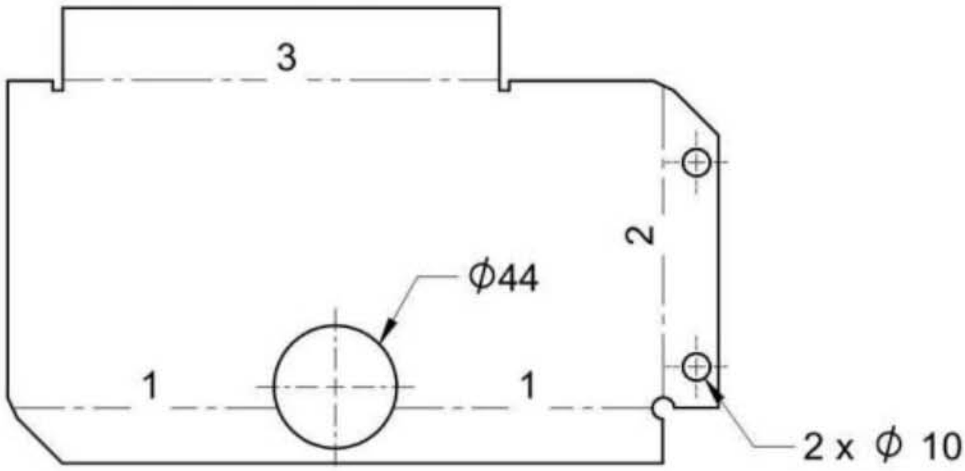

88
33
22
4- ∅4.5
6
4
30
16
22
3
3
4-M3X0.5
4-Notching ∅1
35
21
13

Exercise 4



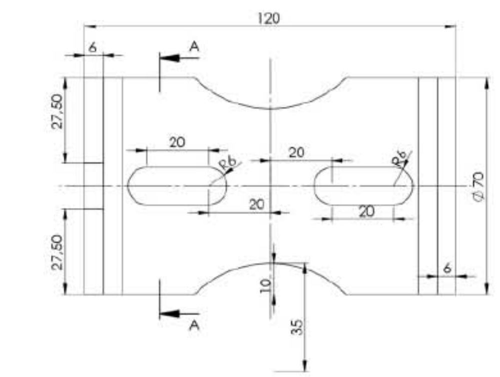
Exercise 5

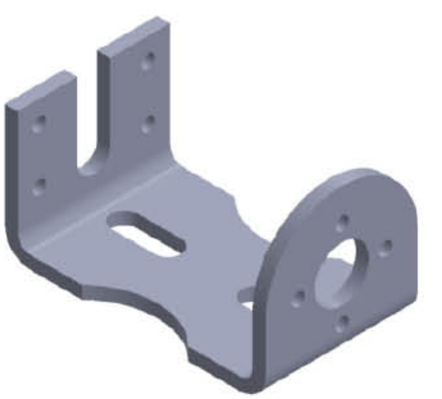


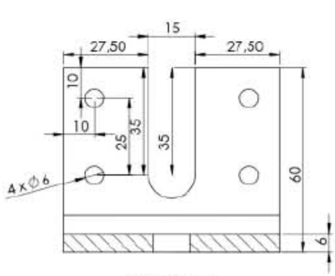


Sequence	Feature	Radius	Angle	Direction	Included Angle
1	Bend 1	3.58 mm	90.00 deg	Down	90.00 deg
2	Bend 2	3.58 mm	90.00 deg	Down	90.00 deg
3	Bend 3	3.58 mm	90.00 deg	Up	90.00 deg

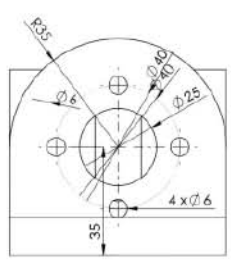
Exercise 6

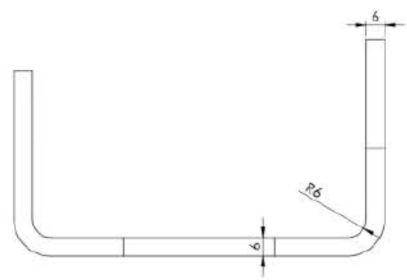




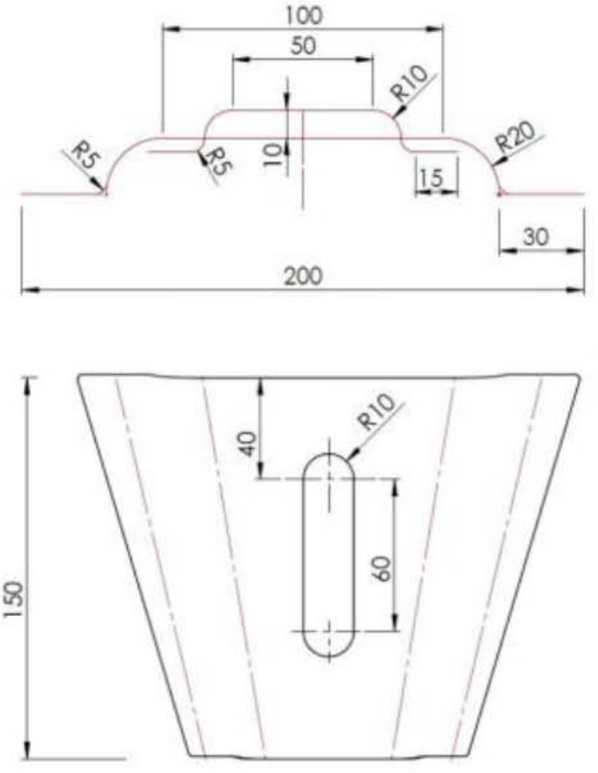


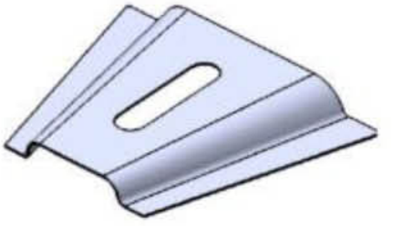
SEZIONE A-A



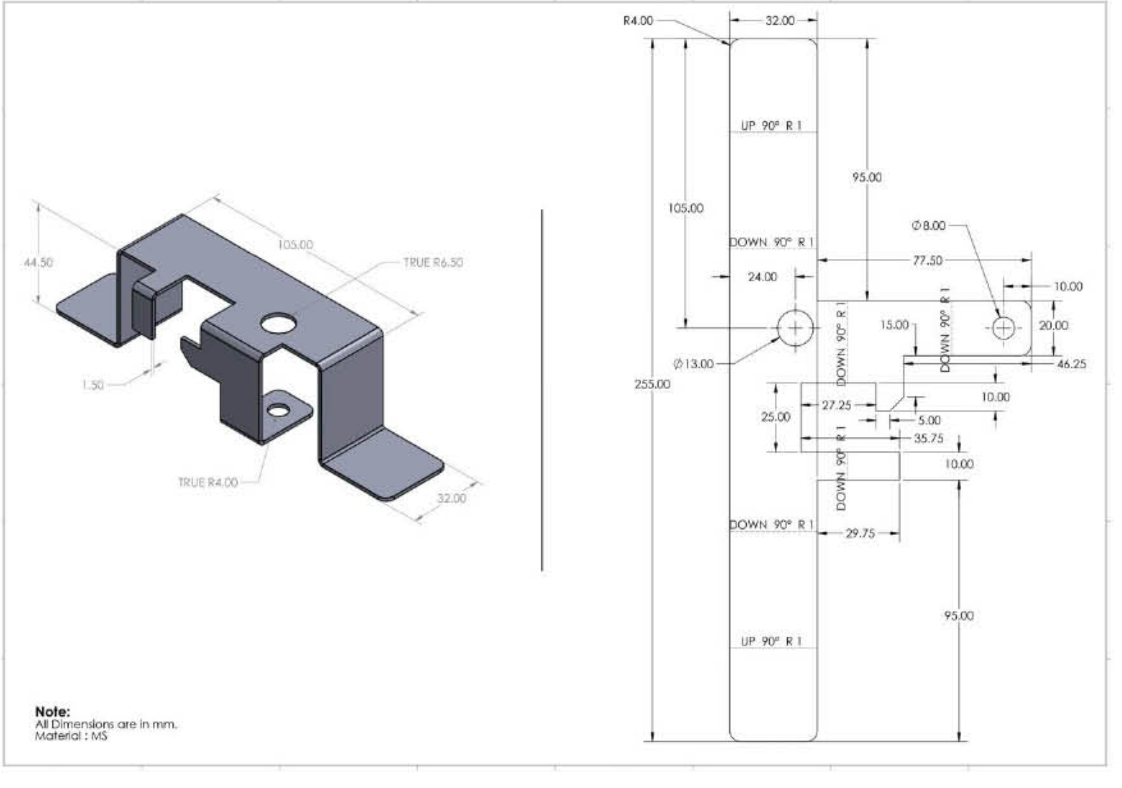


Exercise 7

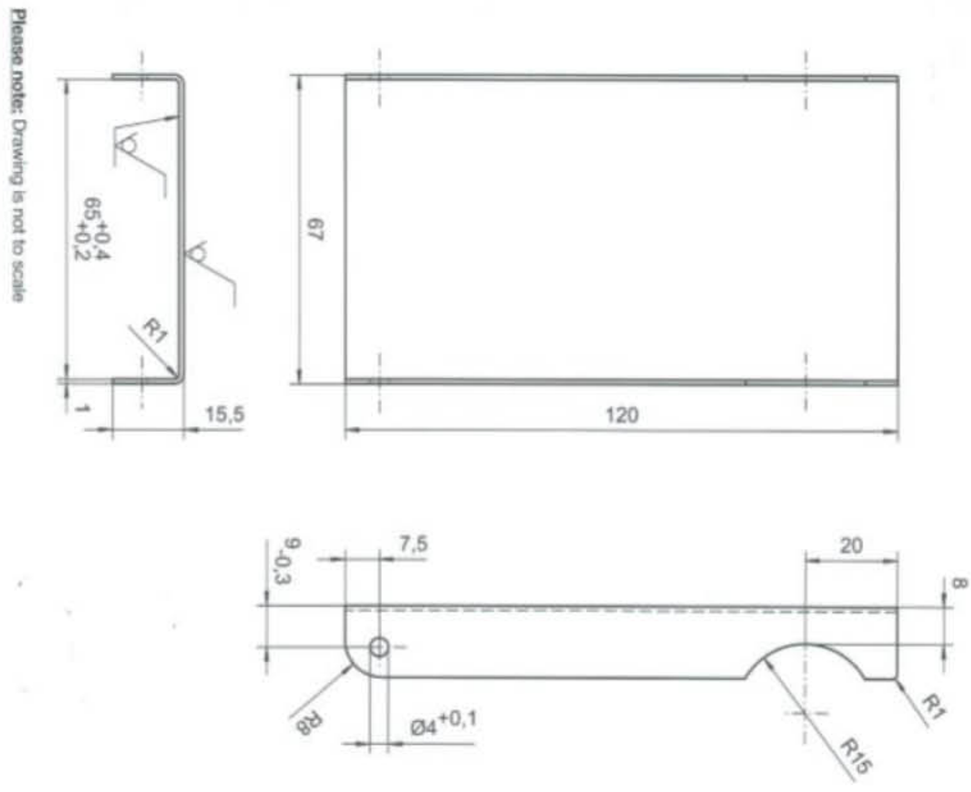


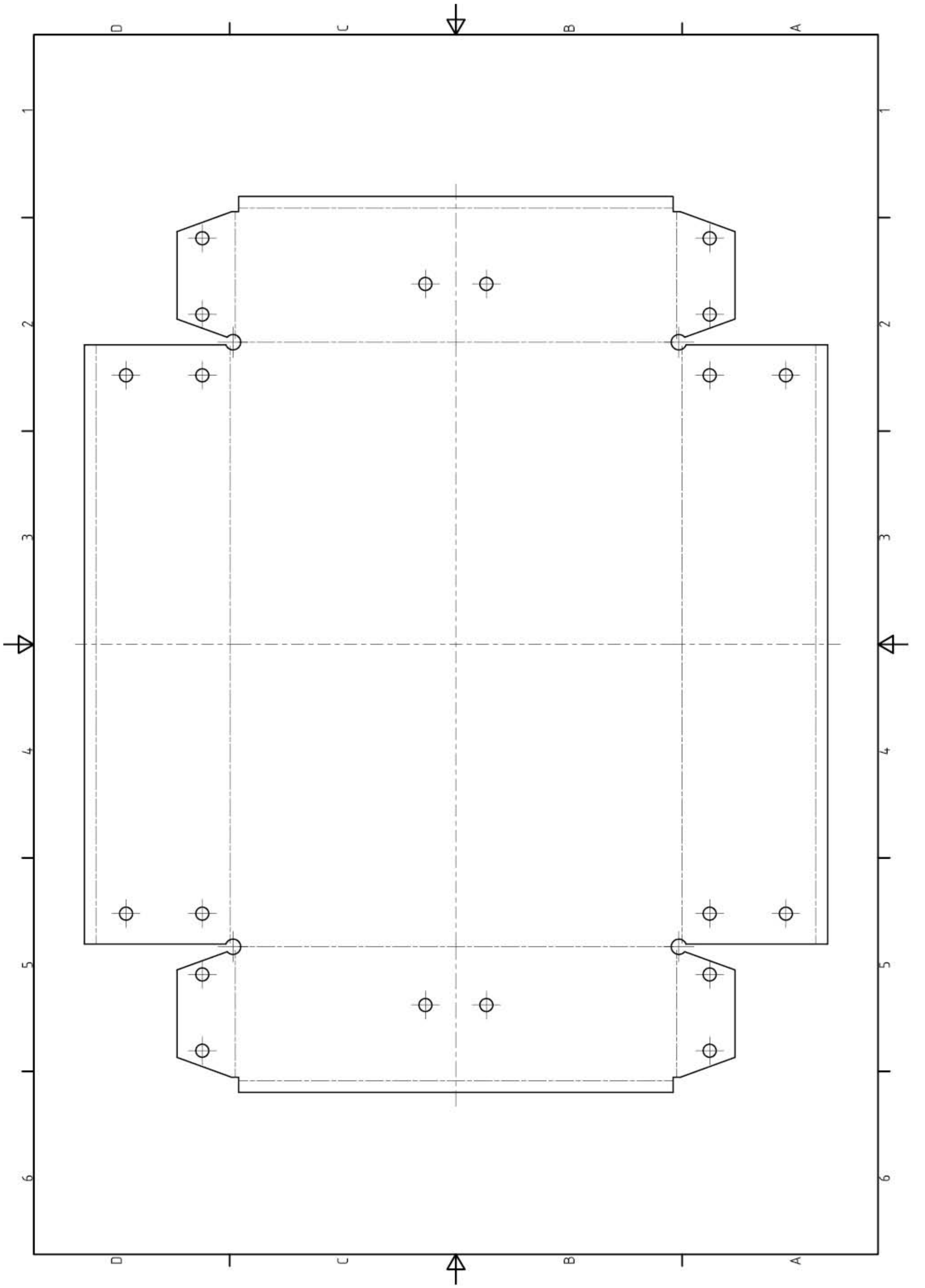


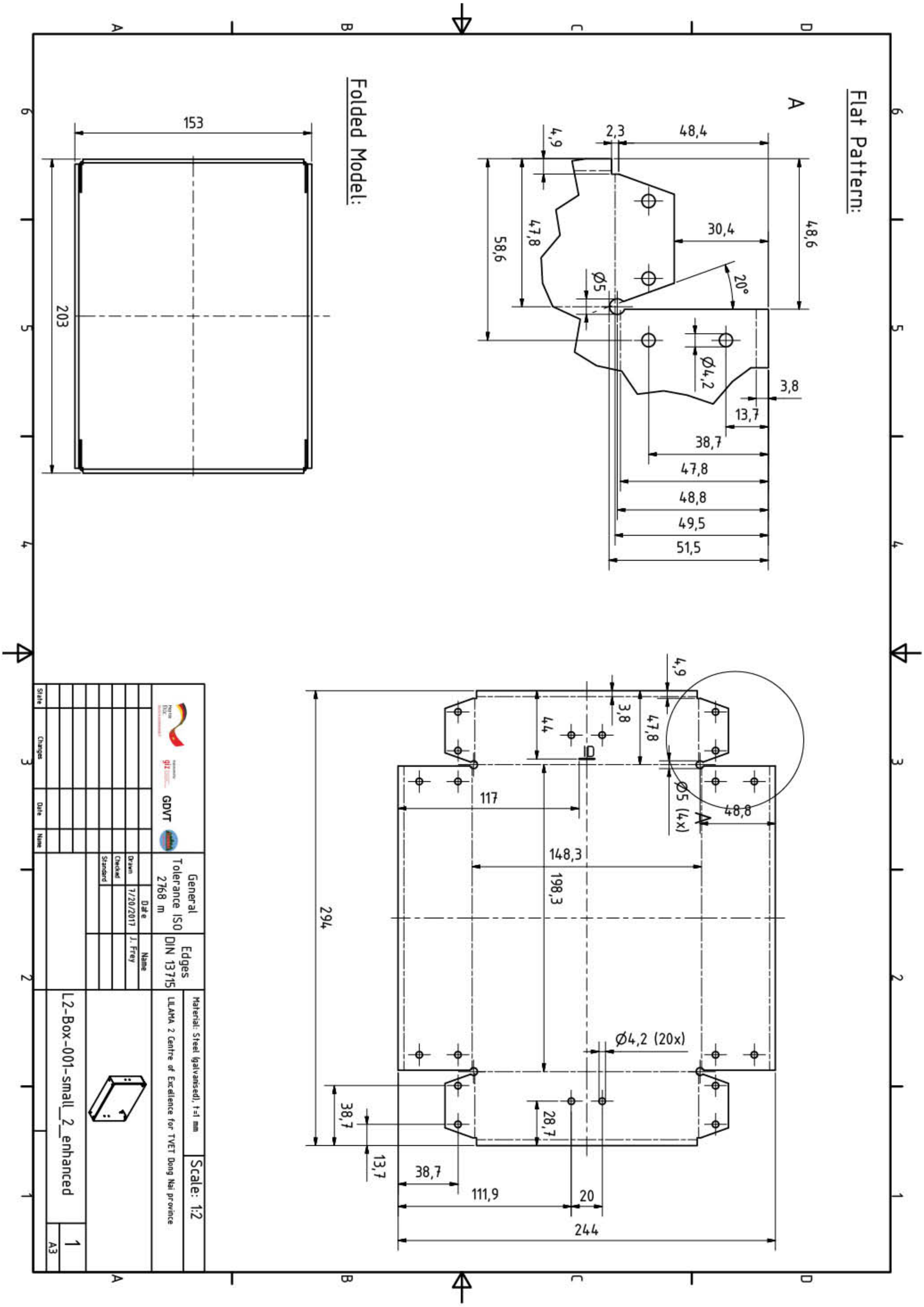
Exercise 8



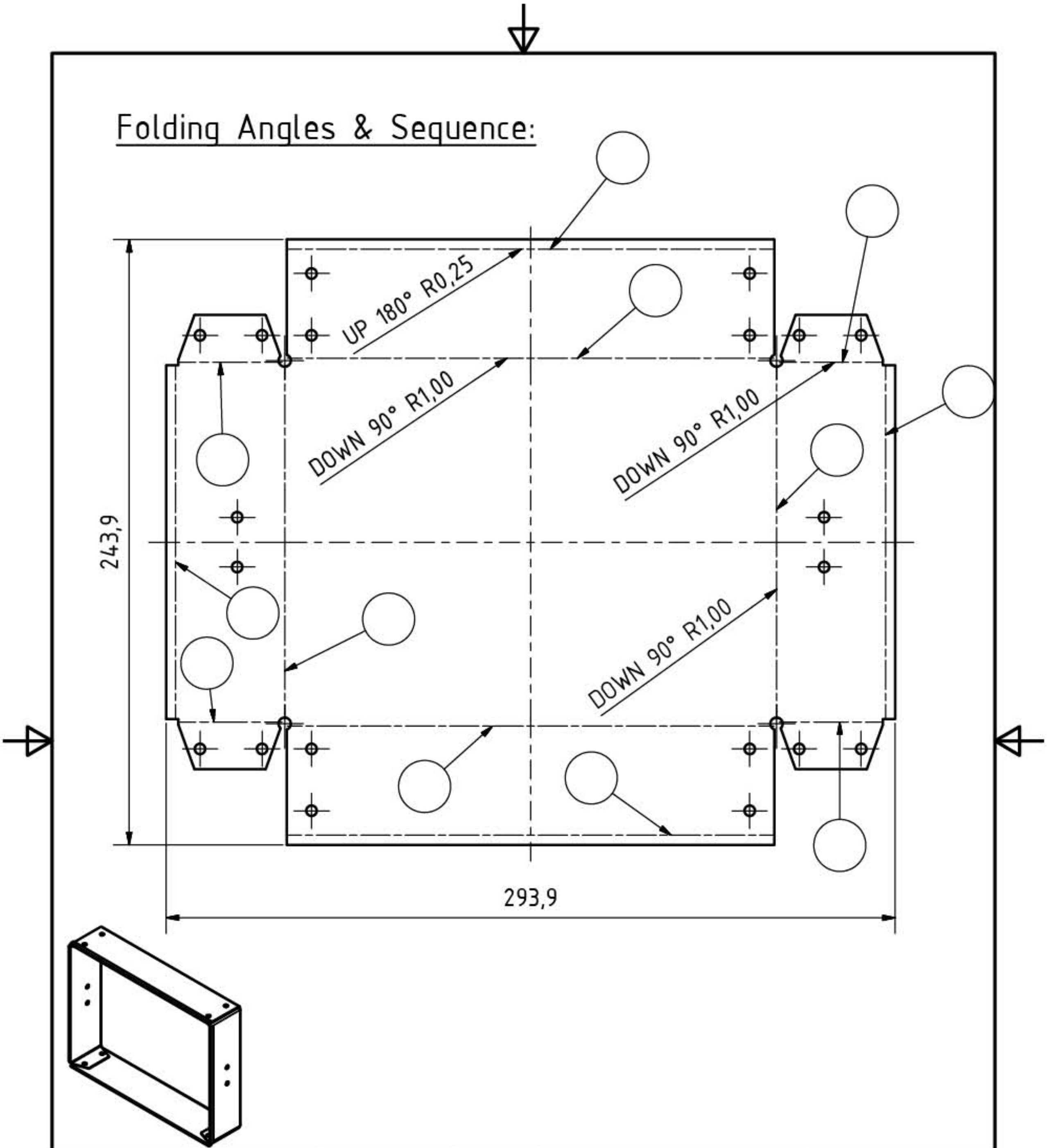
Exercise 9


Please note: Drawing is not to scale
$65^{+0,4}_{+0,2}$
R1
1
15,5
67
120
$9_{-0,3}$
7,5
20
8
R8
$Ø4^{+0,1}$
R15
R1


D
C
B
A
1
2
3
4
5
6


Flat Pattern:
Folded Model:
A
153
203
48,4
2,3
4,9
47,8
58,6
30,4
48,6
20°
Ø5
Ø4,2
3,8
13,7
38,7
47,8
48,8
49,5
51,5
294
117
148,3
198,3
44
3,8
47,8
4,9
Ø5 (4x)
48,8
Ø4,2 (20x)
38,7
28,7
13,7
38,7
111,9
20
244
General Tolerance ISO 2768 m
Edges DIN 13715
GDVT
Drawn 1/20/2017 J. Frey
Checked
Standard
Date
Name
Material: Steel (galvanised), t=1 mm
LILAMA 2 Centre of Excellence for TVET Dong Nai province
Scale: 1:2
L2-Box-001-small_2_enhanced
1
A3
State
Changes
Date
Name

Folding Angles & Sequence:



General Tolerance ISO 2768 m	Edges DIN 13715	Material: Steel (galvanised), t=1 mm	Scale: 1:2
		LILAMA 2 Centre of Excellence for TVET Dong Nai province	

GDVT

	Date	Name
Drawn	7/20/2017	J. Frey
Checked		
Standard		

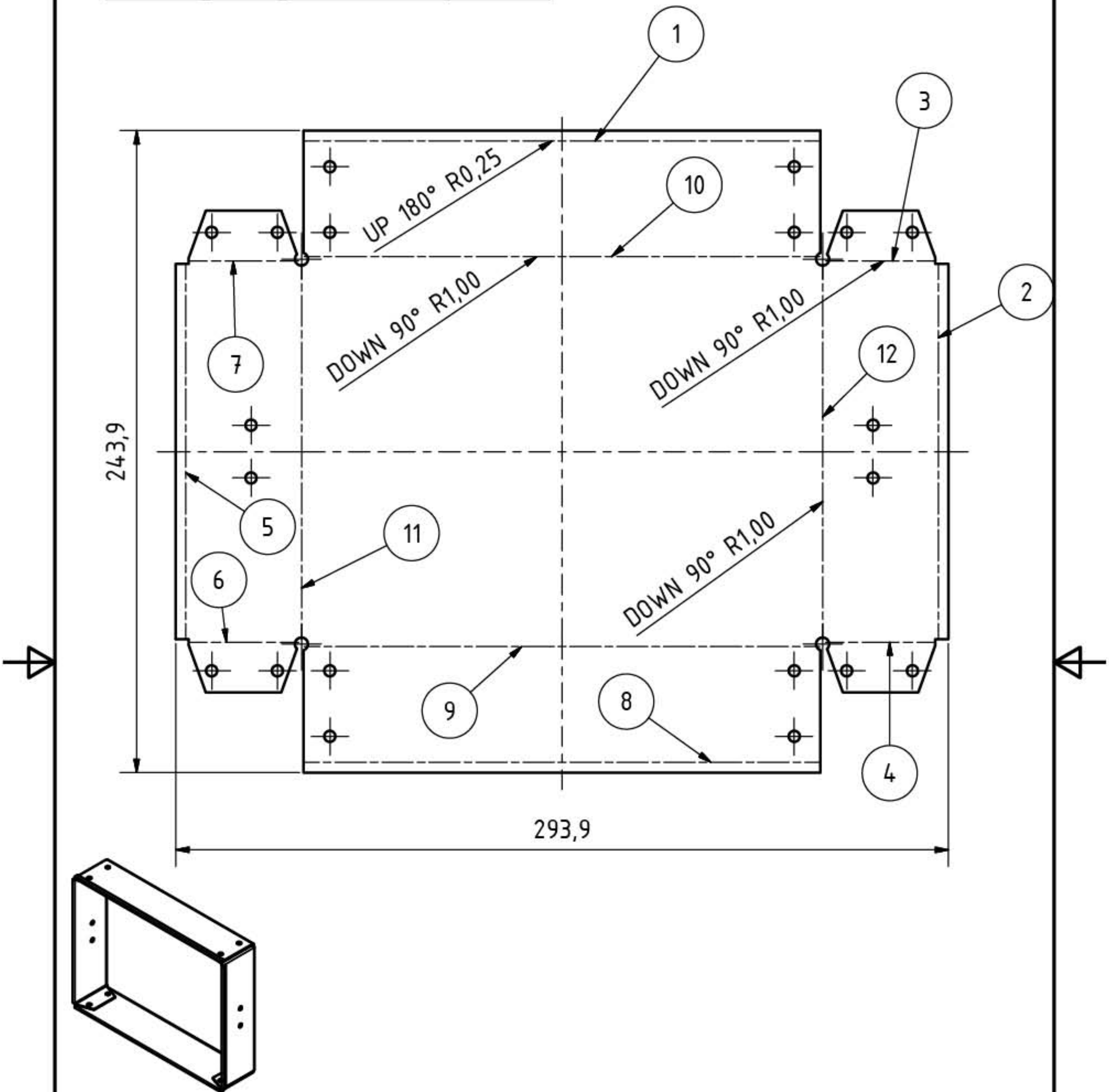
L2-Box-001-small_2_enhanced

2

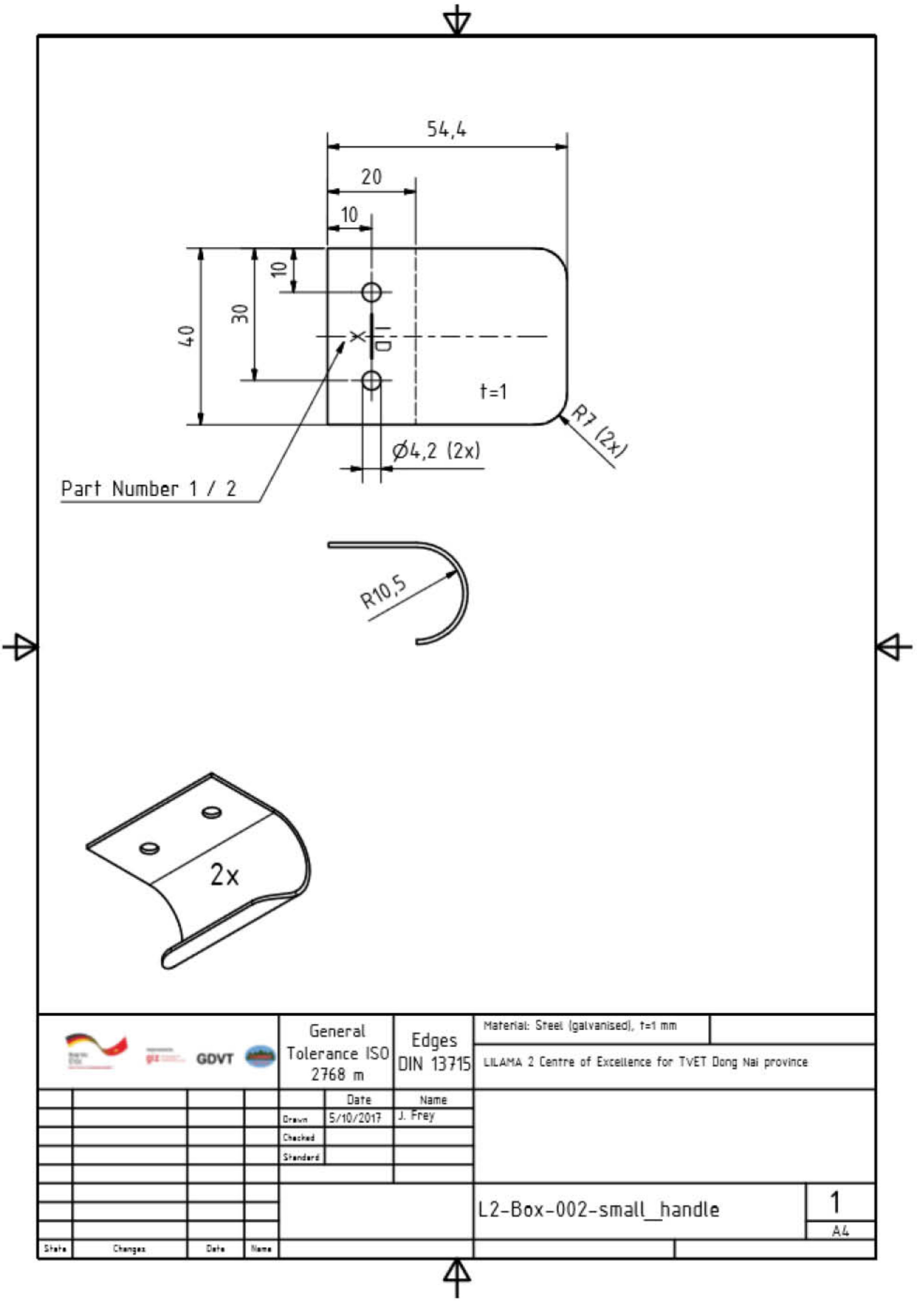
A4

State	Changes	Date	Name

Folding Angles & Sequence:



GDVT	General Tolerance ISO 2768 m	Edges DIN 13715	Material: Steel (galvanised), t=1 mm — Scale: 1:2
			LILAMA 2 Centre of Excellence for TVET Dong Nai province
		Date	Name
	Drawn	7/20/2017	J. Frey
	Checked		
	Standard		
State / Changes / Date / Name		L2-Box-001-small_2_enhanced	3 A4


54,4
20
10
10
30
40
t=1
R7 (2x)
Ø4,2 (2x)
Part Number 1 / 2
R10,5
2x
GDVT
General Tolerance ISO 2768 m
Edges DIN 13715
Material: Steel (galvanised), t=1 mm
LILAMA 2 Centre of Excellence for TVET Dong Nai province
Date
Name
Drawn
5/10/2017
J. Frey
Checked
Standard
L2-Box-002-small_handle
1
A4
State
Changes
Date
Name

UNIT 3: DESIGN OF PLASTIC INJECTION MOLDS WITH NX: FROM CONCEPT TO DETAILING

Durations:80 hours

Objectives: By the end of this training unit, the trainees will be able to:

Understand the overview of mold design and manufacturing technology

Understand the structure and classification of plastic injection molds

Understand the process of designing and manufacturing plastic injection molds

Understand the design standards of plastic injection molds

Calculating the technical parameters of plastic injection molds

Perform automatic mold design steps with NX software

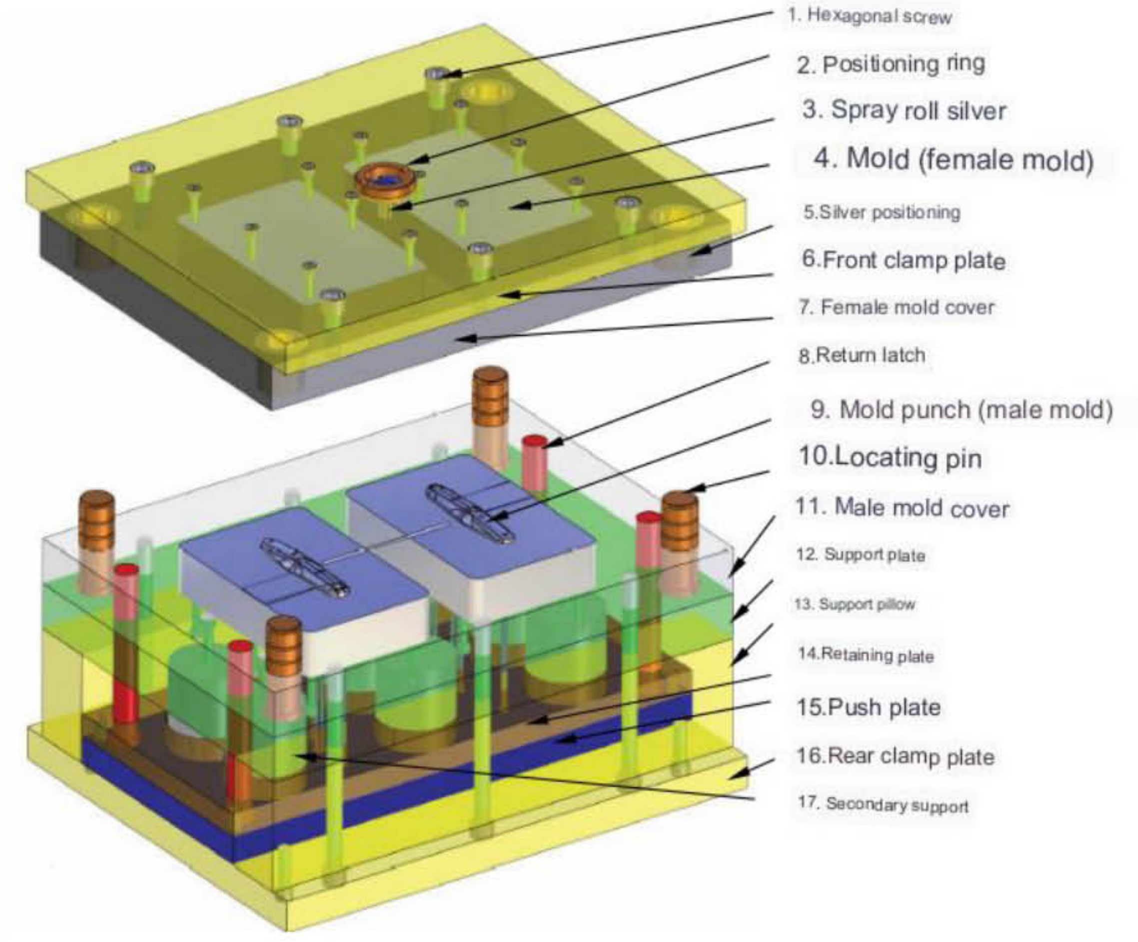
Implement the simulation process to analyze and optimize the mold design process with NX software

Implement the management of mold data after design (BOM)

Export drawings to manufacture mold details.

Contents:

Mold Overview.



Hexagon screw: Connects mold plates and creates aesthetics.

Positioning ring: centering between the nozzle bush and the nozzle.

Spray stem bush: guides plastic from the plastic injection machine into the plastic channels.

Mold: creates shape for the product.

Positioning silver: ensures relative position between male and female molds.

Front clamp plate: firmly holds the fixed part of the mold to the plastic injection machine.

Female mold shell: usually made of cheaper materials than the female mold, so it helps reduce mold costs but still ensures technicality.

Return pin: returns the push system to its original position when the mold closes.

Male mold: shapes the product.

Positioning pin: inserted into the positioning bush when the mold is closed, helping the male and female molds connect accurately.

Male mold shell: usually made of cheaper materials than female molds, so it helps reduce mold costs but still ensures technicality.

Support plate: increases mold durability during injection molding process.

Support bearing: creates space for the push plate to operate.

Retaining plate: holds the push pins.

Push plate: push the push pins to release the product from the mold.

Back clamp plate: firmly holds the movable part of the mold on the plastic injection machine.

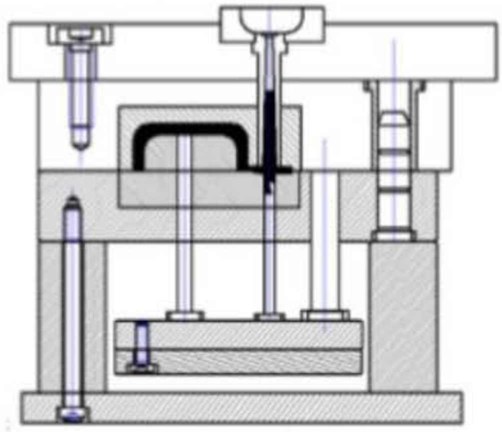
Secondary support: increases the durability of the mold during the injection molding process.

1. Two-plate mold.

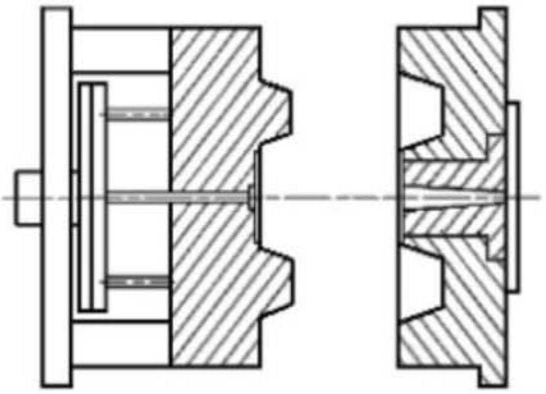
The 2-plate mold is an injection mold using a cold guide system, the channel is horizontal to the mold parting surface, the plastic entrance gate on the side of the product v2 when opening the mold, there is only a gap to get the product and the plastic channel.

For a 2-plate mold, the plastic entrance gate can be designed so that the product and the plastic channel automatically separate when the product and the plastic channel (glue bone) are removed from the mold.

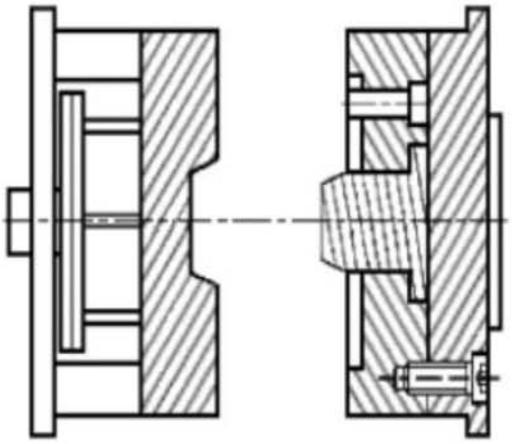
The 2-plate mold method is very common in injection molding systems. The mold consists of 2 parts: negative mold and positive mold. The mold structure is simple and easy to manufacture, but 2-plate molds are usually only used for products where the plastic entrance is easy to arrange.



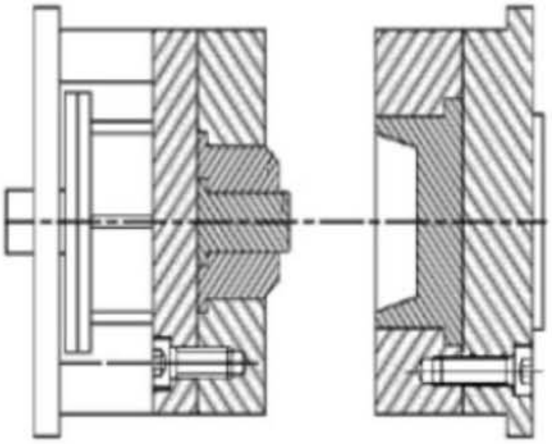
The two-plate mold has one mold cavity



The two-plate mold has many cavities



Two-Plate Mold with assembled Core



The two plate mold has a side-jointed core in the composite core

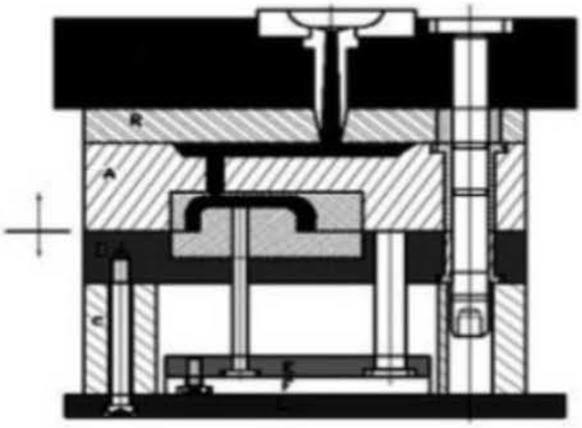
2.Three-plate mold.

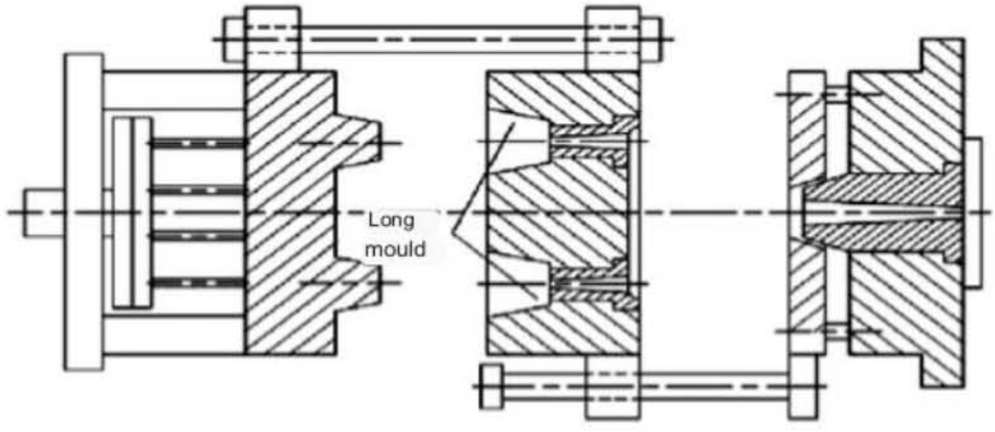
The 3-plate mold is an injection mold that uses a cold plastic pipeline system. The channel is arranged on two flat surfaces and when opening the mold, there are two openings, one opening to take out the product and the other opening to take out the glue bone. Therefore, if the product is taken into the channel out of the mold using the ejector system, two systems must be arranged, so the mold structure will be more complex and larger than a 2-plate mold.

For 3-plate molds, the product and plastic channel are always automatically separated when the product and plastic channel are removed from the mold.

For large products that need many nozzles or multi-cavity molds that need many nozzles, a 3-plate mold can be used.

The disadvantage of the 3-plate mold system is that the distance between the machine's nozzle and the mold cavity is long, which can reduce injection pressure when plasticizing the mold cavity. This can be overcome by using a hot resin pipeline system





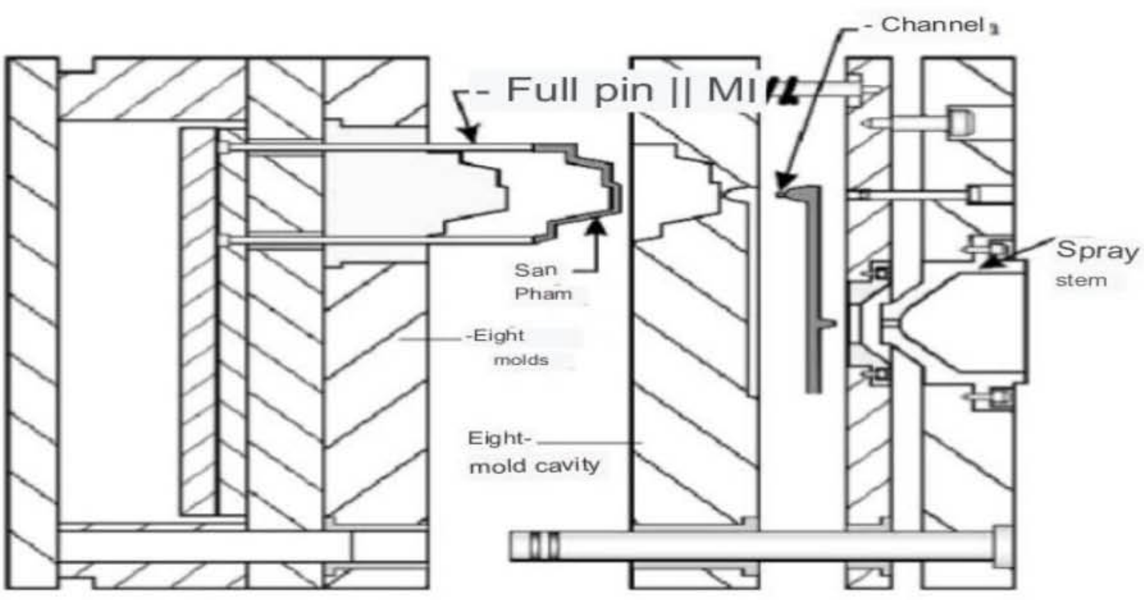
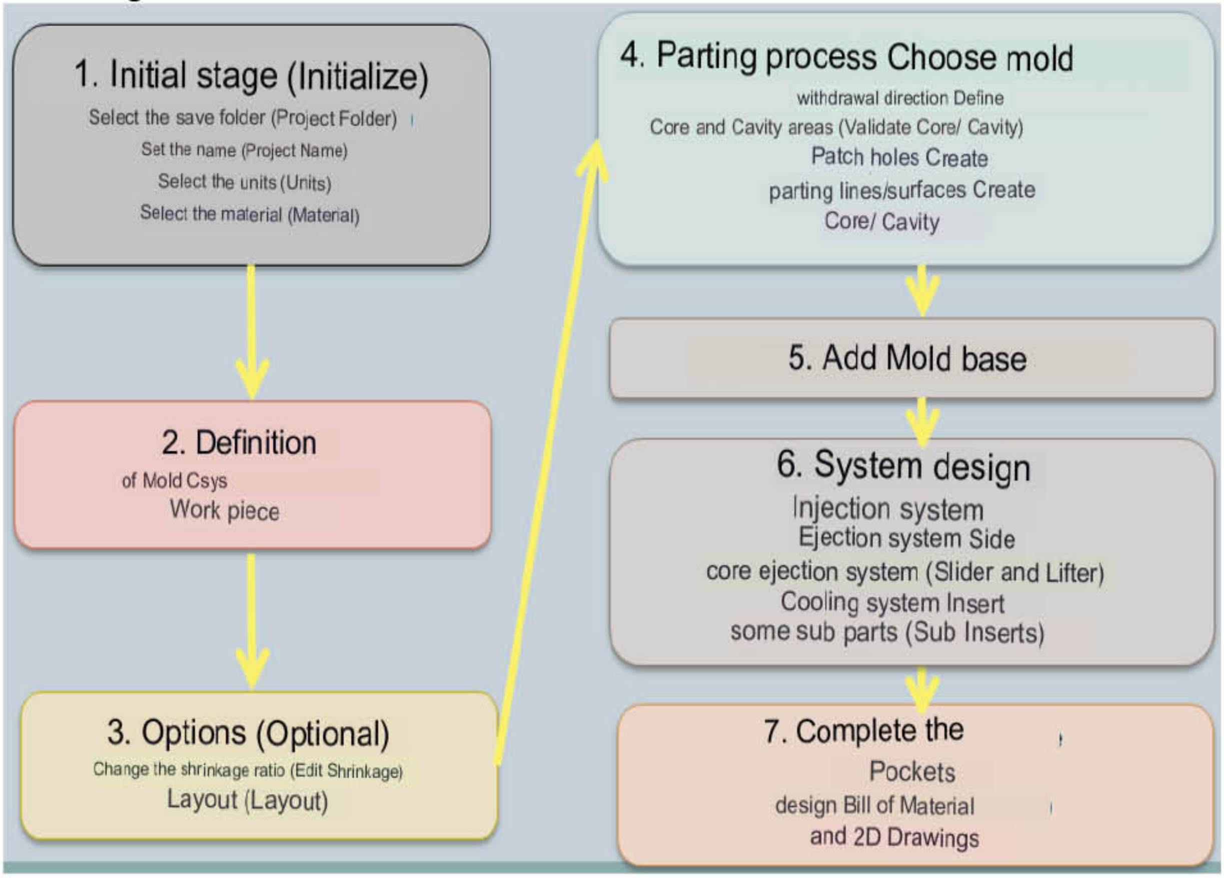


Figure 1.9.2.1. 3-plate mold structure

Mold Design with NX Software



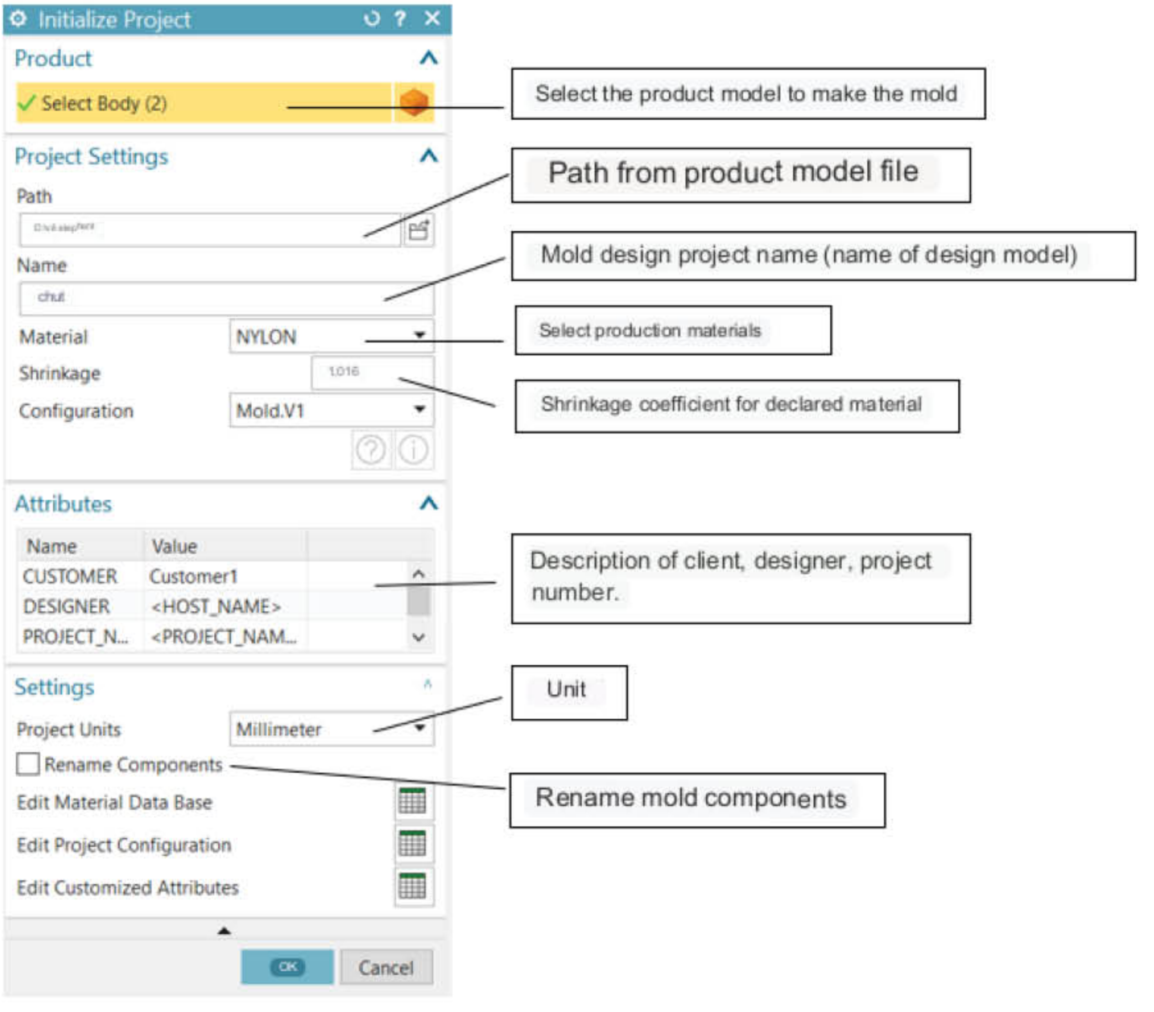
3.1. Initial stage. (Innitiallize)

Open the model file that needs to be molded.

Select mold design environment: Select File All Applications → Process Specific → Mold wizard.

The Mold Wizzard toolbar displays.

Select the Initialize Project initialization command. The Initialize Project dialog box appears

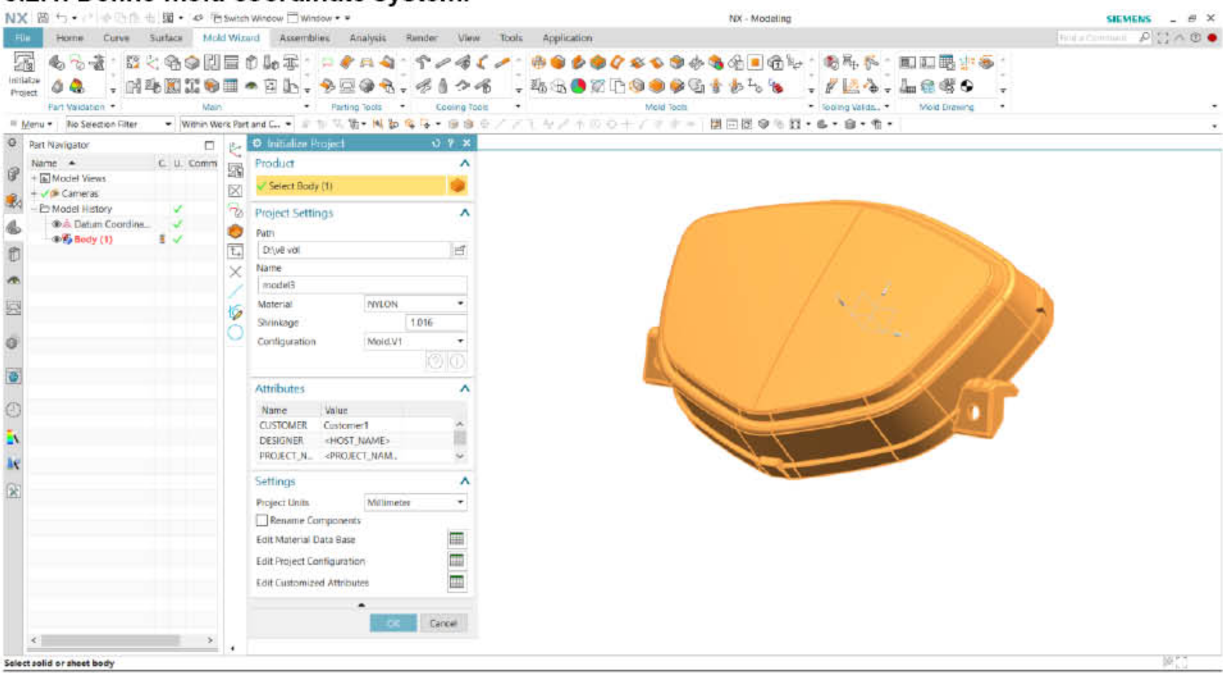


Mold Wizard includes a material library system (nylon, ABS, PPO, PS...) available, allowing users to choose directly; or create a new material library for your own use in the Edit Material Data Base section.

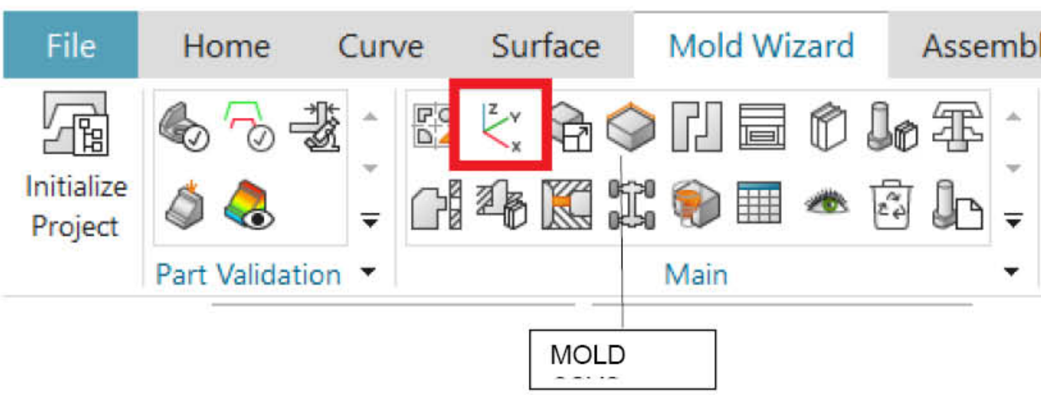
After the injection molding process, the product shrinks with a certain coefficient corresponding to each material being injected. The software will automatically update the corresponding shrinkage coefficient (Shrinkage), this ensures that the mold created has the right shape. suitable, the designed product is not deformed or missing in size. In addition, the shrinkage coefficient can be changed by the user by entering the coefficient in the Edit Material Data Base section.

3.2. Define

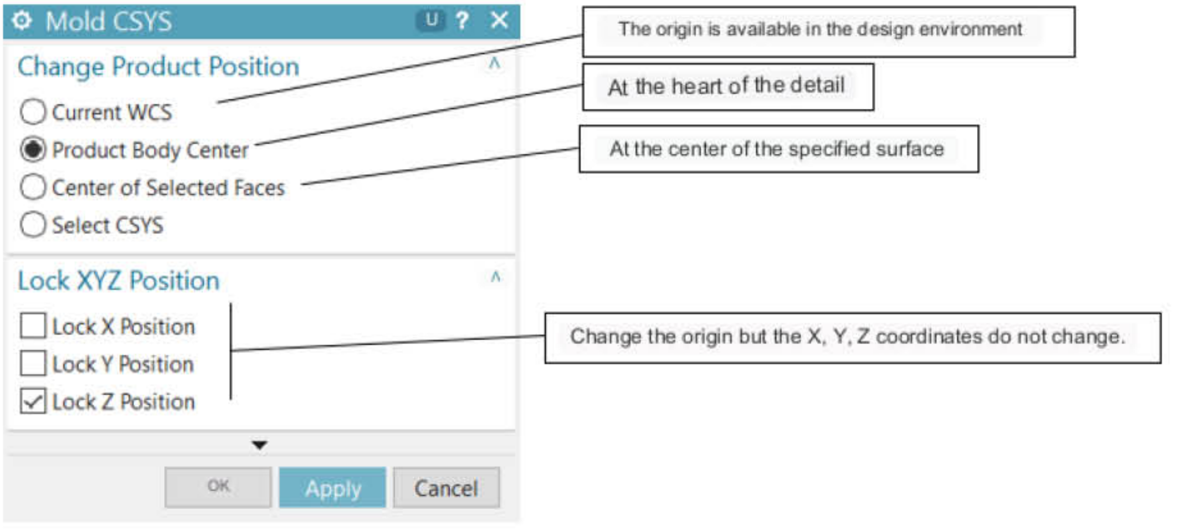
3.2.1. Define mold coordinate system.



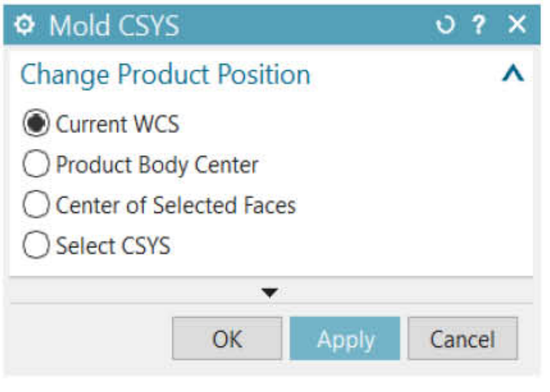
On the Mold Wizard toolbar, select Mold CSYS in the Main tab.

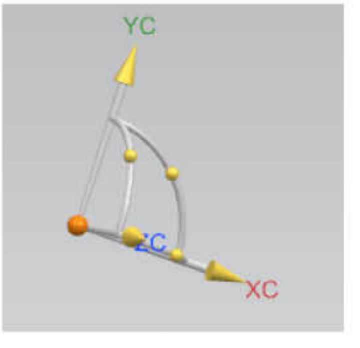


The mold coordinate system determines the workpiece position, mold withdrawal direction and mold parting surface. Normally, choose the mold withdrawal direction to coincide with the Z axis.



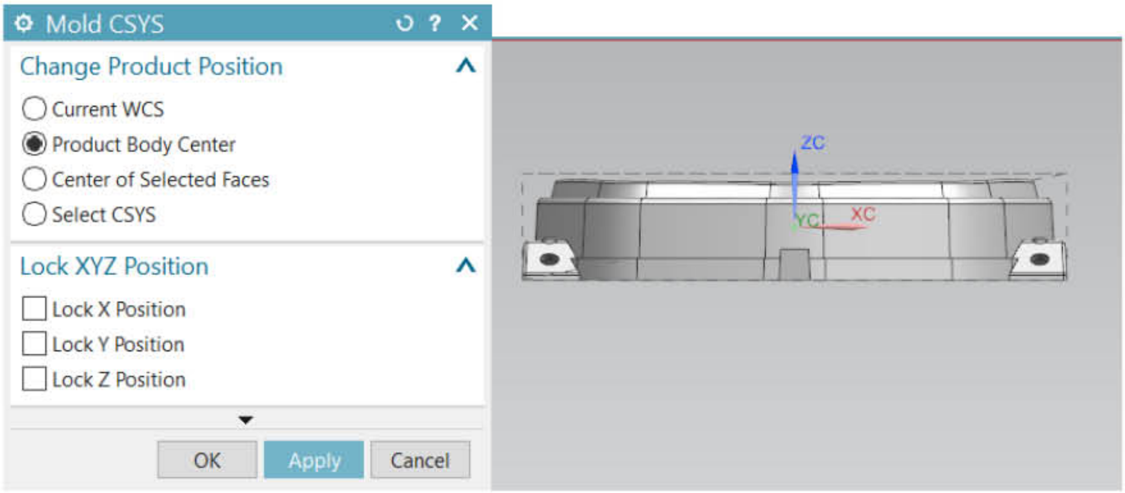
Current WCS: When selecting Current WCS, users can still change the coordinate system from the original design coordinates by double-clicking the left mouse button on the coordinate system, control buttons as below will appear.



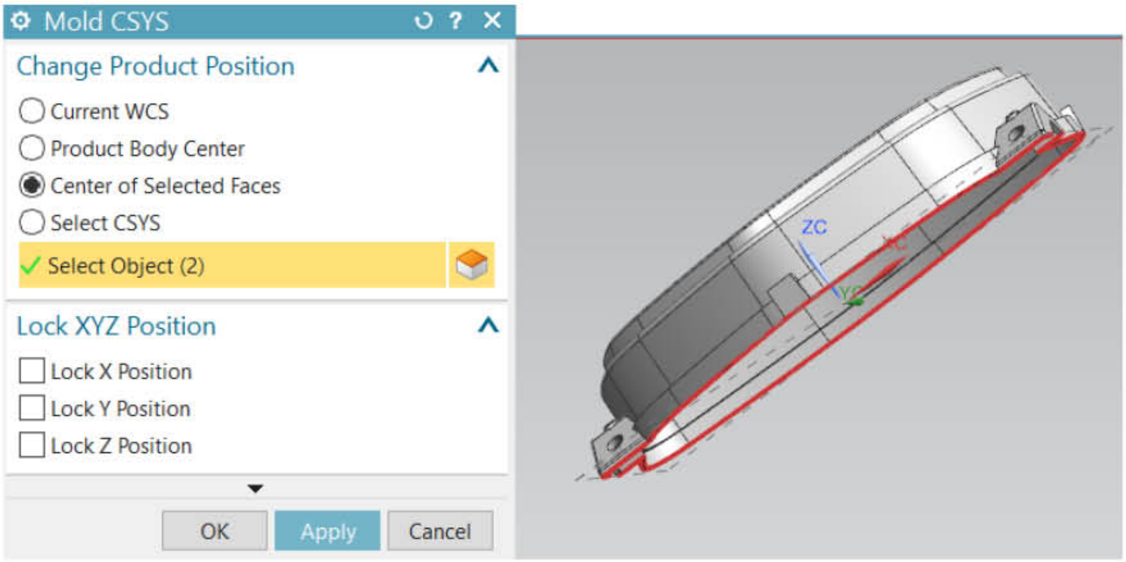


The direction of the arrow shows how to move the mold TK coordinate system according to the arrow. The yellow sphere at the control planes rotates the plane around the remaining axis, the orange sphere at the origin is used to control the coordinate system. move to any position, the mold coordinate origin is usually located at the mold parting surface.

Product Body Center: When selecting Product Body Center, the coordinate system will automatically jump to the center of the entire object in all three basic directions (X, Y, Z), however the designer can lock the system movement coordinates according to those basic directions by clicking on the square icon in the Lock XYZ Position group.

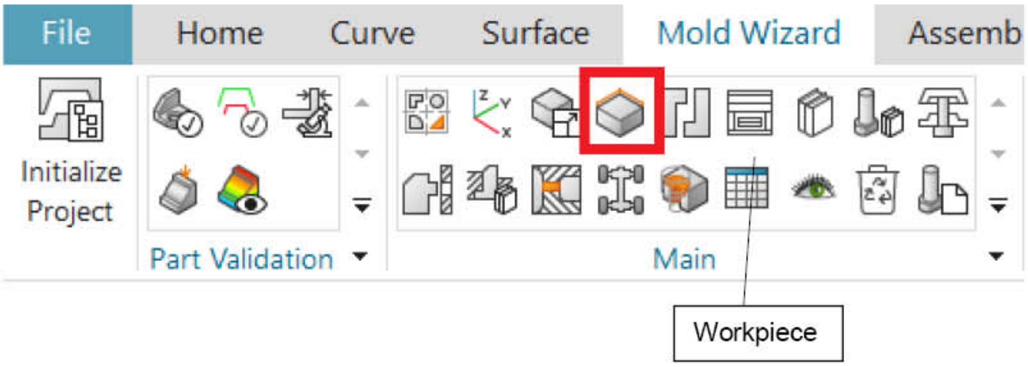


Center of selected faces: Command allows placing the coordinate system at the center of a selected face. This selection face is specified with the Select Object command. Similar to the Product Body Center command, the Center of Selected Faces command allows control to lock one or more basic axes in the Lock XYZ Position group.

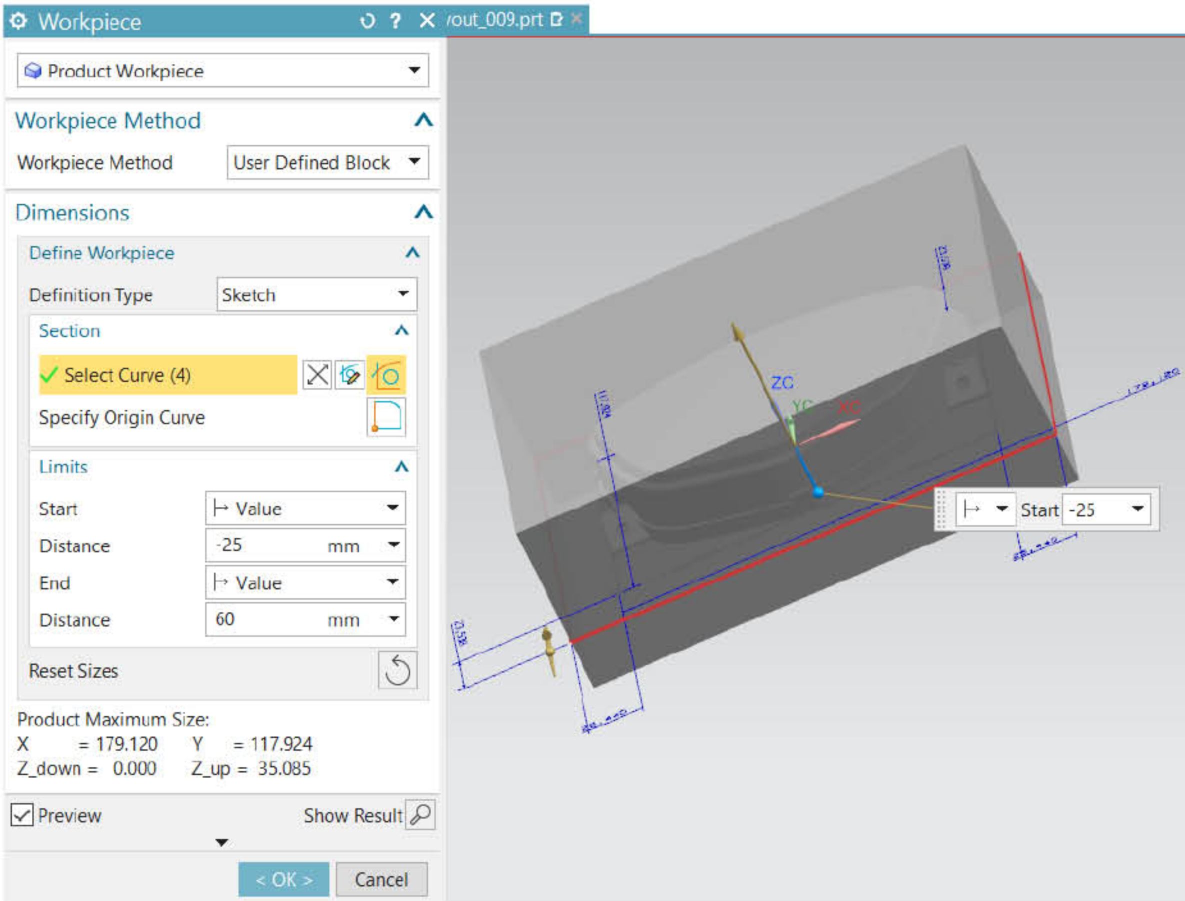


3.2.2. Definition of embryo.

On the Mold Wizard toolbar, select Workpiece in the Main tab.

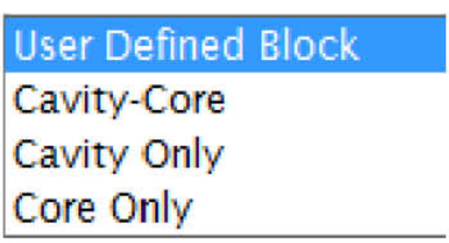


NX automatically updates the workpiece size or the user can customize the size or create a new workpiece cross-section

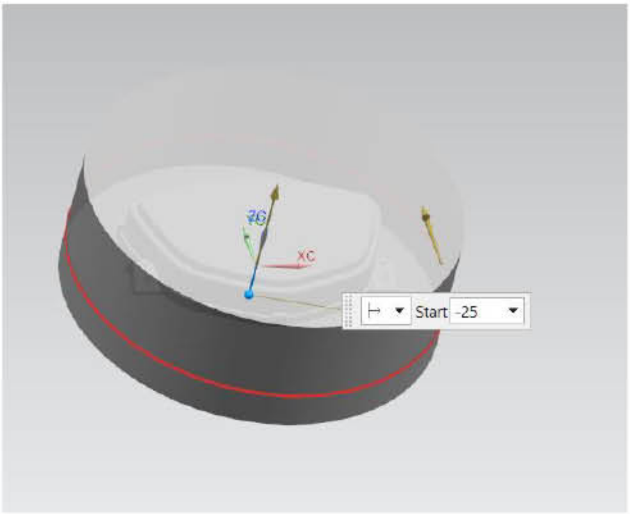


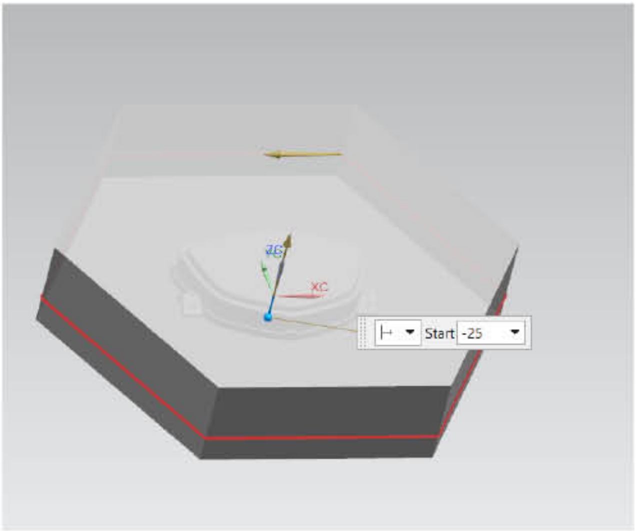
Select the type of Workpiece, there are 2 types: Product Workpiece and Combined Workpiece.

Choose embryo creation method :



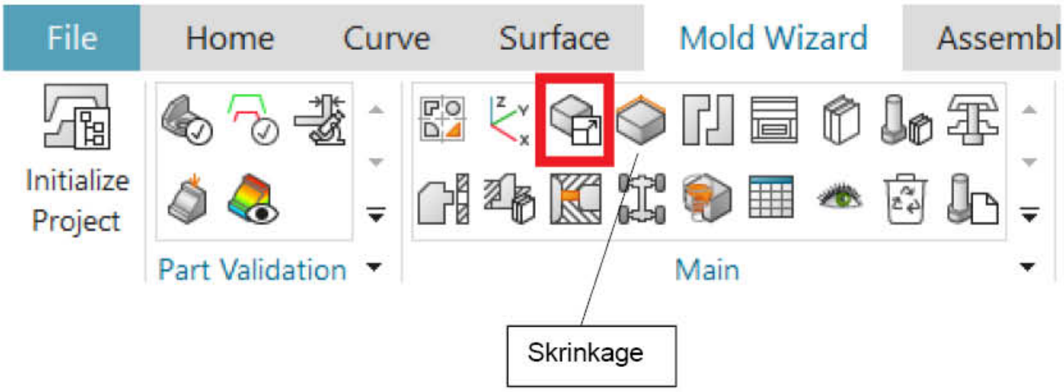
User-defined: use User Defined Block. In this option, users can define the workpiece type according to their ideas by defining the workpiece cross-section. An existing line can be selected using the Curve command  or create a new section by selecting Sketch 



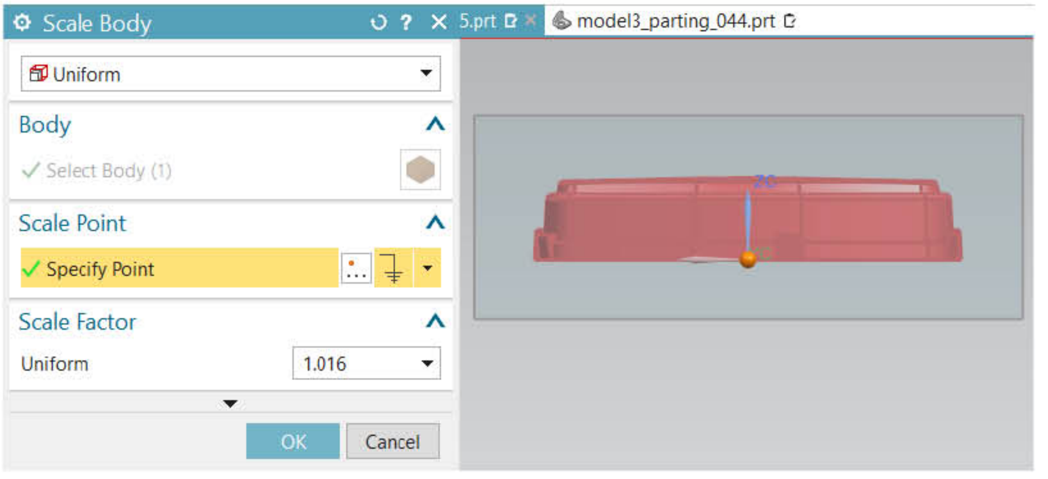


3.3. Option. (Optional)

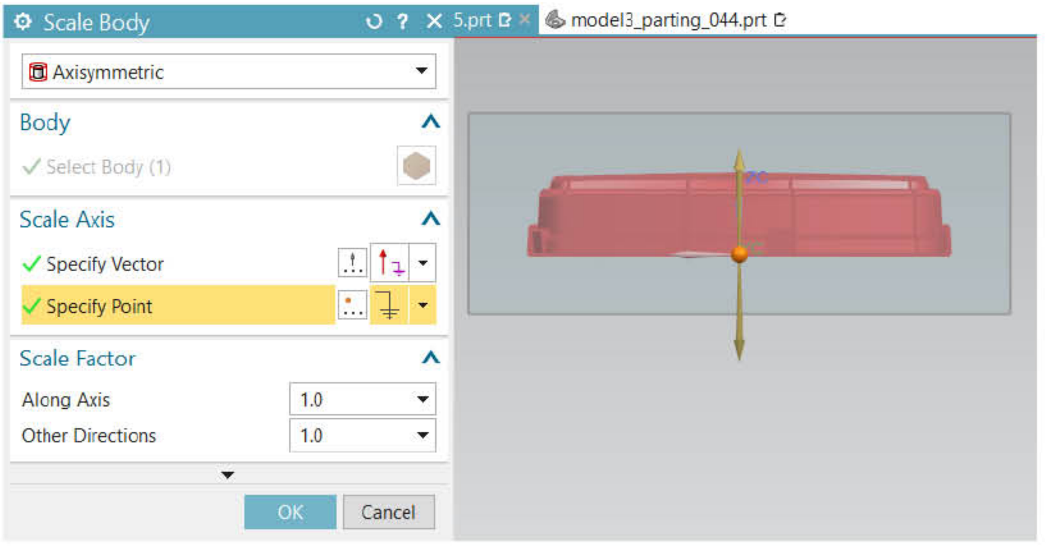
3.3.1. Change the shrinkage coefficient.
On the Mold Wizard toolbar, select Skrinkage in the Main tab.



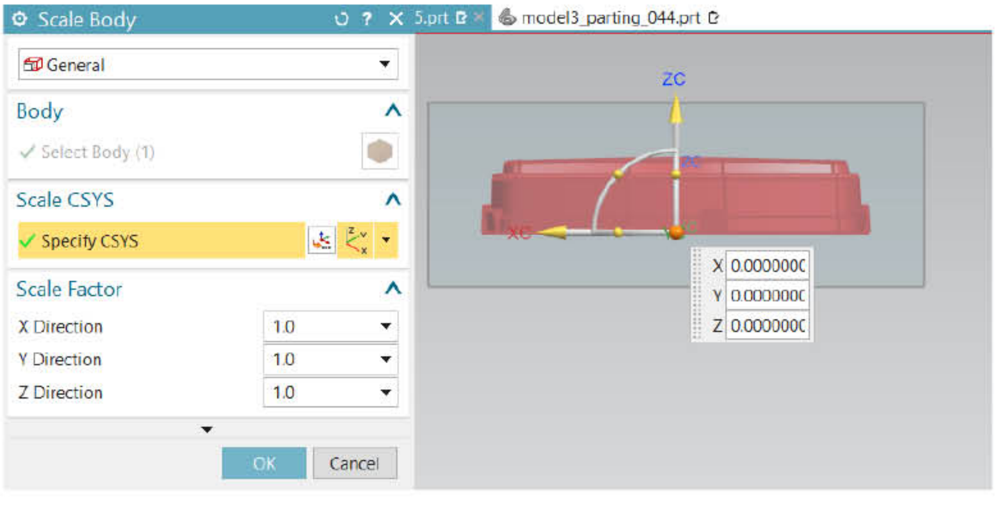
The shrinkage factor is automatically updated by the software during initialization, however, the software provides 3 additional methods to change the shrinkage factor.



Uniform: Shrinkage coefficient around a point, the product will expand around this point.

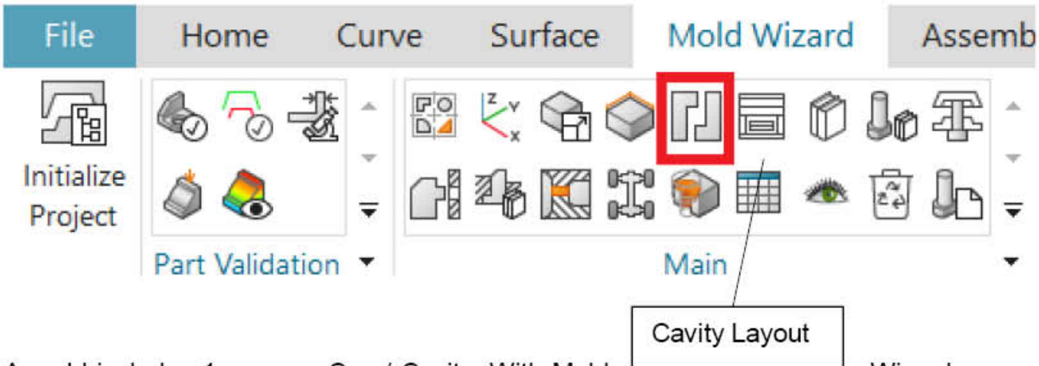


Axisymmetric: Cylindrical shrinkage coefficient, the product will expand around this cylindrical axis

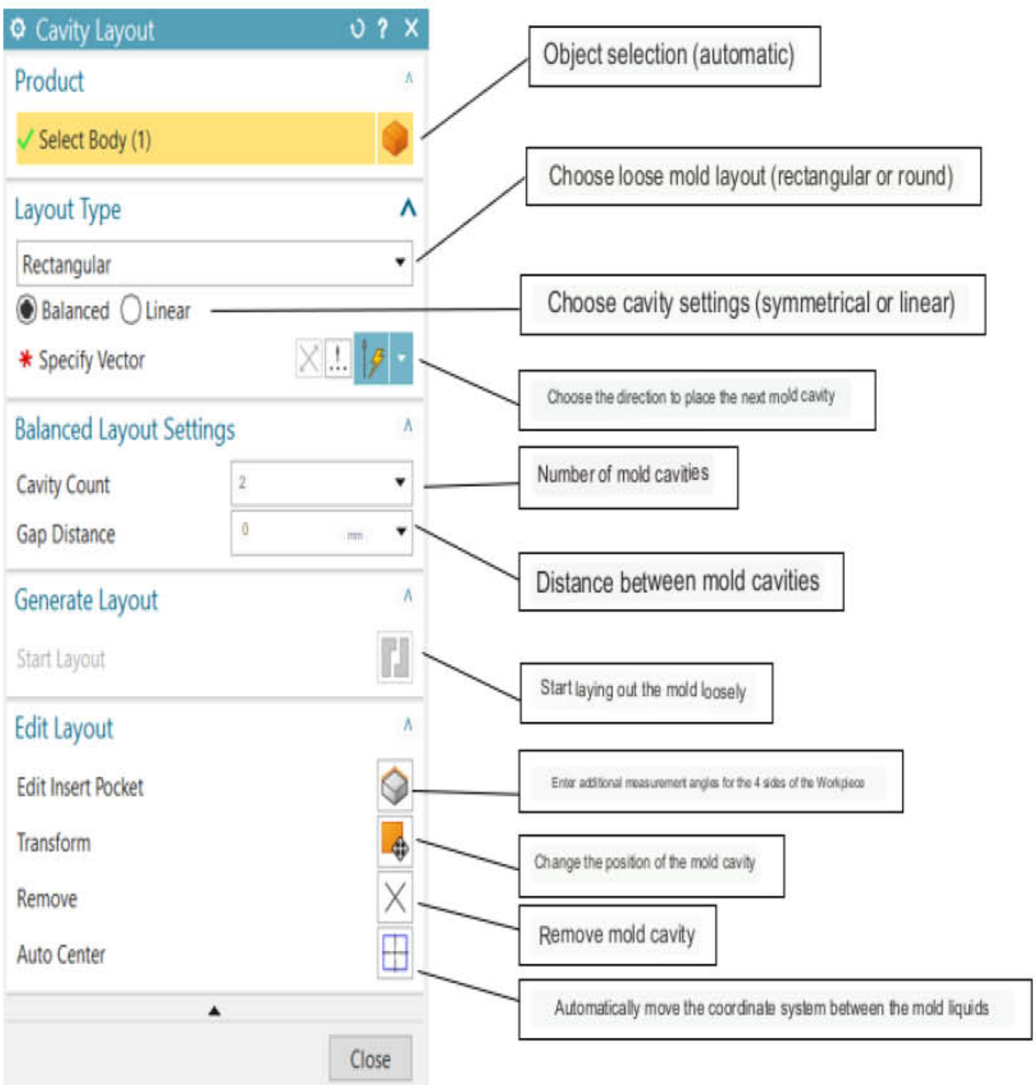


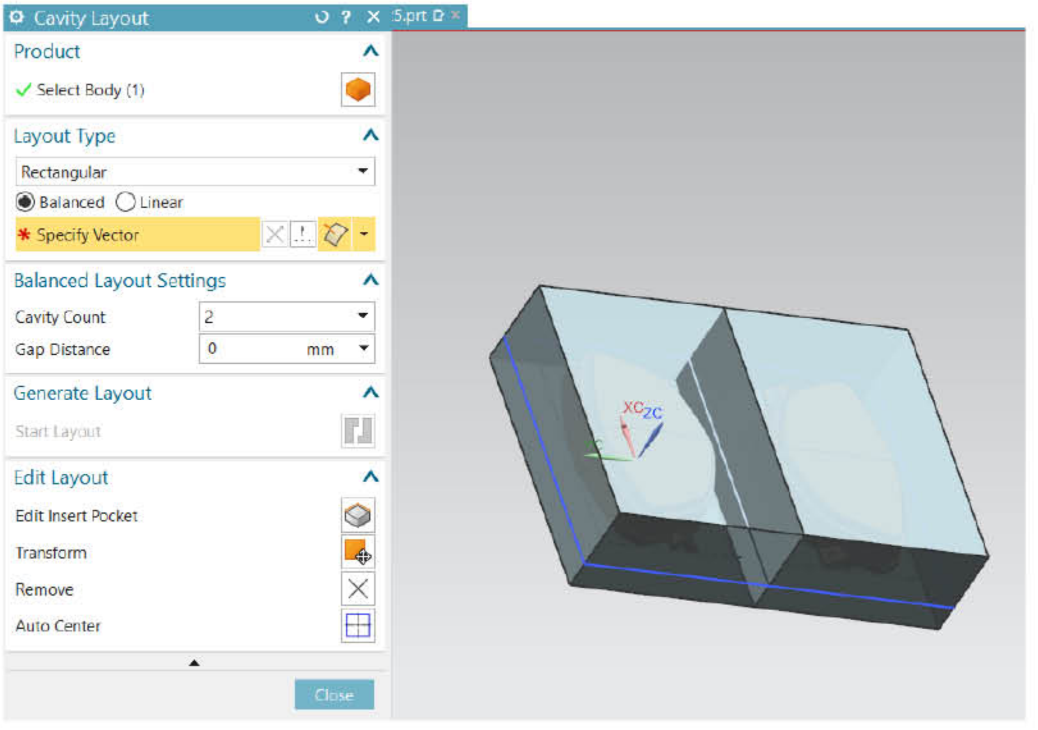
General: Shrinkage coefficient along 3 coordinate axes corresponding to different coefficients X, Y, Z

3.3.2. Arrange the number of cavities and mold cores (Layout).
On the Mold Wizard toolbar, select Cavity Layout in the Main tab

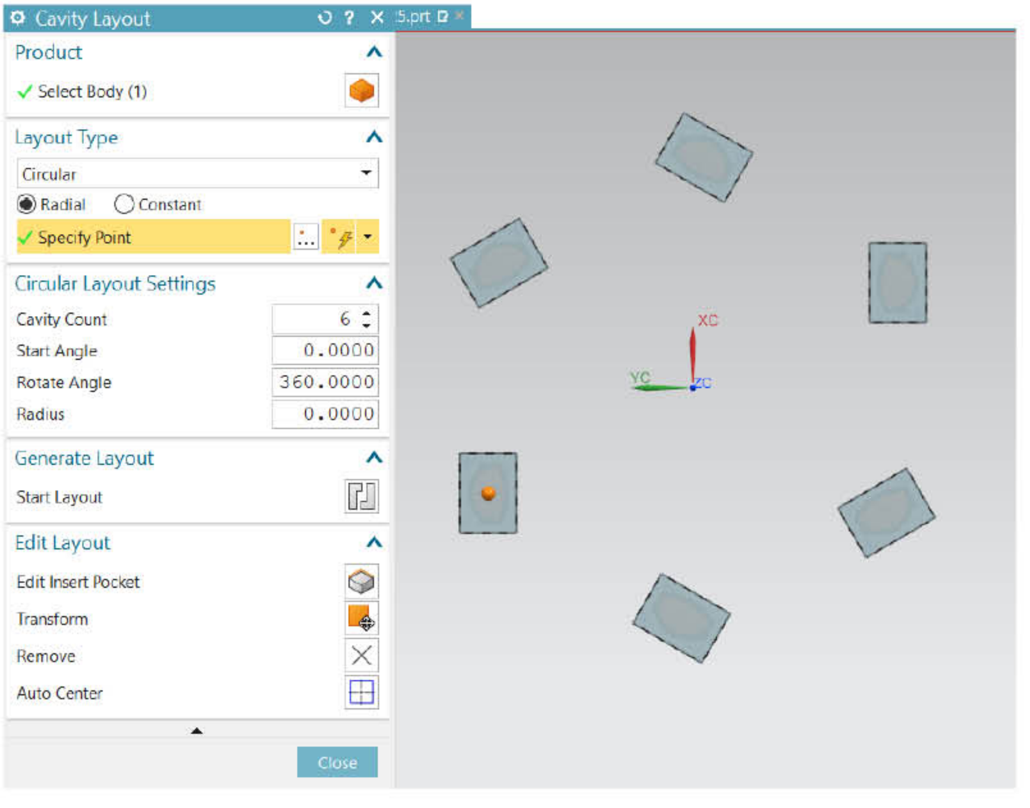


A mold includes 1 or more Core/ Cavity. With Mold Wizard, users can easily choose how to arrange the Core/Cavity cluster in a mold to facilitate the placement of nozzles and subsystems, simple operation:





Arrange 2 rectangular mold cavities.



Arrange 6 round molds.

3.4. Mold separation process (Parting Process)

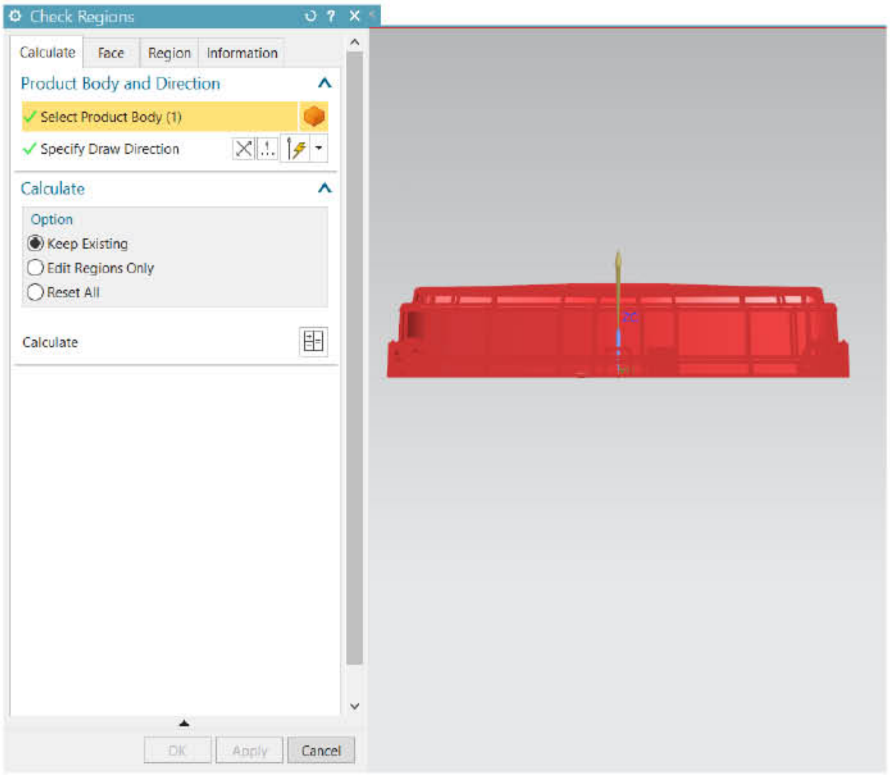
3.4.1. Choose the mold withdrawal direction

On the Mold Wizard toolbar, select Check Regions in the Parting Tool tab.

To withdraw the mold smoothly, the product must be designed with a reasonable bevel angle.

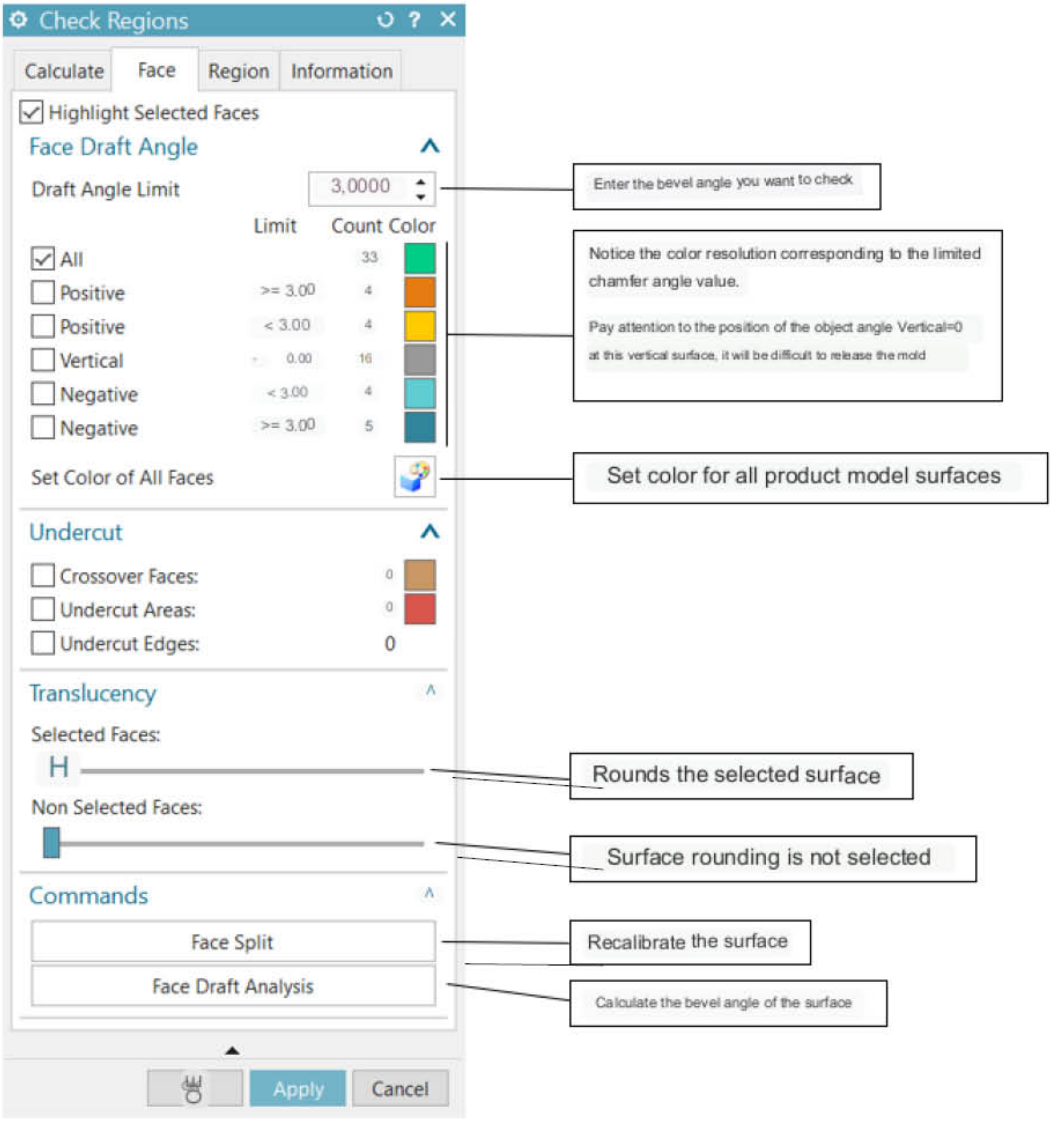
In Mold Wizard, there is a function to measure the bevel angle using a color strip with a limited bevel angle specified by the mold designer.

Areas that do not meet the requirements can be corrected right in the mold design environment itself.

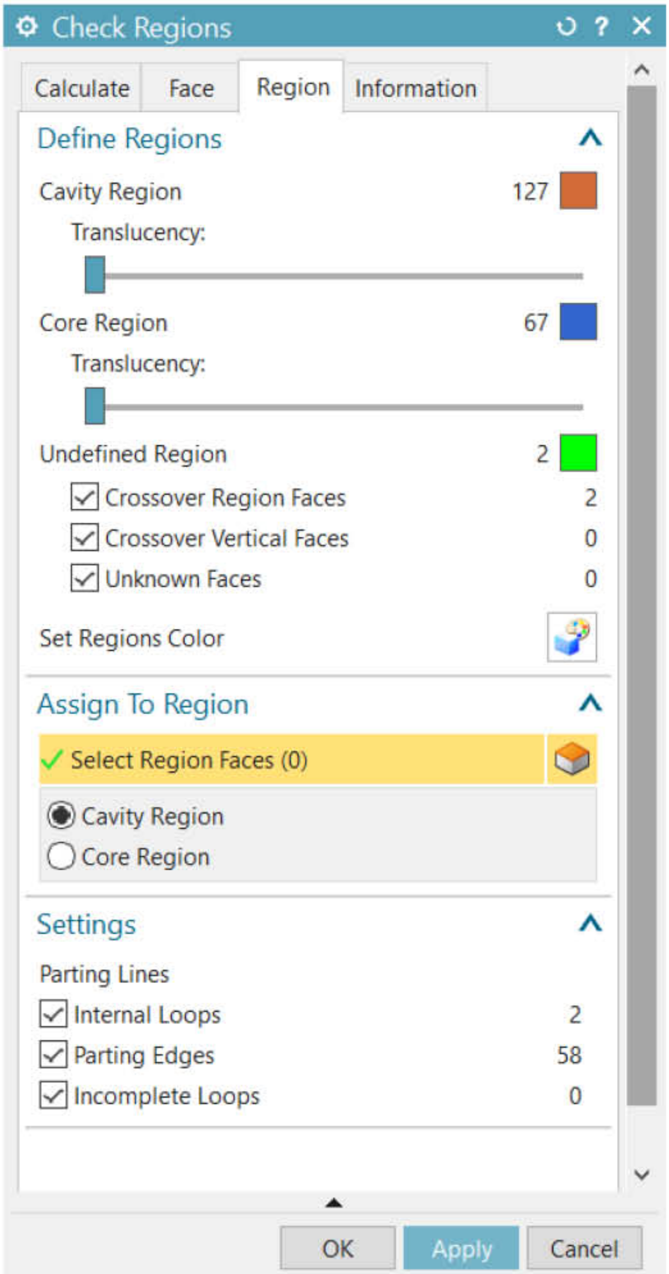


Select Calculate to calculate then select Apply

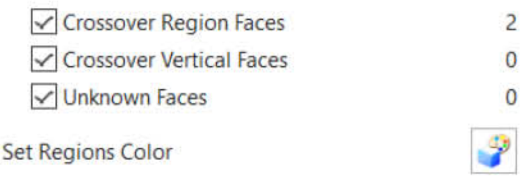
Switch to Face card


Check Regions
Calculate
Face
Region
Information
Highlight Selected Faces
Face Draft Angle
Draft Angle Limit
3,0000
Limit
Count Color
All
33
Positive
>= 3.00
4
Positive
< 3.00
4
Vertical
0.00
16
Negative
< 3.00
4
Negative
>= 3.00
5
Set Color of All Faces
Undercut
Crossover Faces:
0
Undercut Areas:
0
Undercut Edges:
0
Translucency
Selected Faces:
H
Non Selected Faces:
Commands
Face Split
Face Draft Analysis
Apply
Cancel
Enter the bevel angle you want to check
Notice the color resolution corresponding to the limited chamfer angle value.
Pay attention to the position of the object angle Vertical=0
at this vertical surface, it will be difficult to release the mold
Set color for all product model surfaces
Rounds the selected surface
Surface rounding is not selected
Recalibrate the surface
Calculate the bevel angle of the surface

3.4.2. Determine Core and Cavity areas
Switch to the Region tab

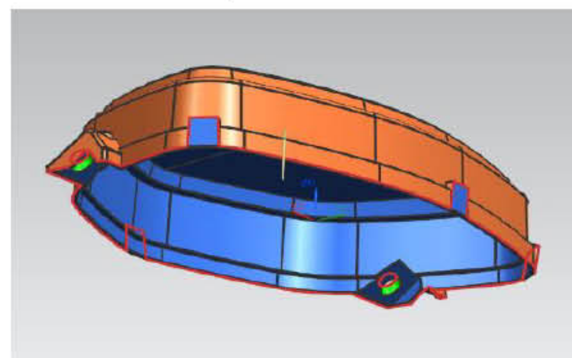


Click to select 3 items in the Undefined Region.



select Set Regions Color to separate the detailed color into 2 parts :Core and Cavity

Details after color separation



Orange (Cavity)

Blue (Core)

Green (Undetermined)

You can change the color of each area by clicking on the color box you want to change and choosing a familiar color.

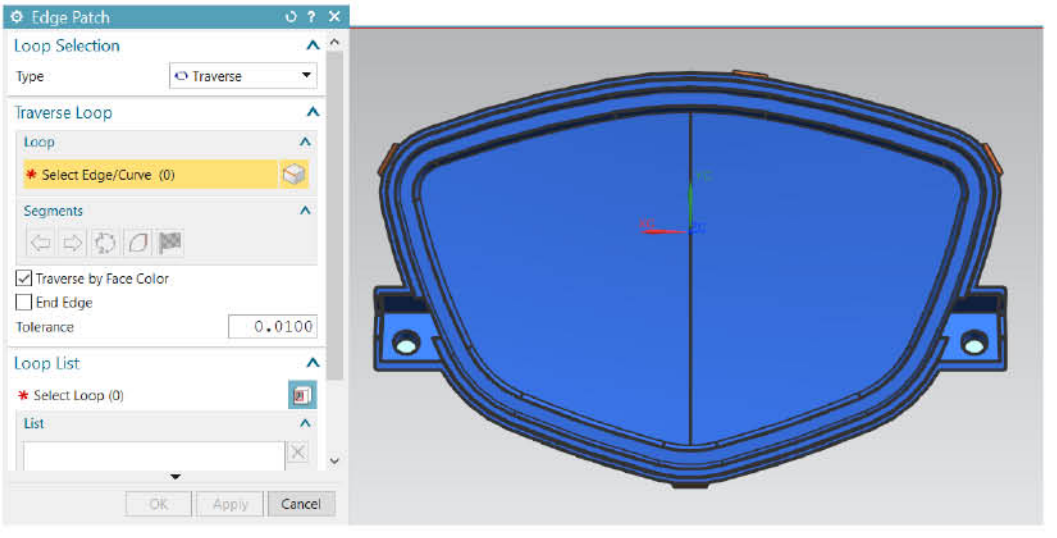
Click Cavity Region or Core Region and then select faces to add or remove desired faces.

3.4.3. Patch holes
There are 2 ways to patch the hole, use Patch Surface or Use the surface creation tools in the Surface toolbar to create patch surfaces.

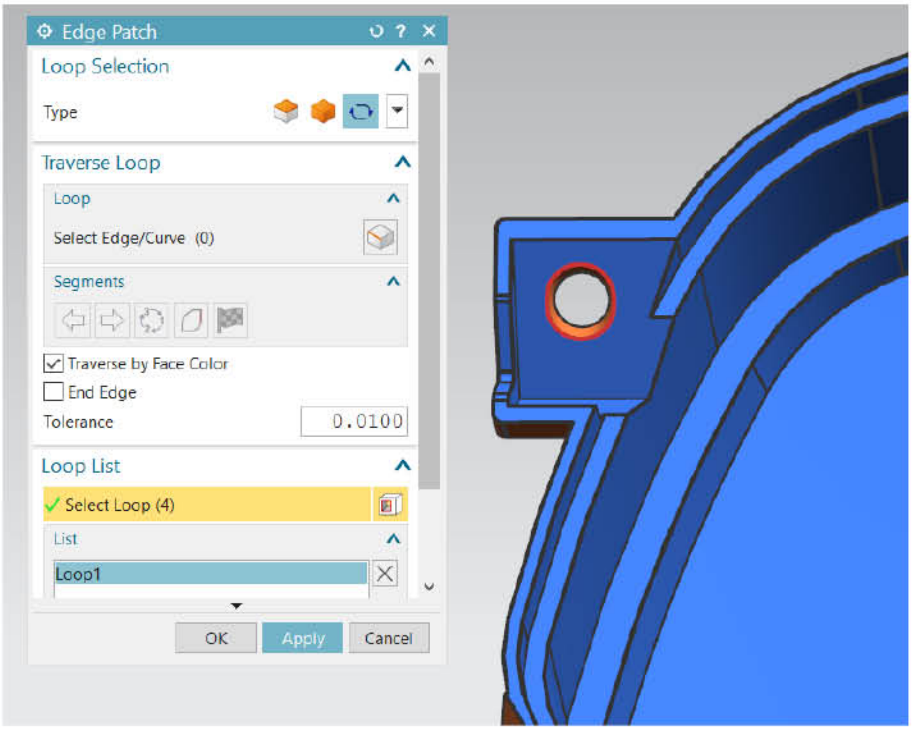
Method 1. Use Patch Surface

On the Mold Wizard toolbar, select Patch Surface in the Parting Tool tab.

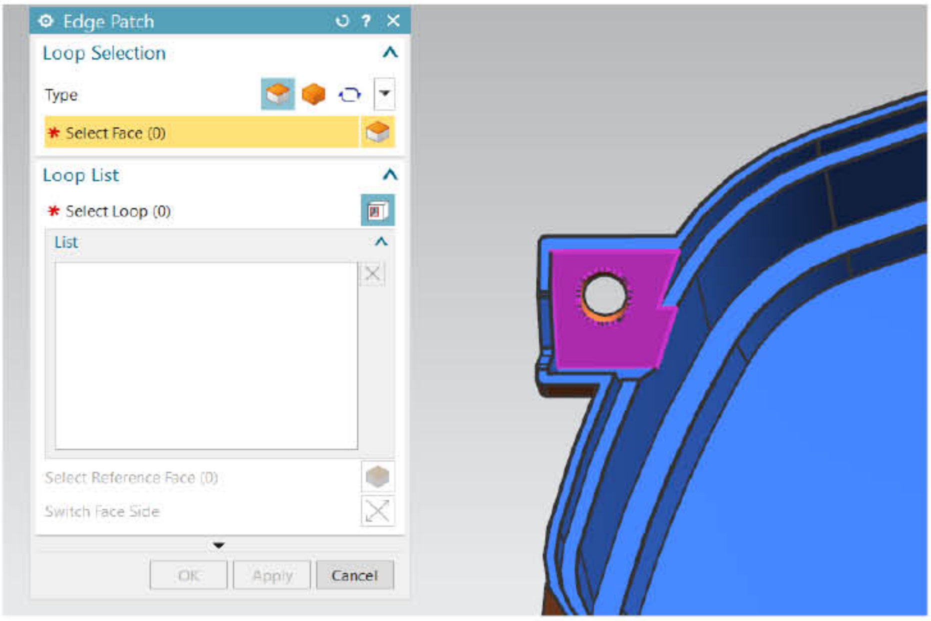
It is necessary to seal the openings to create the secondary molding surface



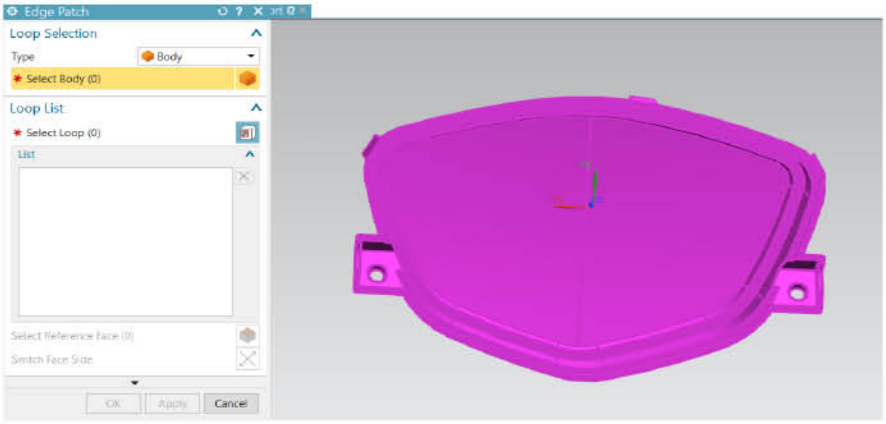
Patch Surface interface



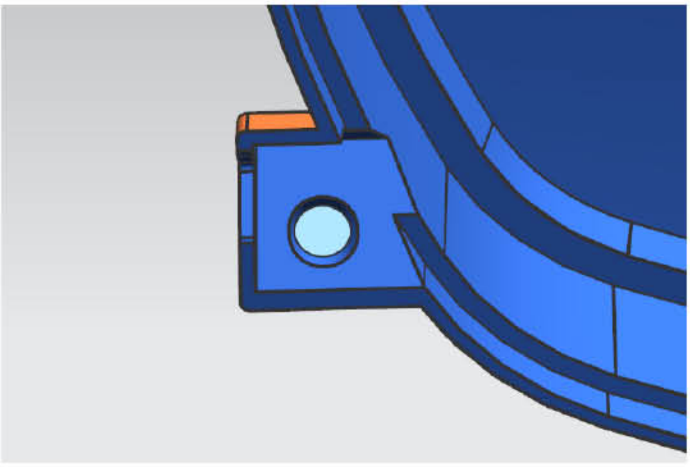
Traverse: select each hole you want to patch, the product will be patched one by one



Face: Select the surface containing the hole you want to patch, the holes on that surface will be patched.

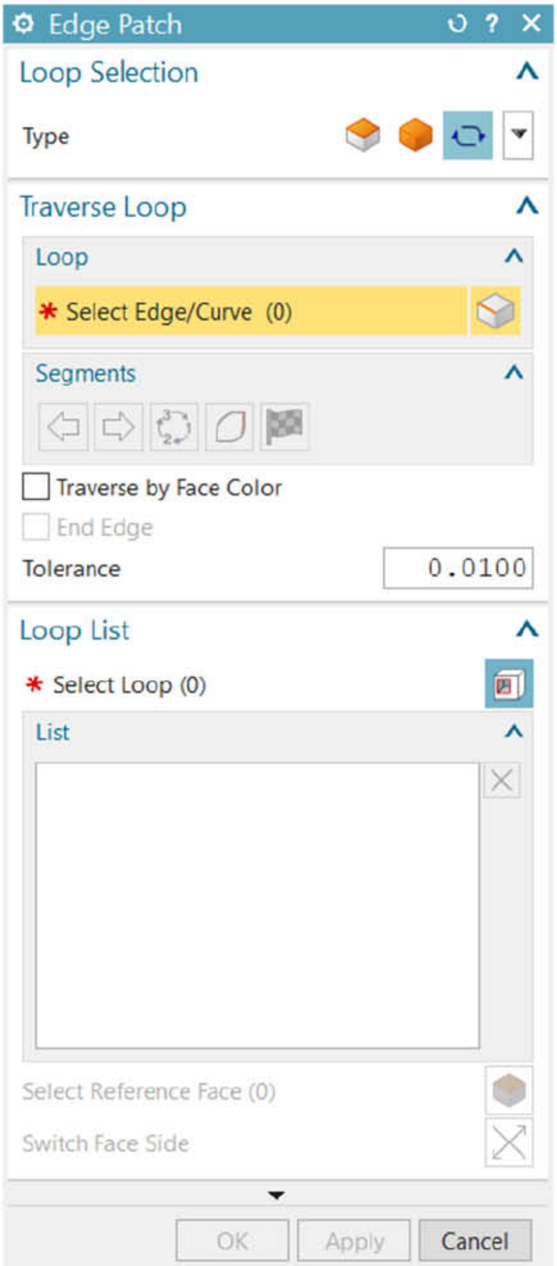


Body: select the detail with holes you want to patch. All holes on this detail will be patched



The product after being patched with holes using the Patch Surface tool

On the Mold Wizard toolbar, select Patch Surface in the Parting Tool tab.

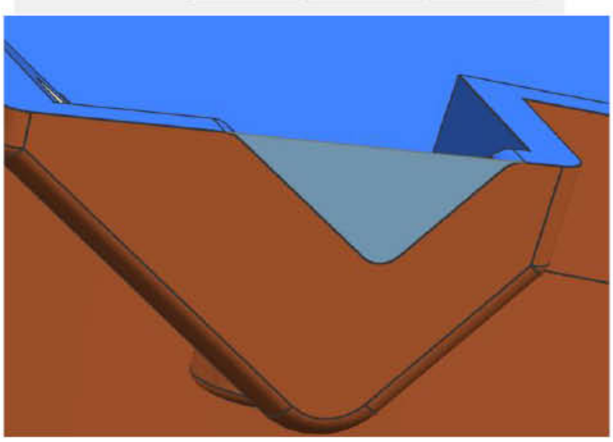


Select the Traverse method

Unchecked Traverse by Face Color

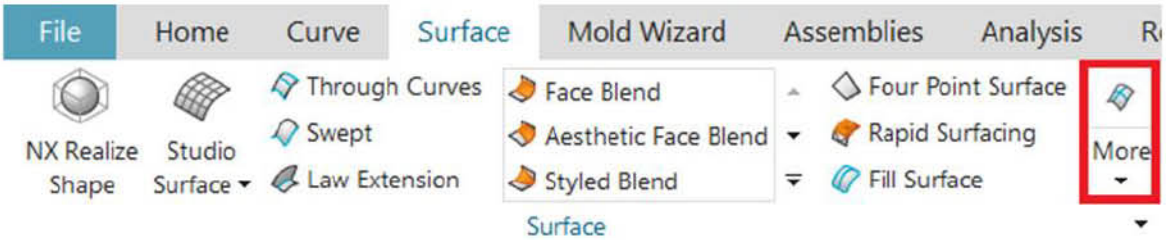
Click to select the first Curve line of the hole, then press Accept until the end of the open profile and then press Close Loop

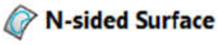
If the path does not have the correct profile, select Cycle Candidates to change direction.



Details after being patched with Patch

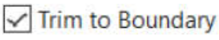
Method 2. Use the surface creation tools in the Surface toolbar to create patch surfaces.



Select **N-Sided Surface** 

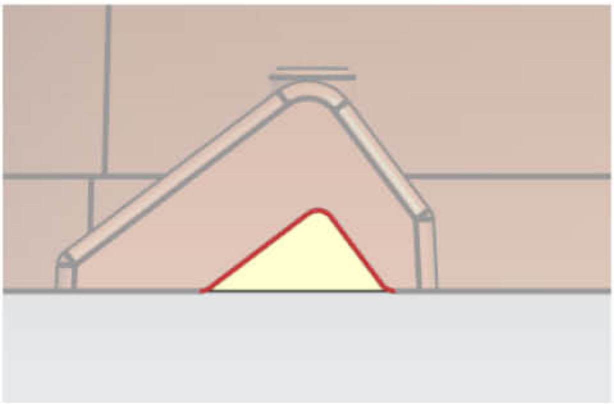
in More.

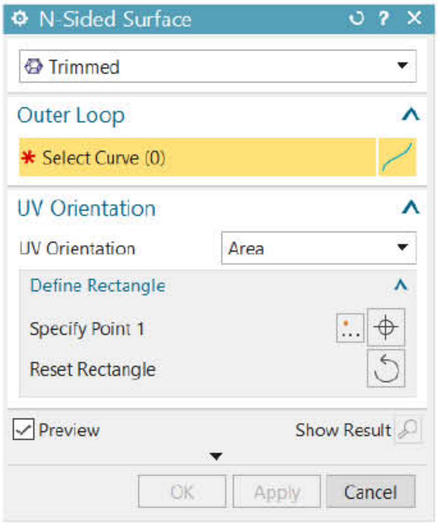
Click the **Dialog Options icon**  select **N-Sided Surface (More).**

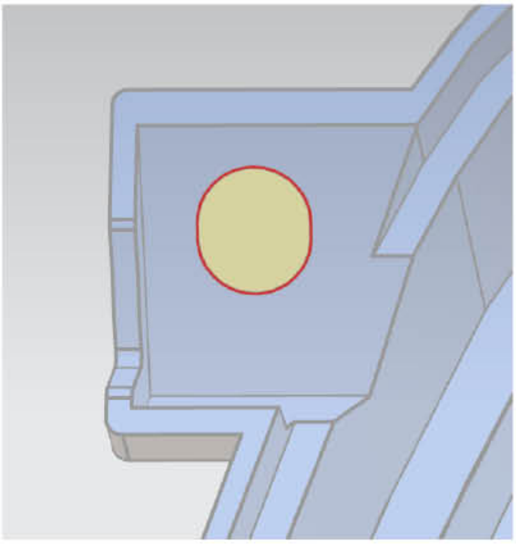
Select 

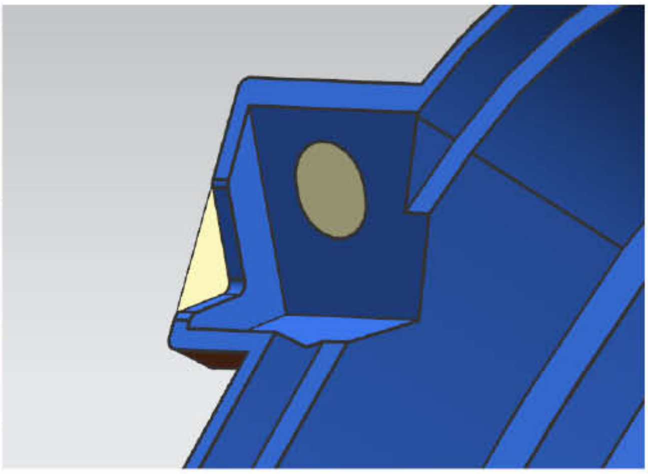
in Settings.

Select Curve: Select the edges you want to constrain as limits for the created face.



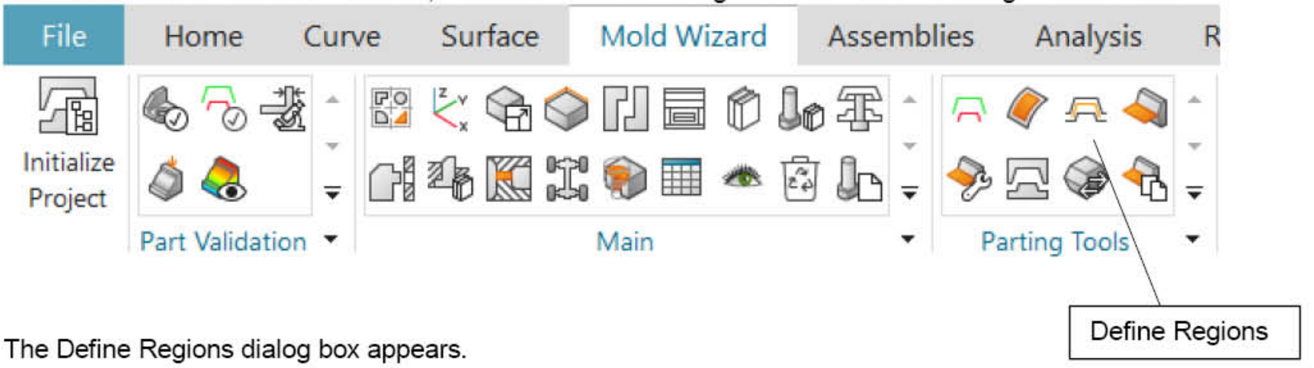




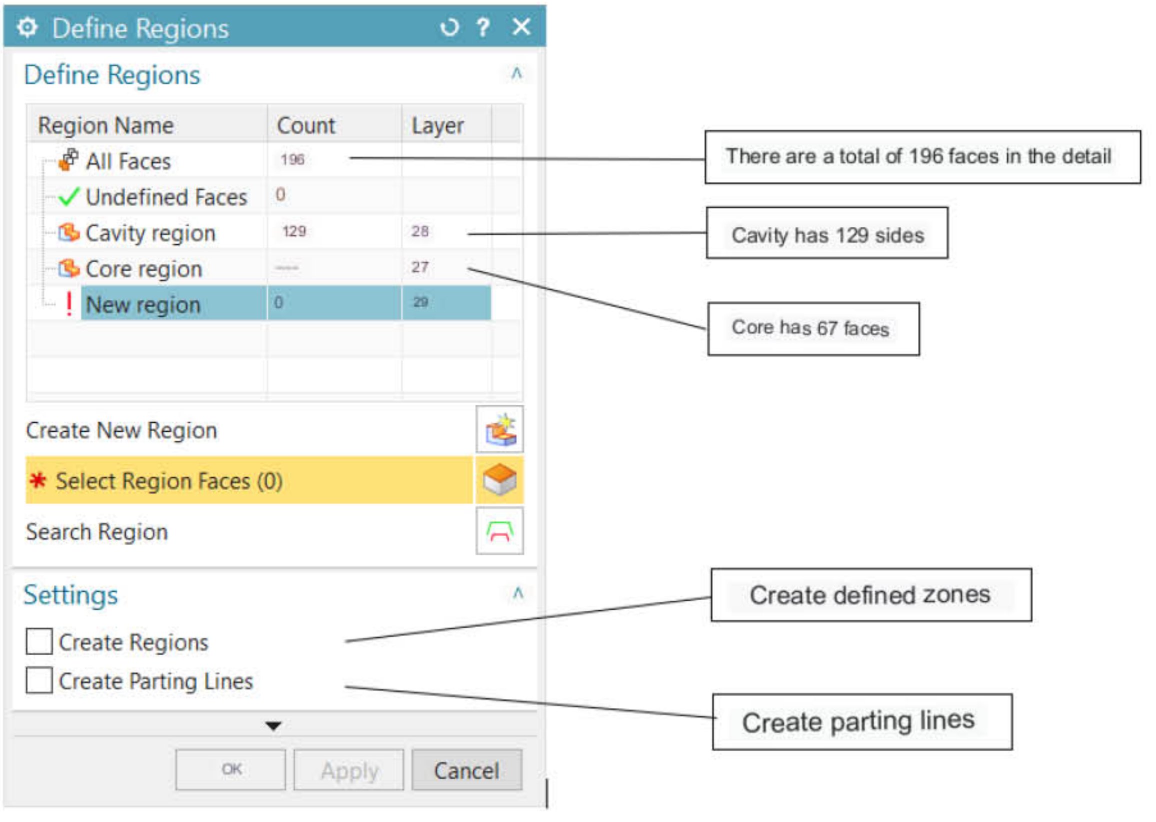


It is necessary to adjust the Cavity and Core part faces so that the mold can be separated

On the Mold Wizard toolbar, select Define Regions in the Parting Tool tab.

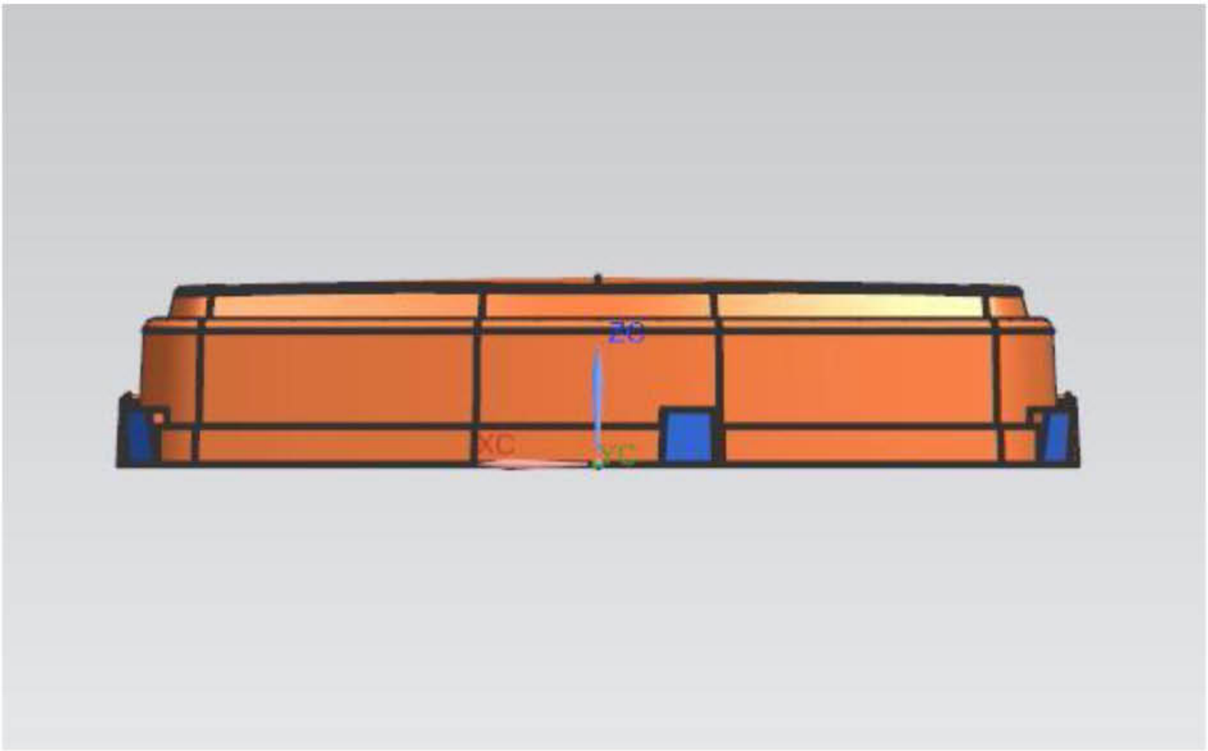


The Define Regions dialog box appears.

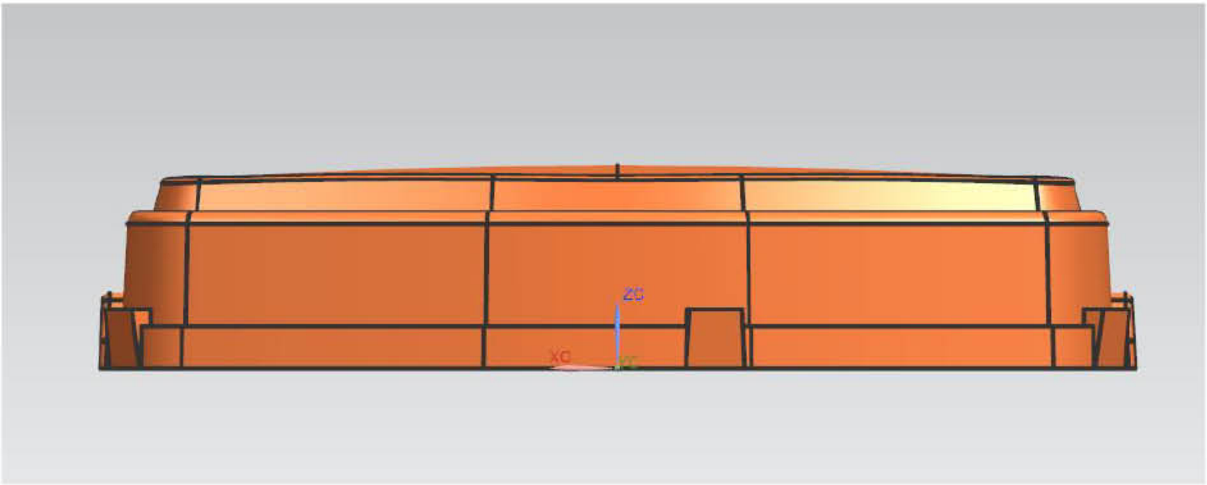


Click Cavity Regions then click to select the surfaces you want to add to the Cavity region

Click Core Regions then click to select the faces you want to add to the Core region

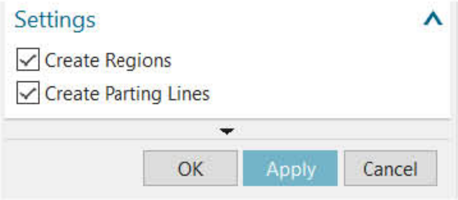


Details before correction (Cavity and Core are mixed up)



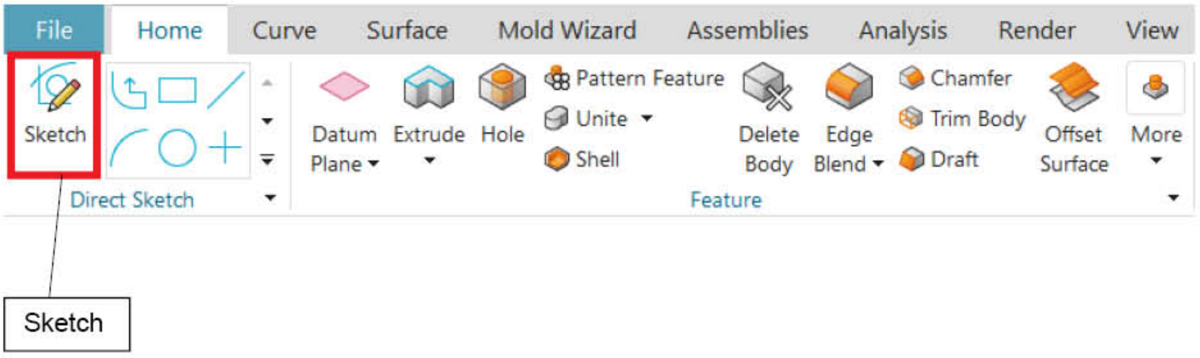
Details after calibration (Cavity and Core are separate)

Click Create Region and Create Parting line Apply

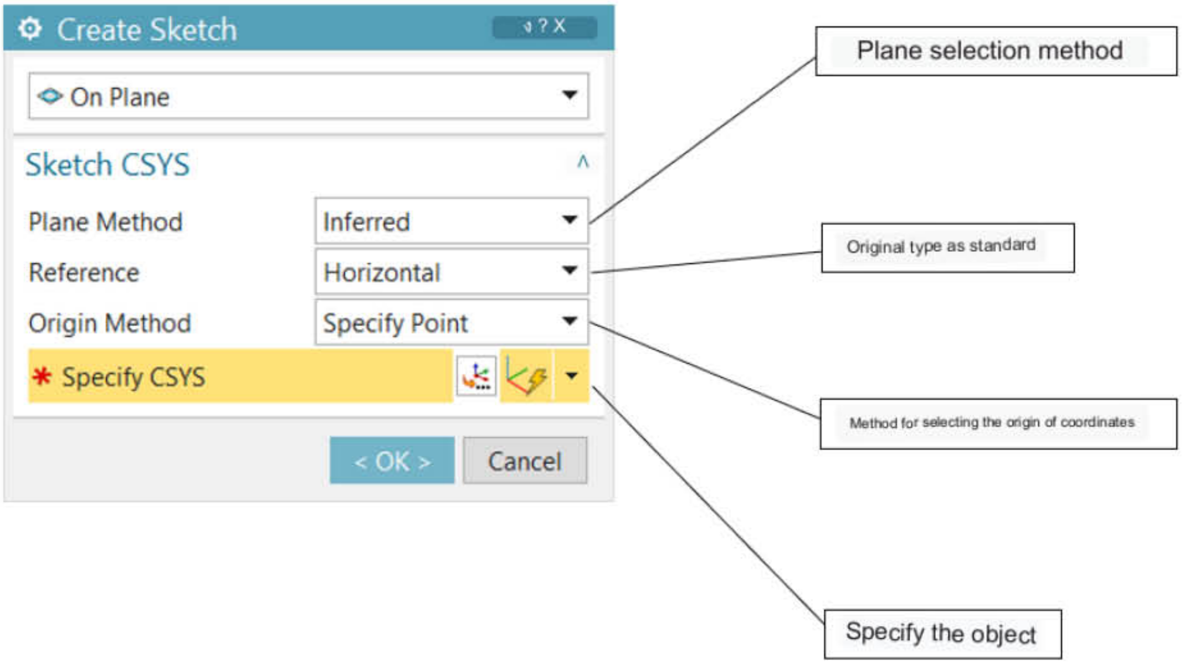


3.4.4. Create parting surface.
3.4.4.1 Creating hand parting faces (manually)

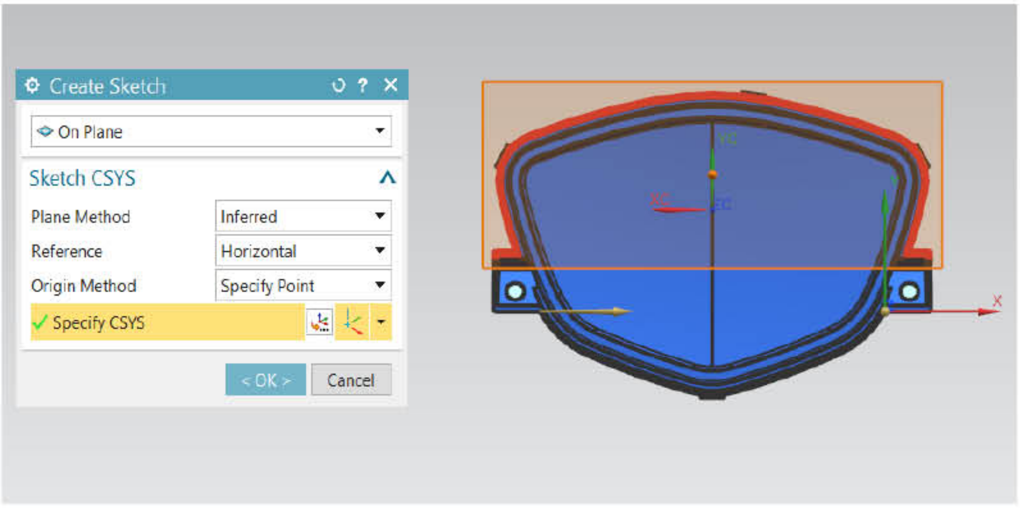
On the Home toolbar, select Sketch in the Direct Sketch tab.



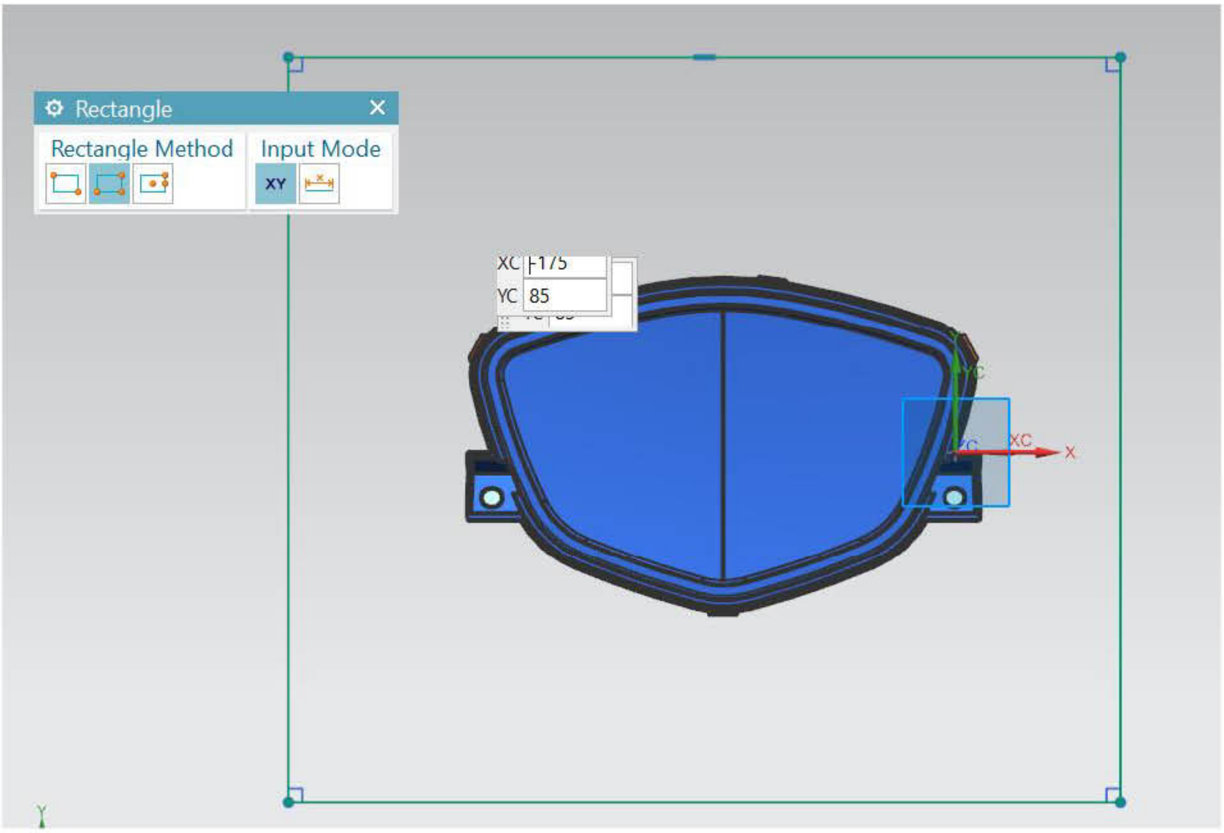
The Create Sketch dialog box appears.



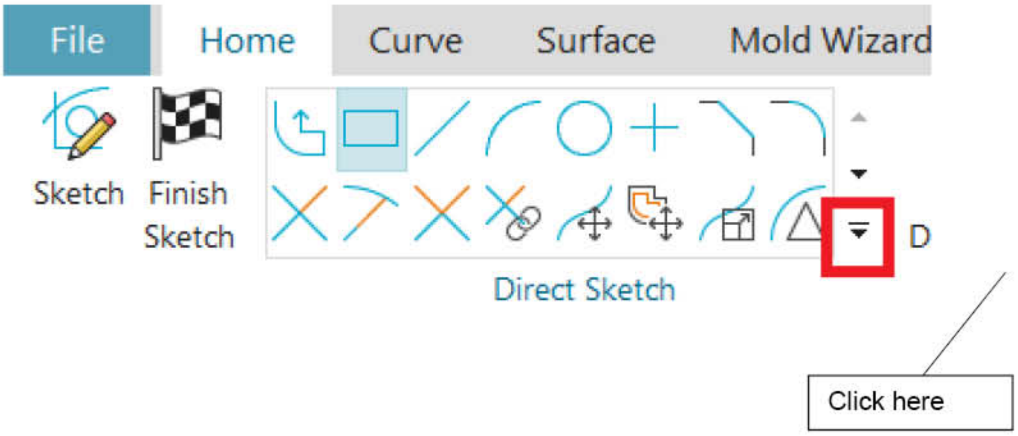
Select the plane you want to draw the parting face.



Select Rectangle , drag a rectangle around the part (must cover the workpiece).

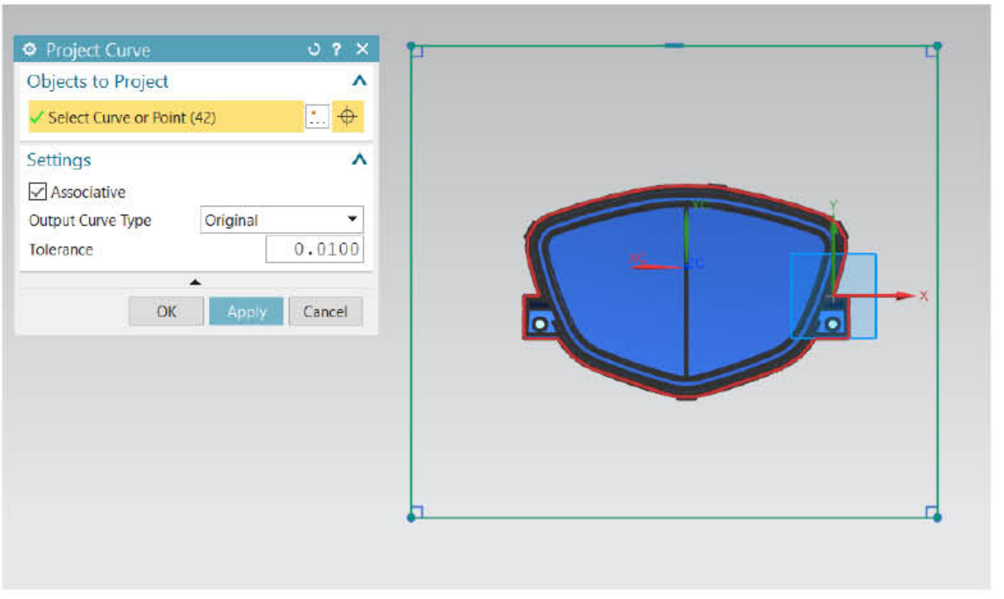


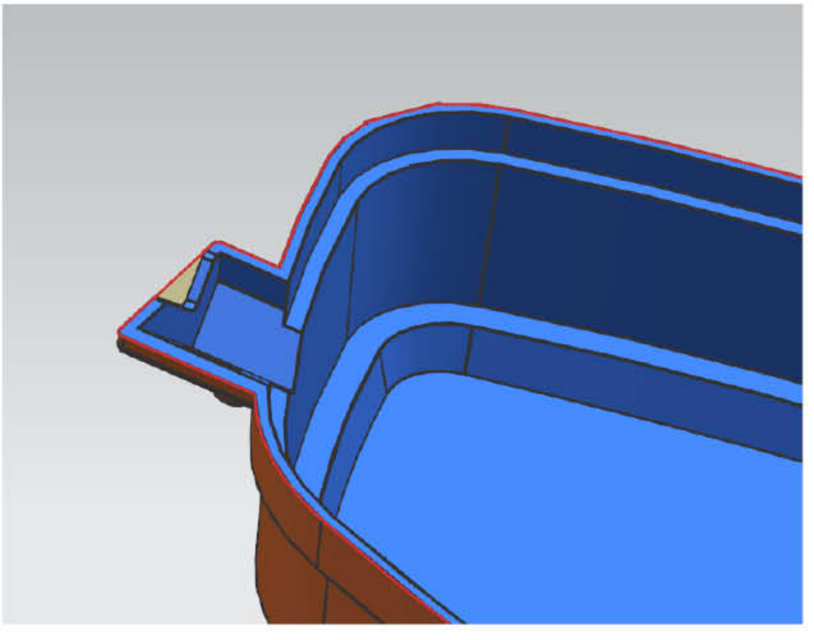
Click the arrow to pull out the tools in the Direct Sketch tab.



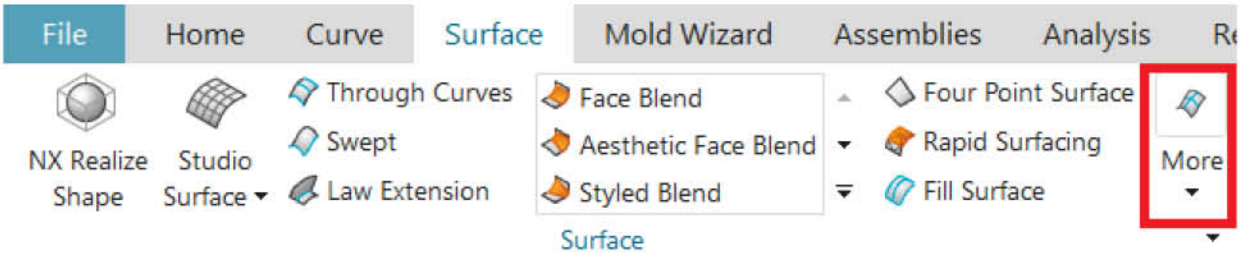
Select Project Curve , click to select all edges surrounding the product that lie on the selected plane.

Select Apply

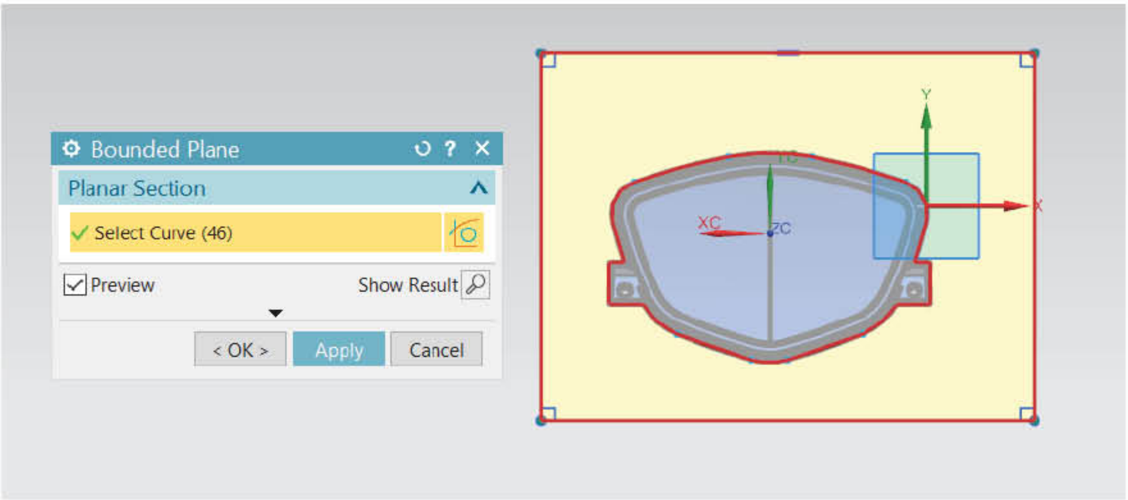




On the Surface toolbar, select More in the Surface tab.

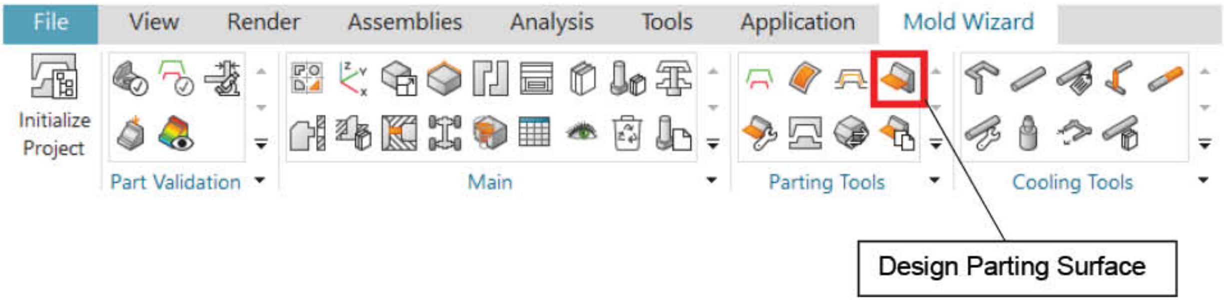


Click **Bounded Plane** Apply to create the plane.

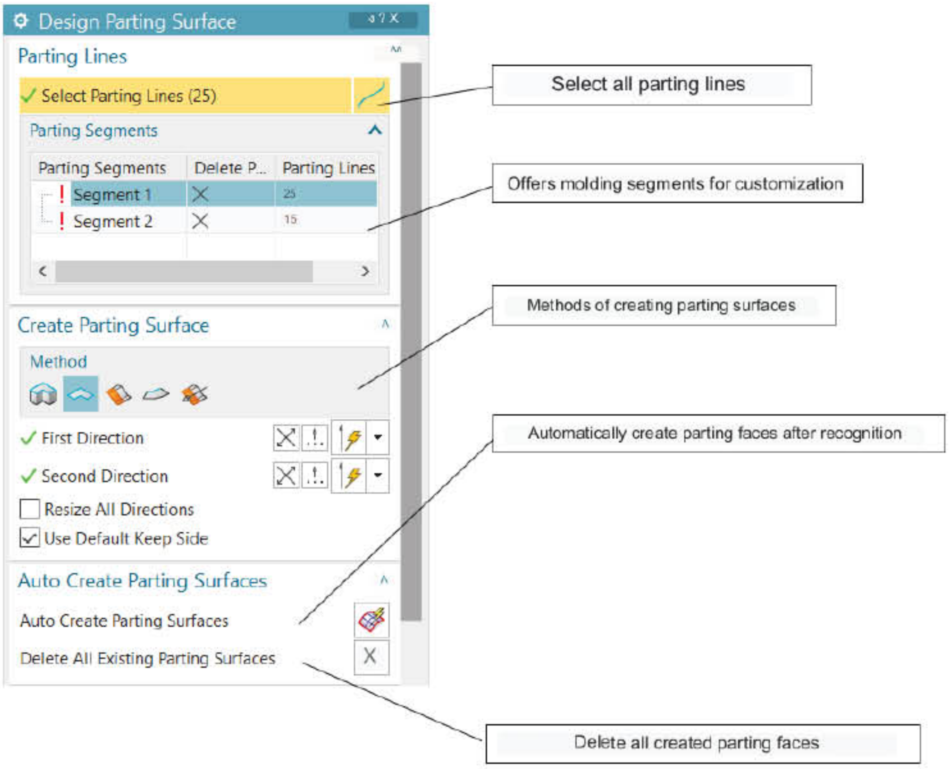


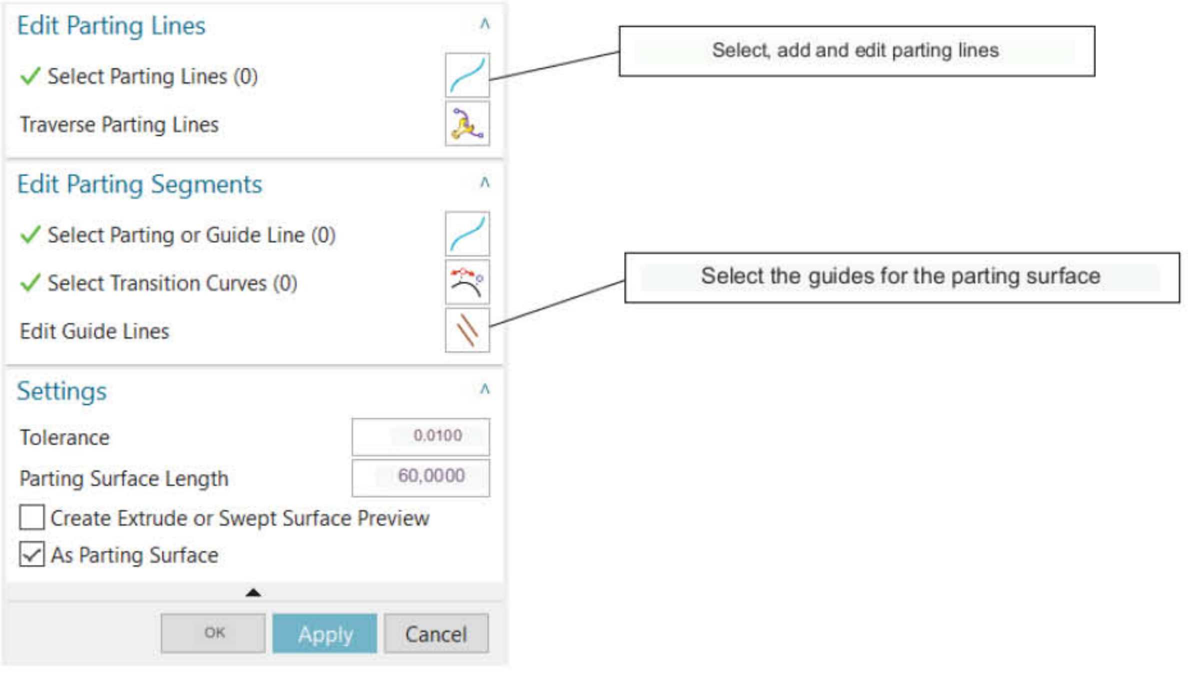
3.4.4.2. Create automatic parting surfaces

On the Mold Wizard toolbar, select Design Parting Surface in the Parting Tool tab.

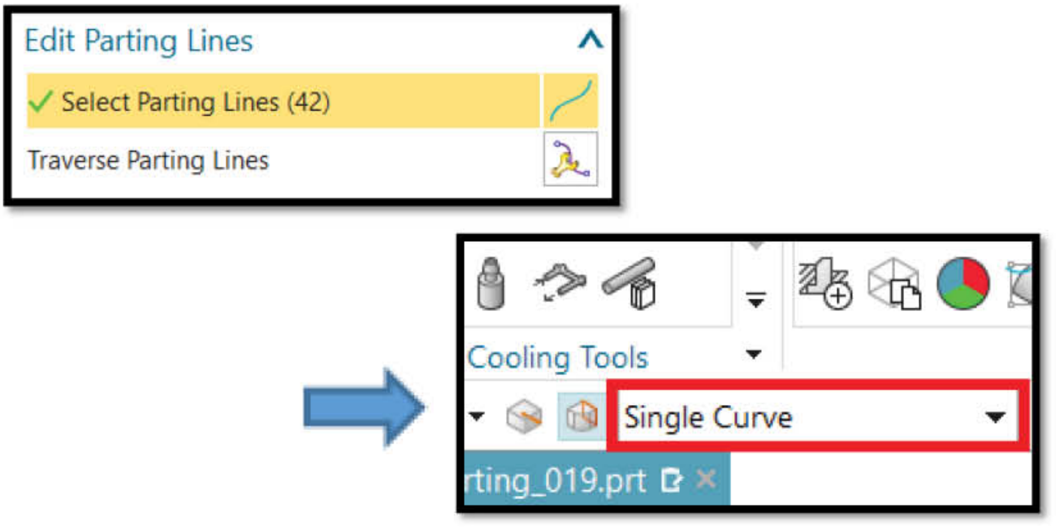


The Design Parting Surface dialog box appears.

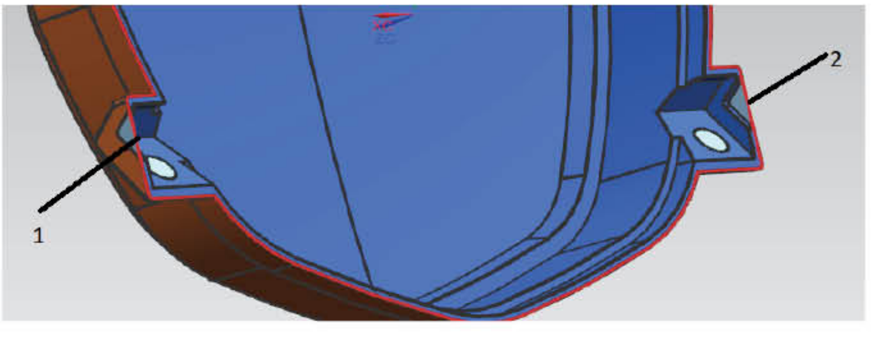




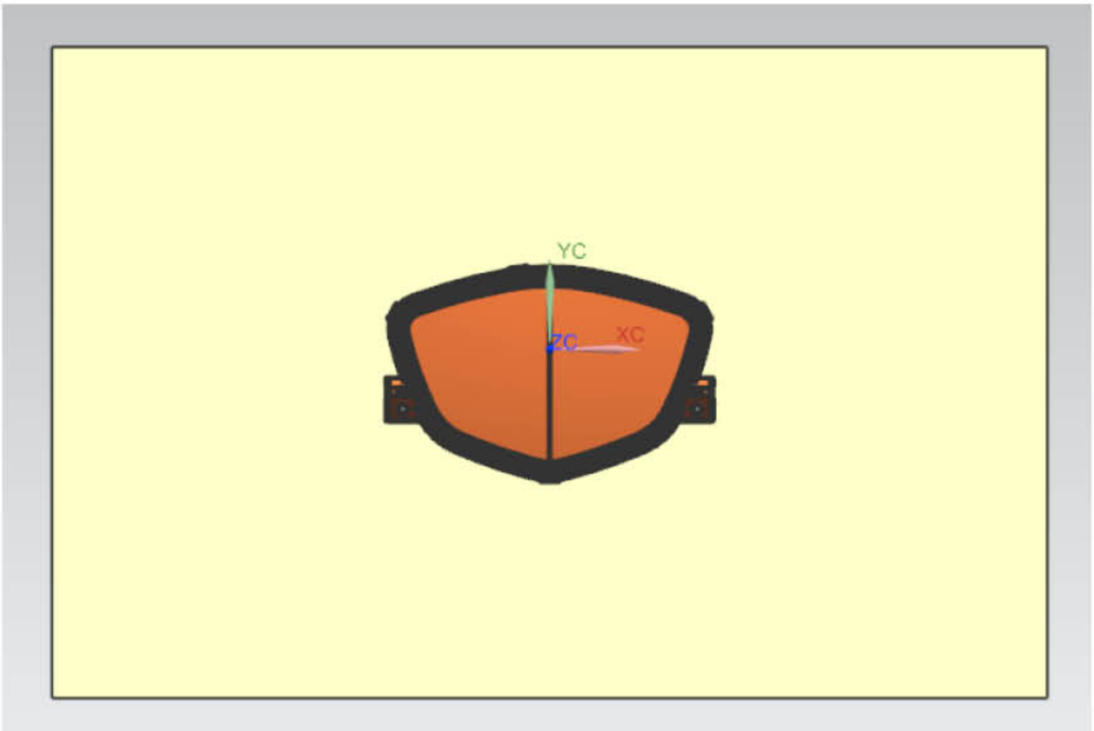
Select Select Parting Lines and select Single Curve in the Curve Rule section.



Select 2 more edges to form a closed line.

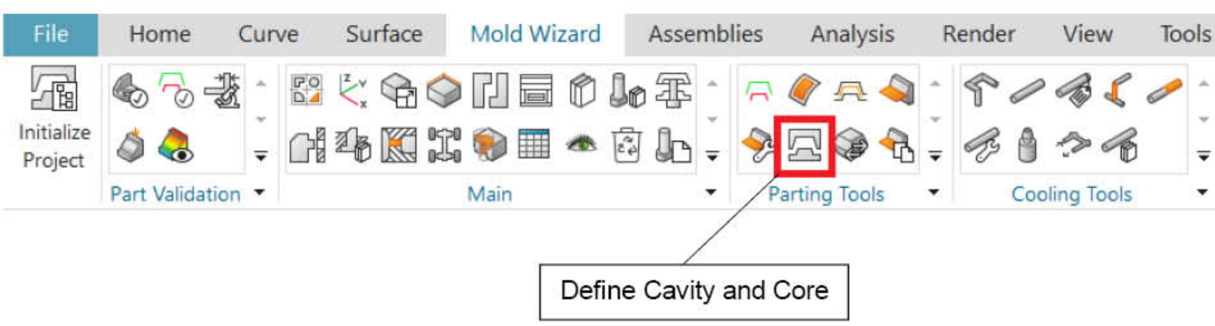


Click Apply then Ok to finish the command.

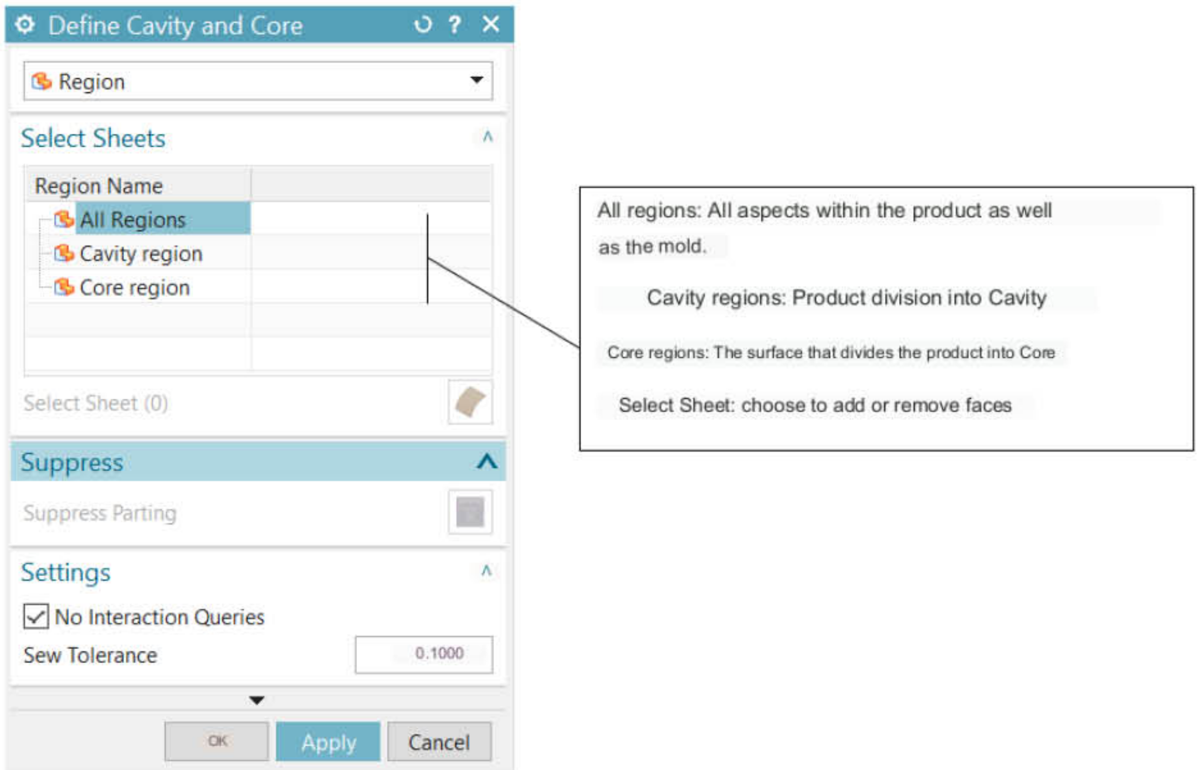


3.4.5. Create Core/Cavity

On the Mold Wizard toolbar, select Define Cavity and Core in the Parting Tool tab

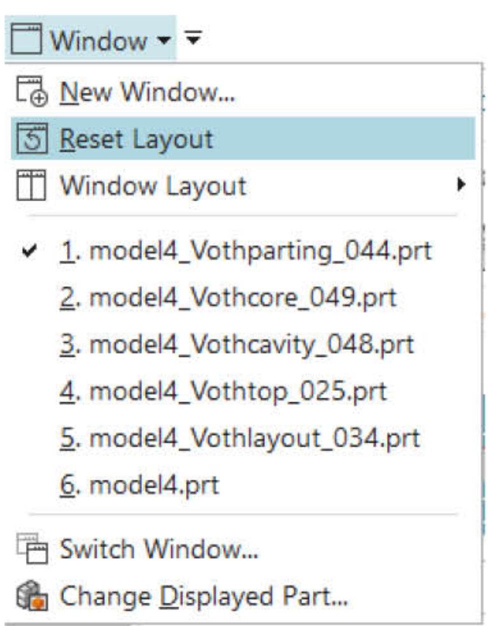


The Define Cavity and Core dialog box appears

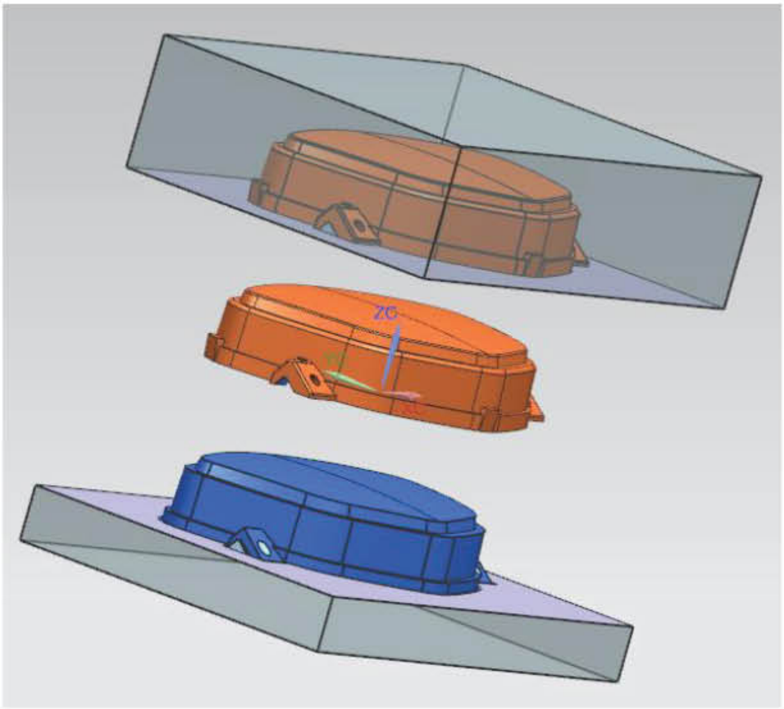


Click All Regions then click Apply then Ok.

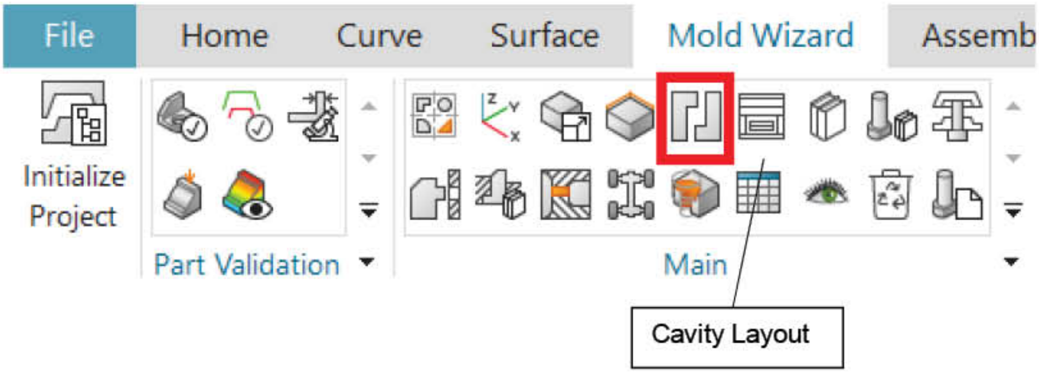
Go to Windows and select the file named "VothTop"



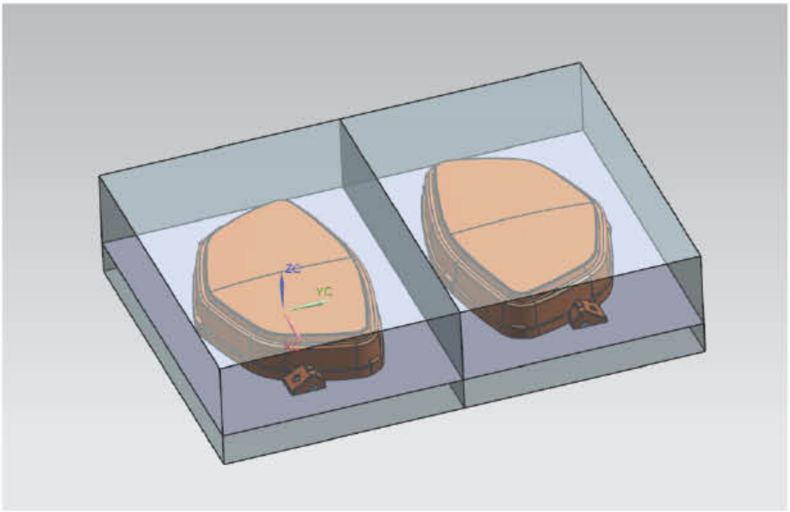
Use **Assembly > Exploded View** Exploded Views ▾ **> New Explotion > Exploded View > Edit Explotion** to see the decomposition of the two mold pieces.



On the Mold Wizard toolbar, select Cavity Layout in the Main tab



How to use is instructed in section 2.3.2. Layout (Layout).



3.5. Insert mold. (Add Mold Base).

Mold Wizard provides a rich mold library system with indexes from small to large. Users choose and can change according to design ideas.

On the Mold Wizard toolbar, select Mold Base Library and Core in the Parting Tool tab

Use the **Mold Base Library command** to:

mold assembly to the complete **Mold Wizard mold project.**

Edit mold structure and size.

Edit existing standards.

Create new standards.

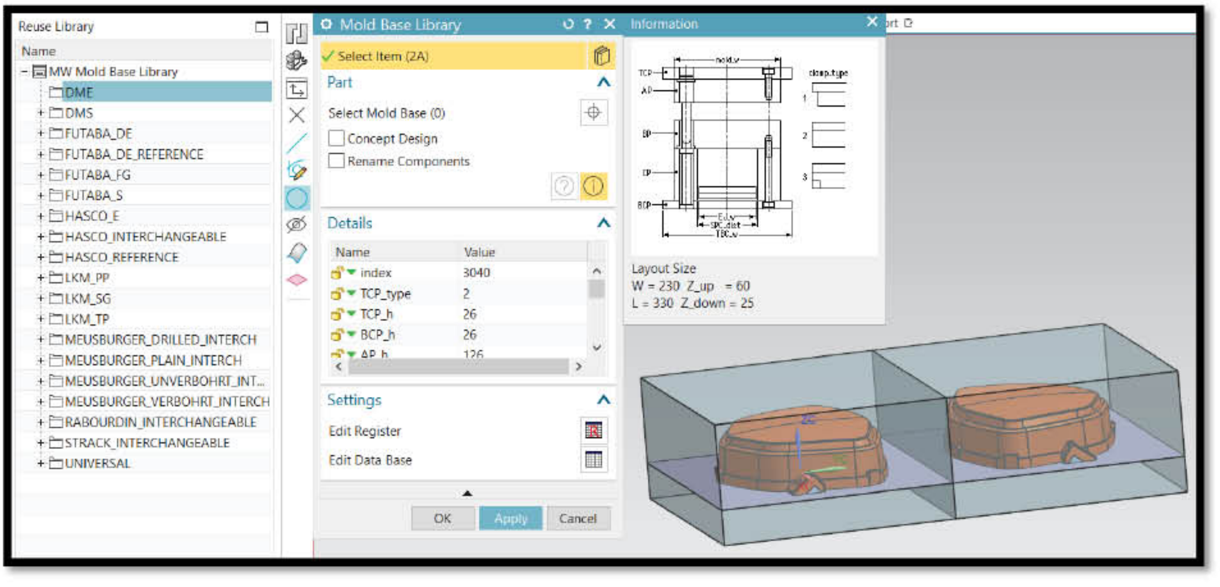
Replace mold jackets with different standards.

Standard Mold Base - Structure one of the components available in the library.

Interchangeable Mold Base - Determines how the panels are arranged and does not use standard dimensions.

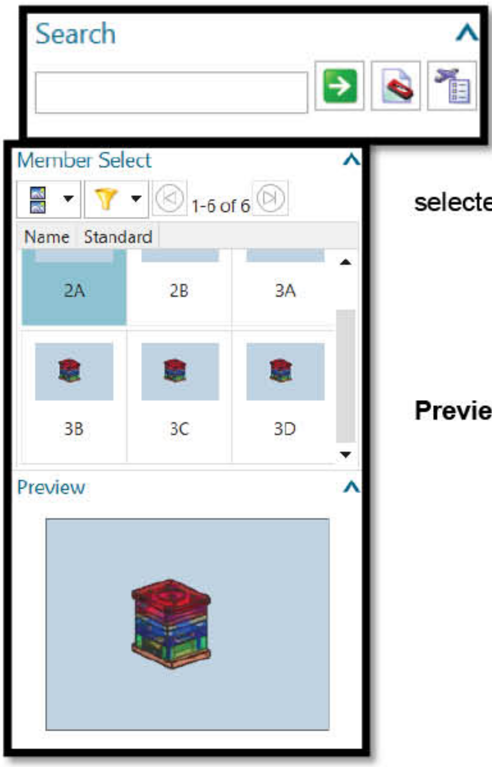
Universal Mold Base - Defines a clear panel layout structure in a combination of thousands of interchangeable parts.

Custom Mold Base – Develop into your company's standard.



On the **Rouse Library bar** in the left corner: all mold standards of mold shell manufacturers are integrated into the software. For example: FUTABA, HASACO... Depending on the working needs as well as which standard the partner chooses, the designer can choose the mold shell company as desired.

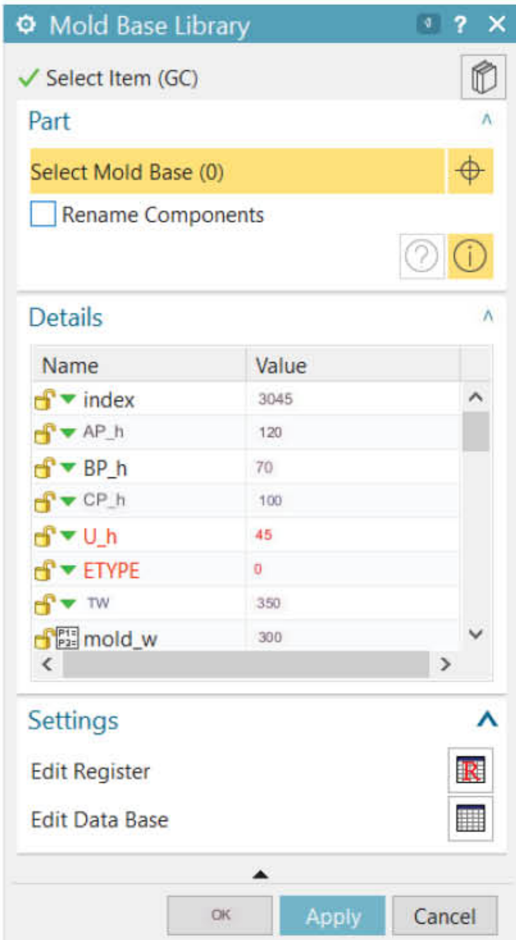
Below the **Rouse Library bar** displays selection and viewing modes.



Searc h: Search for mold standards by name.

Member Select : Displays the mold types within the selected standard

Preview : preview the structure of a selected set of molds



The selected standard, here is GC

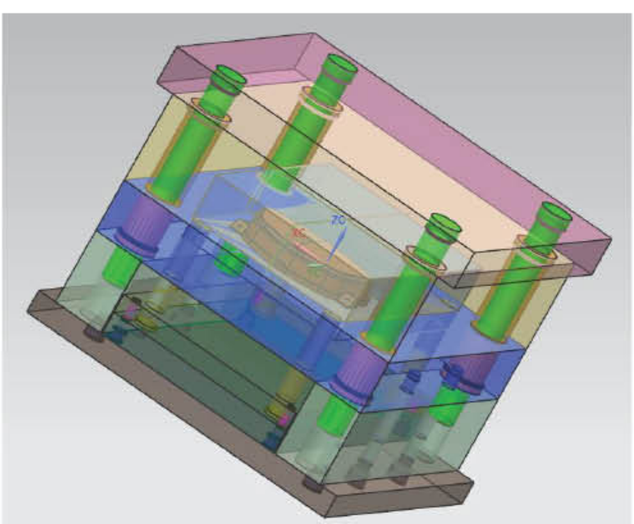
Dimensional parameters when choosing a mold cover can be changed to suit the mold cavity

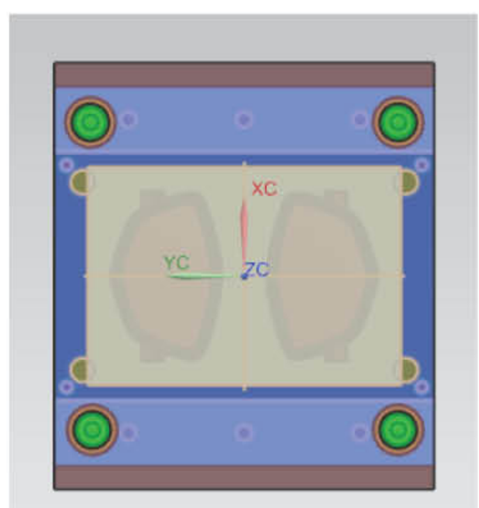
Opens an Excel document that allows you to edit it

Open the data file with detailed tables of contents

Click Apply

Click Apply

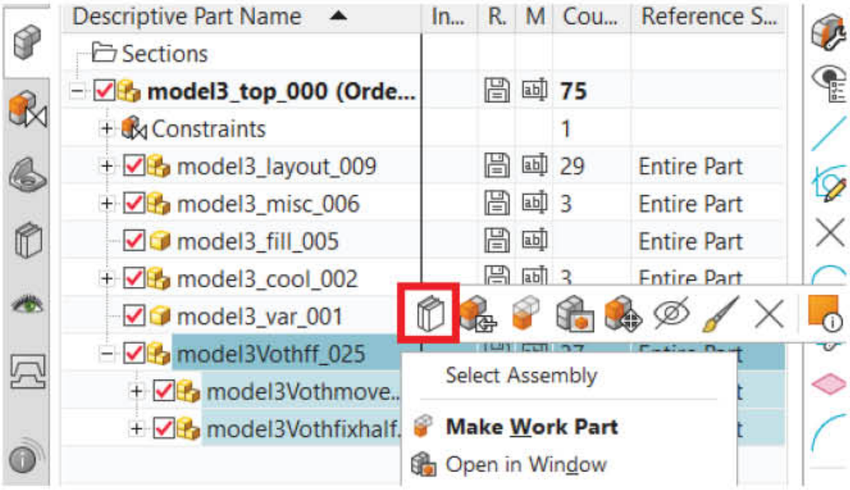




In case you want to correct the parameters of an existing mold, do as follows.

Right click on the assembly name (the assembly for the entire mold is taken from the library). Select Edit Tooling Component to return to the editing window similar to the window when filling in mold parameters.

Or click on the Mold Base Library icon on the Main tab to edit the parameters of the mold.



If you want to replace another type of mold cover, right-click on the mold assembly file name you want to replace and fill in the parameters to get the mold cover.

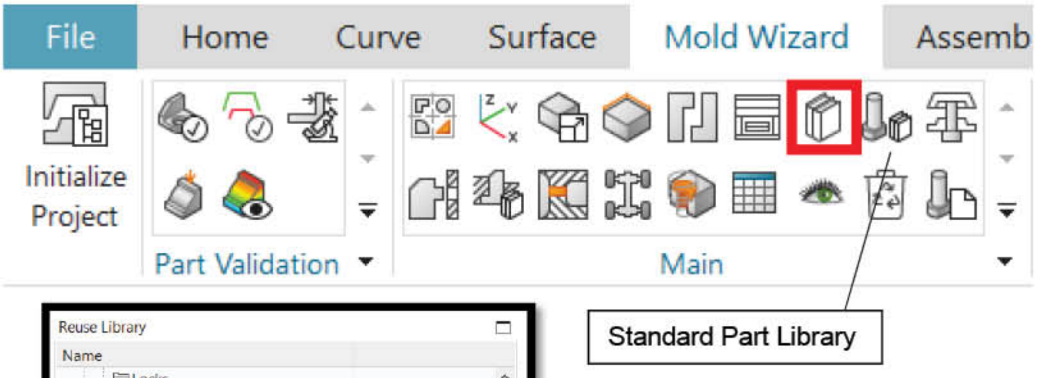
3.6. System design.

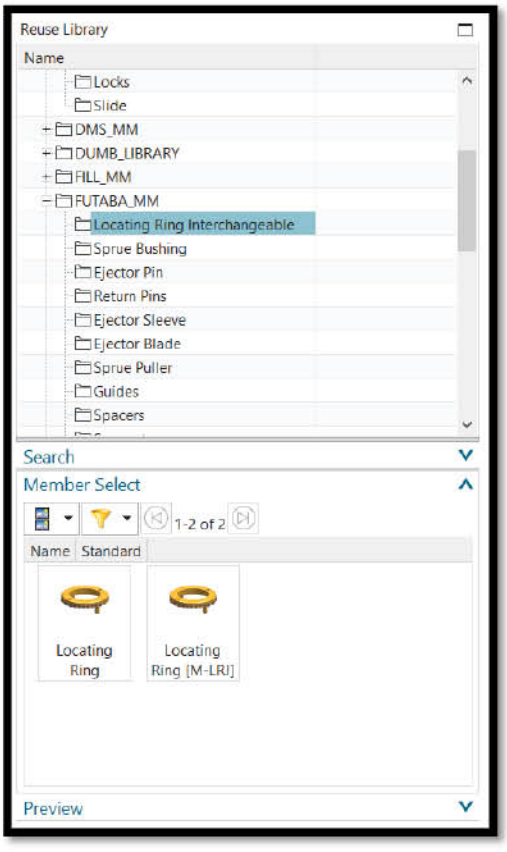
3.6.1. Spray system. (Injection)

Mold Wizard provides a standard library, users can customize appropriate parameters. Selecting details in the spray system (Positioning ring, Spray stem, Channel, Spray nozzle) becomes simple and fast.

Positioning ring

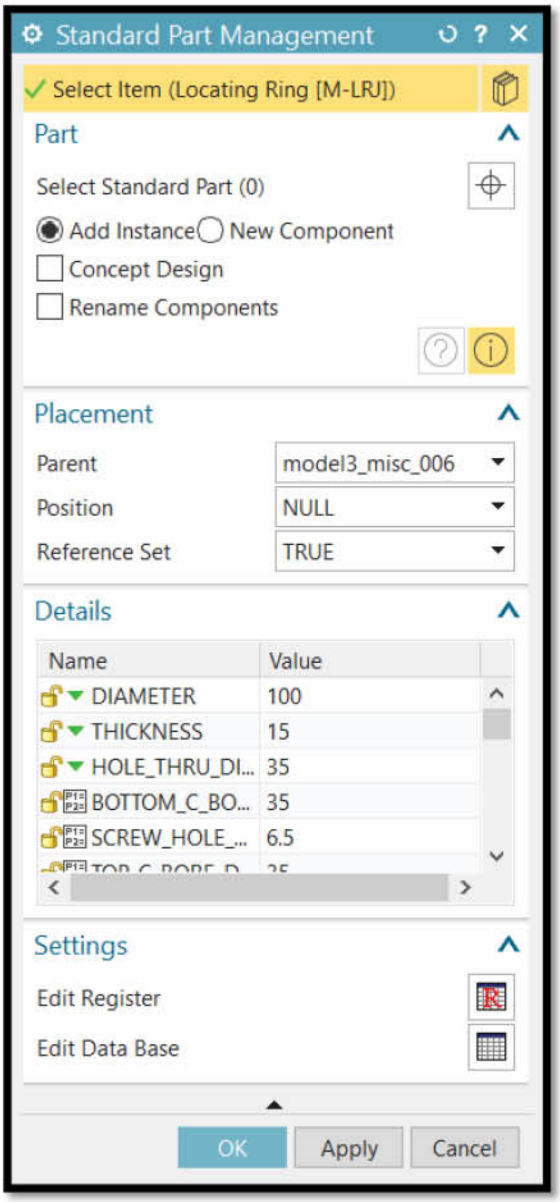
On the Mold Wizard toolbar, select Standard Part Library in the Main tab.





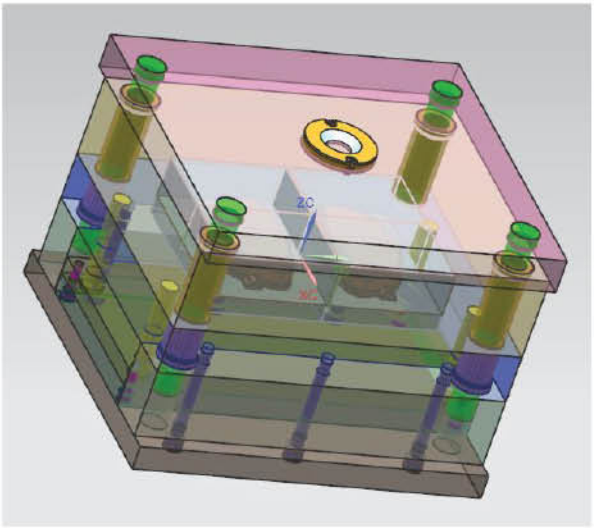
Select the **Rouse Library tab** > in **MW Standard Part Library** > Find the mold row > **FUTABA _MM** > Select **Locating Ring Interchangeable.**

Member Select section : select **locating Ring [M-LR]:** type of locating ring with fixed pins and bolts to locate and clamp tightly on the front clamp plate.



Standard Part Management dialog box, there is a **Details** box that contains the main parameters of the positioning ring that need to be edited.

Click **OK.**



Coat the mold after adding the positioning ring.

To edit the parameters of details in the selected library or change the location, there are the following ways.

Click on Standard Part Library in the Main tab.

Right-click on the detail, select Edit Tooling Component

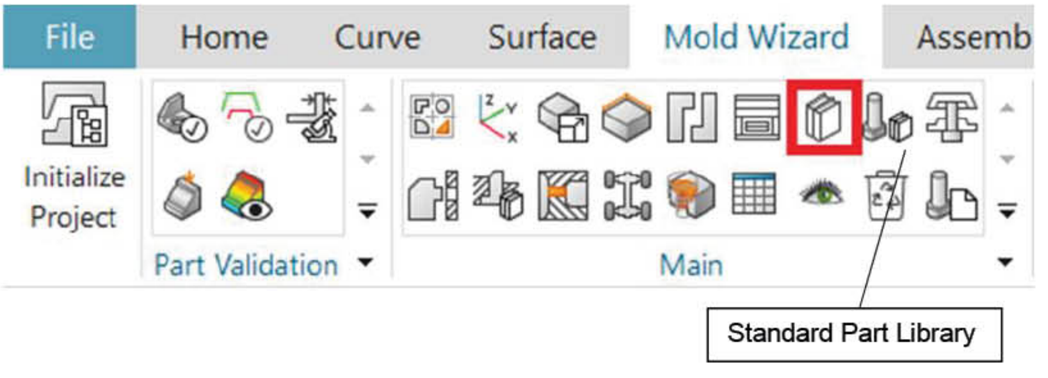
Reposition when you want to change position

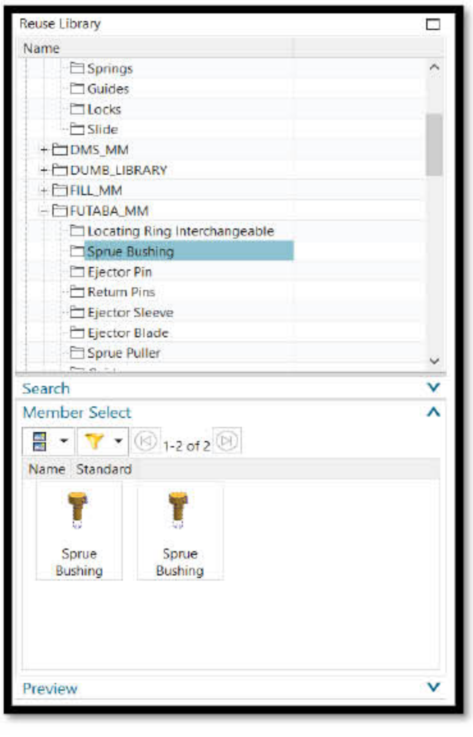
Flip Direction when you want to change the mounting direction

Remove Componcat when you want to delete details

Silver spray stem .

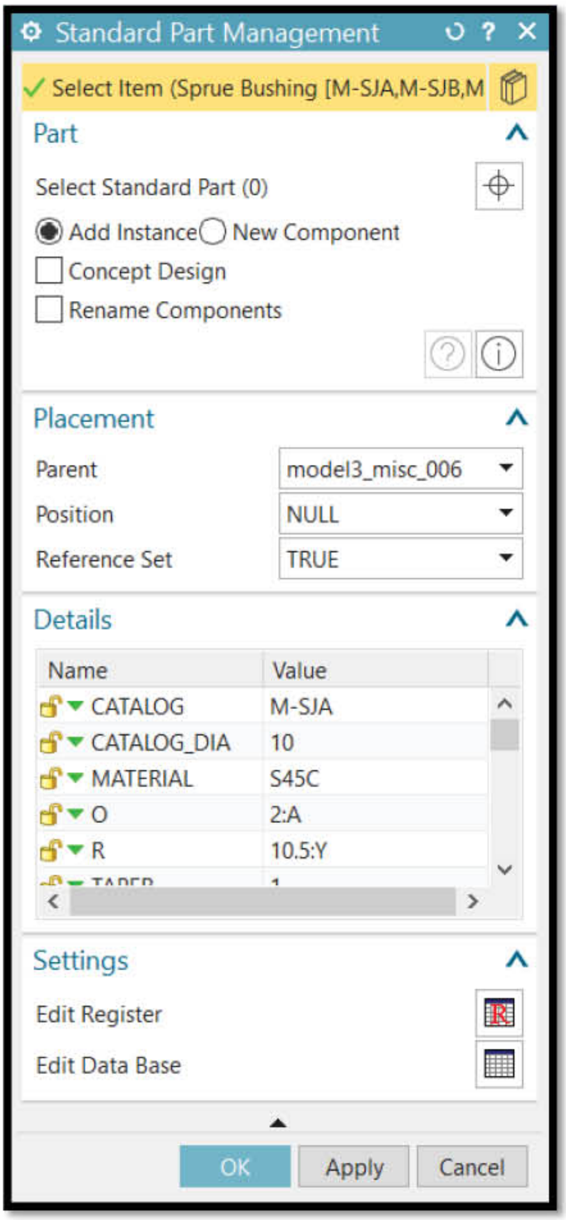
On the Mold Wizard toolbar, select Standard Part Library in the Main tab.





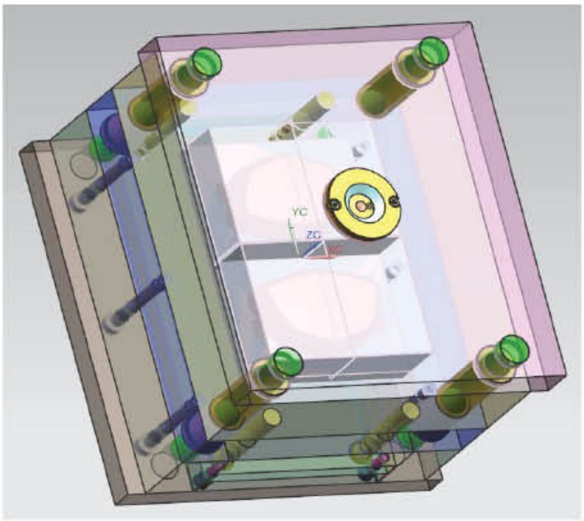
Select the **Rouse Library tab** > in **MW Standard Part Library** > Find the mold row > **FUTABA _MM** > Select **Sprue Bushing.**

Member Select section: select **Sprue Bushing [M-SJA].**



Standard Part Management dialog box , there is a **Details** box that contains the main parameters of the nozzle bush that need to be edited.

Click **OK**



Coat the mold after adding spray stem silver.

To edit the parameters of details in the selected library or change the location, there are the following ways.

Click on Standard Part Library in the Main tab.

Right-click on the detail, select Edit Tooling Component

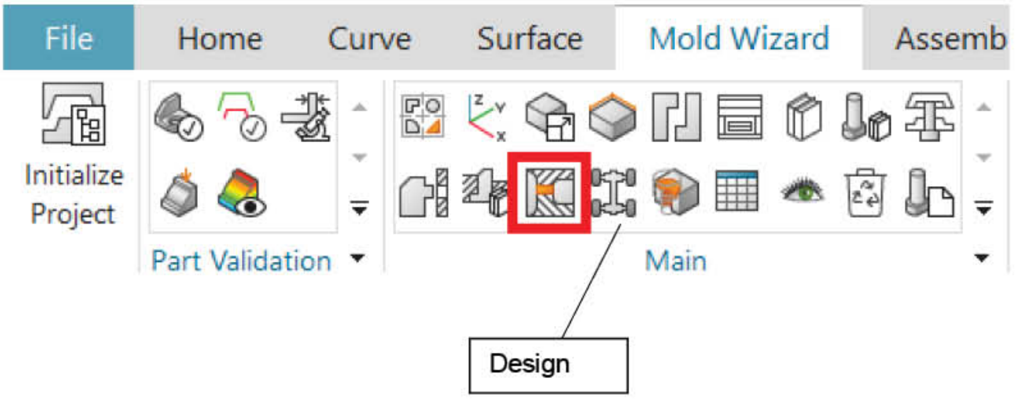
Reposition when you want to change the position of the nozzle bush

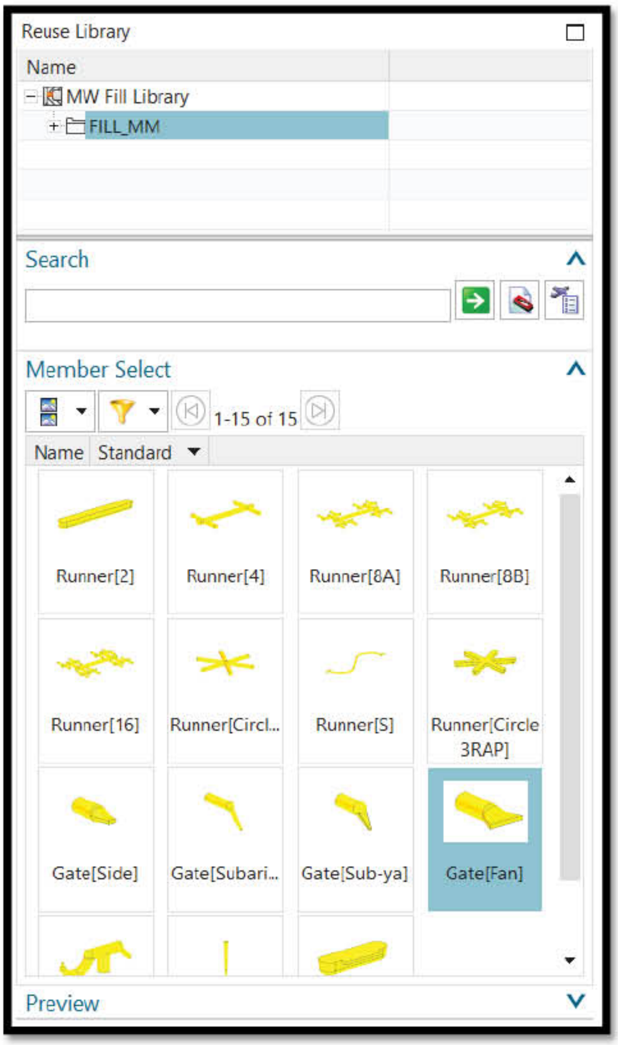
Flip Direction when you want to change the mounting direction

Remove Componcat when you want to delete details

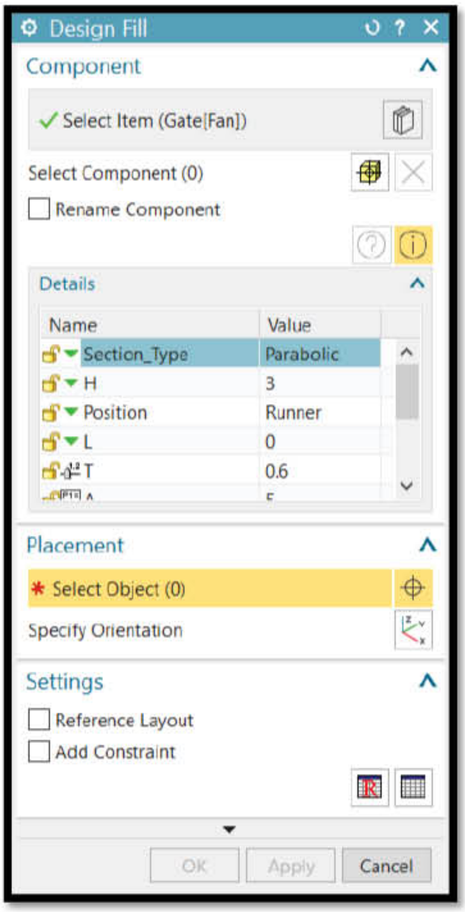
Spray Mouth.

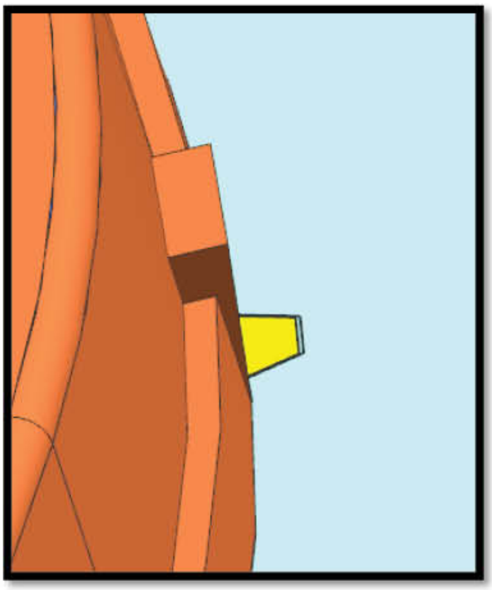
On the Mold Wizard toolbar, select Design Fill in the Main tab.

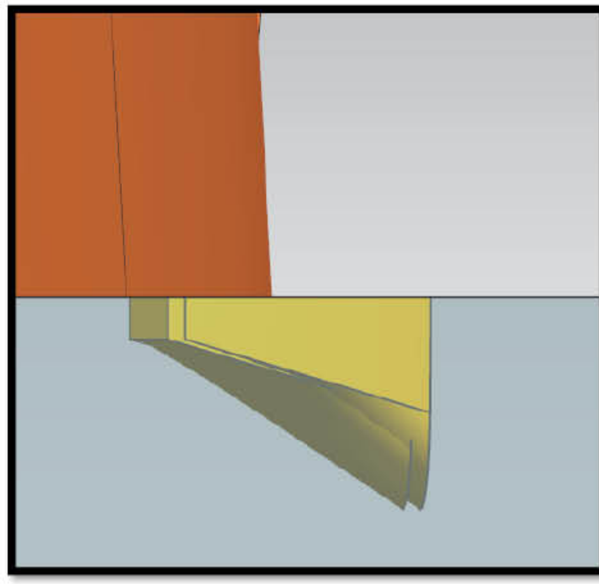




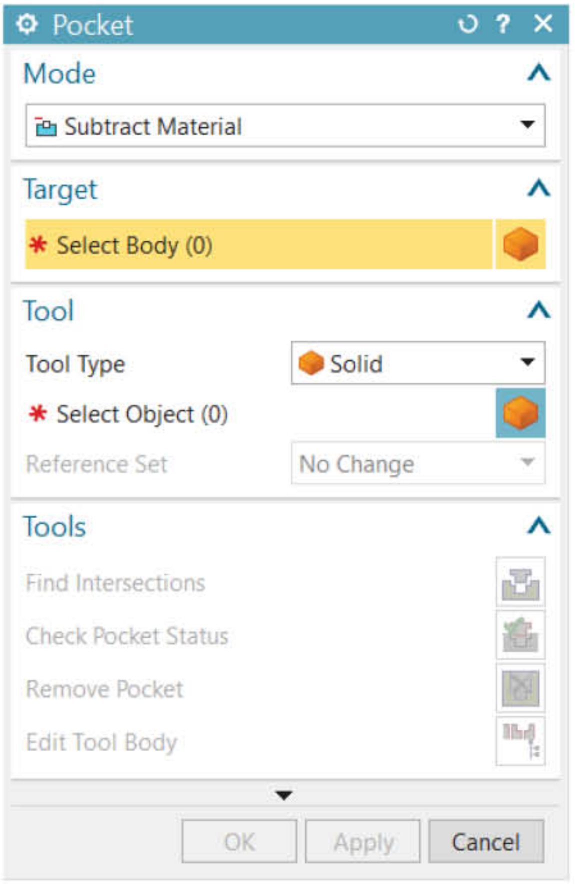
Member Select section: select the appropriate type of plastic input port.



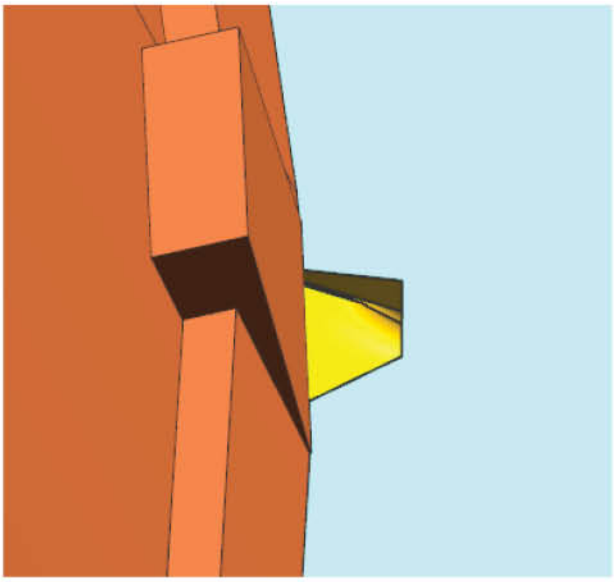




On the Mold Wizard toolbar, select Pocket in the Main tab.

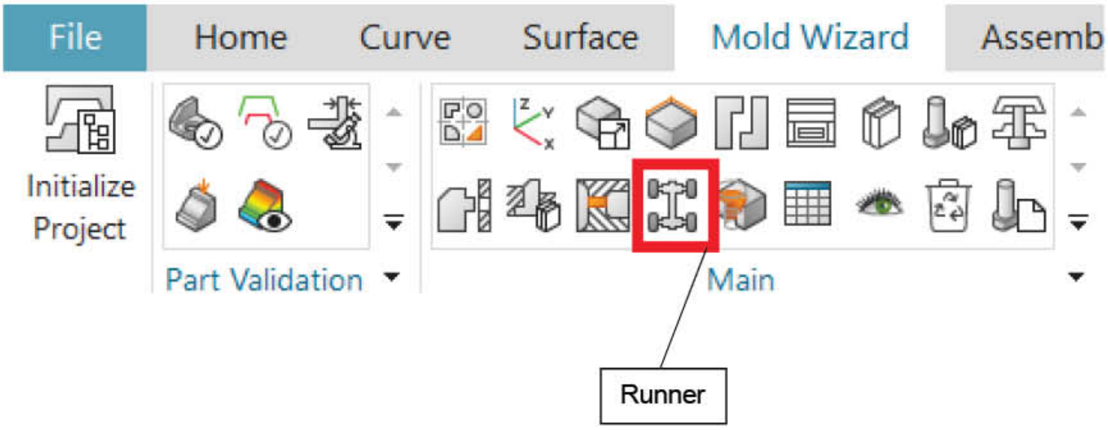


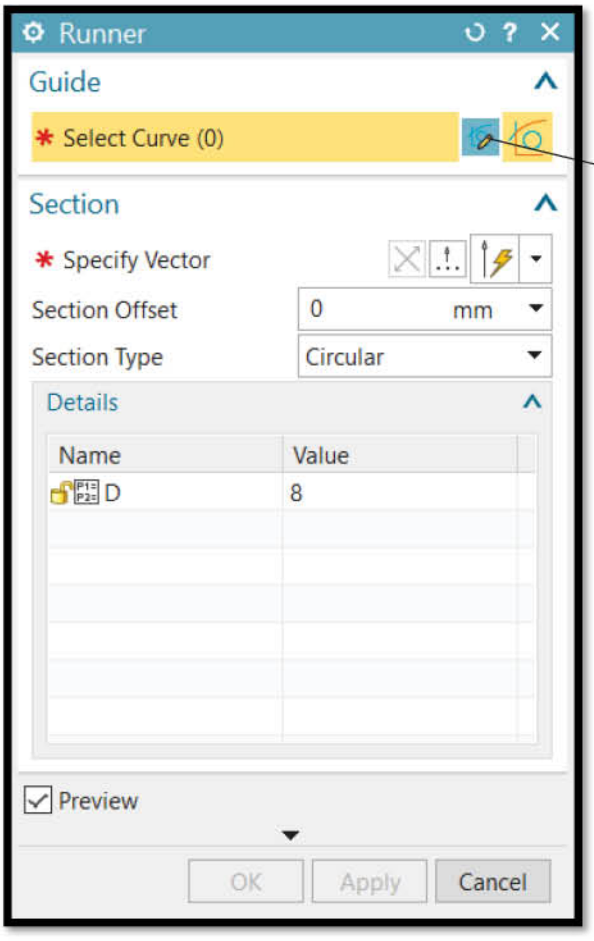
Click on the core plate in Select Body > click on the newly created plastic entrance to create a cavity > select Apply.



Plastic channel .

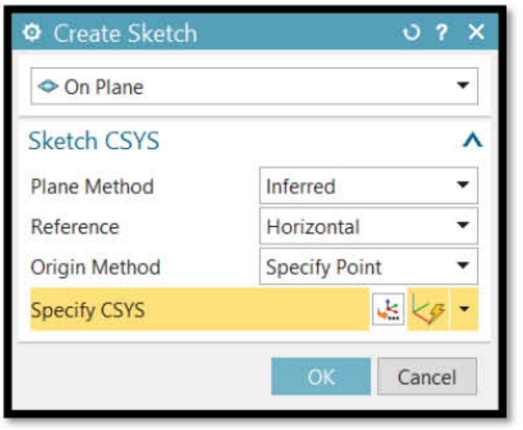
On the Mold Wizard toolbar, select Runner  in the Main tab.





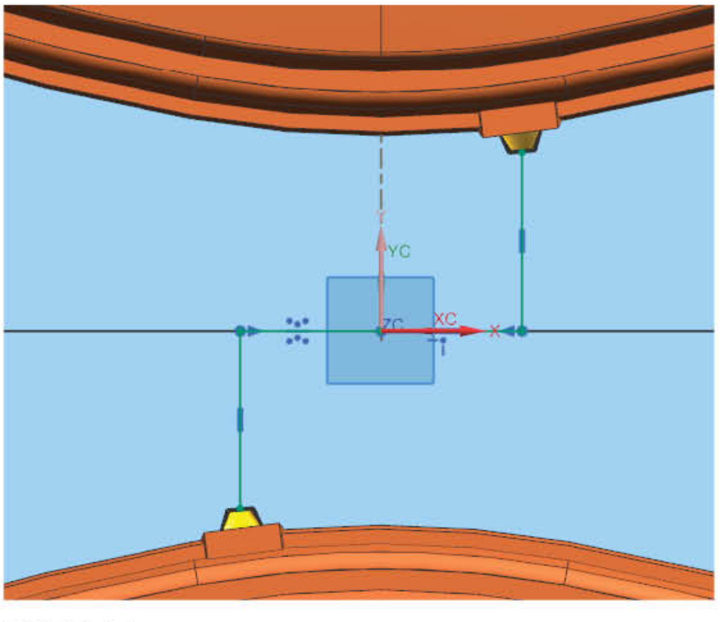
Click Sketch Section in the Runner dialog box . The Create Sketch dialog box appears.

Sketch

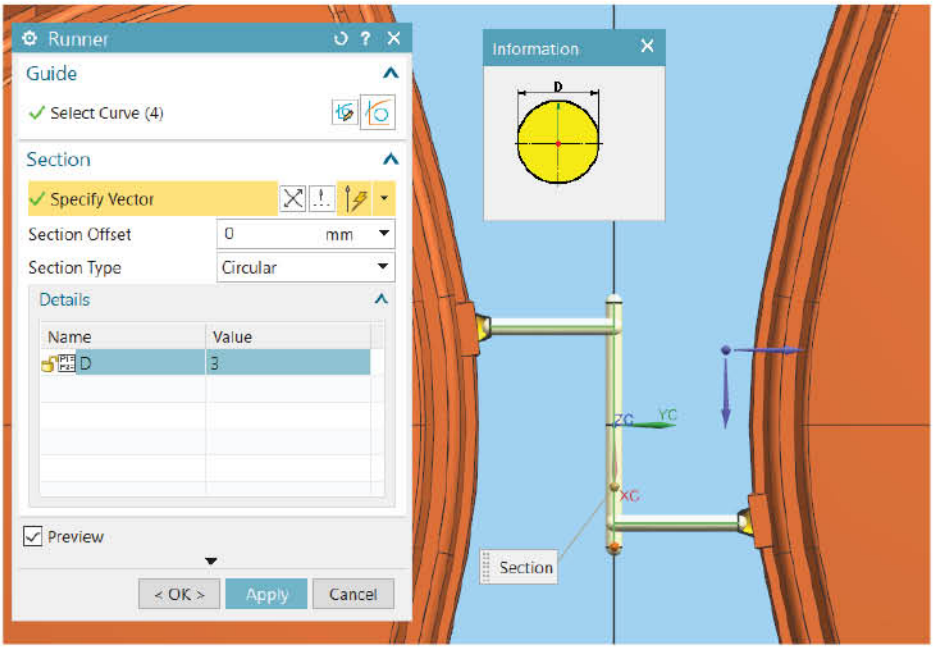


Click OK

Sketch the path of the channel.



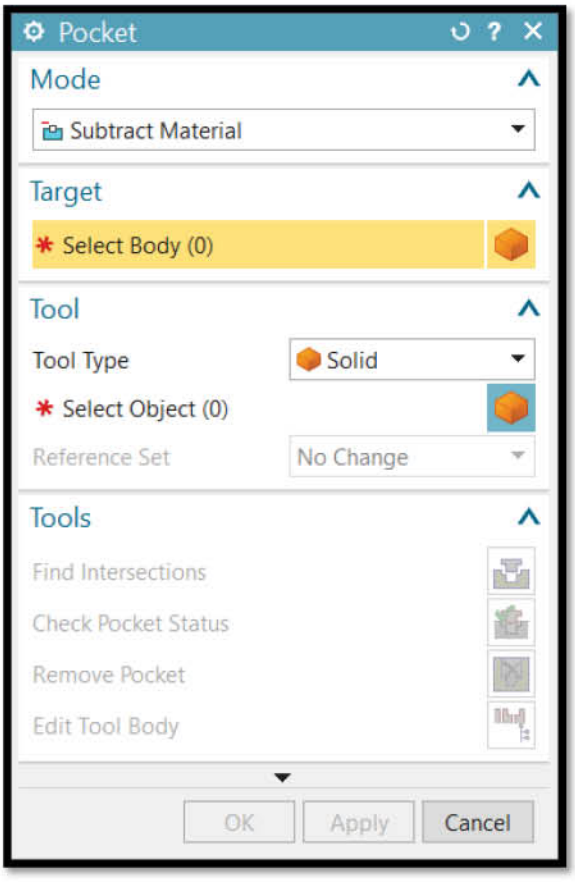
Click Finish



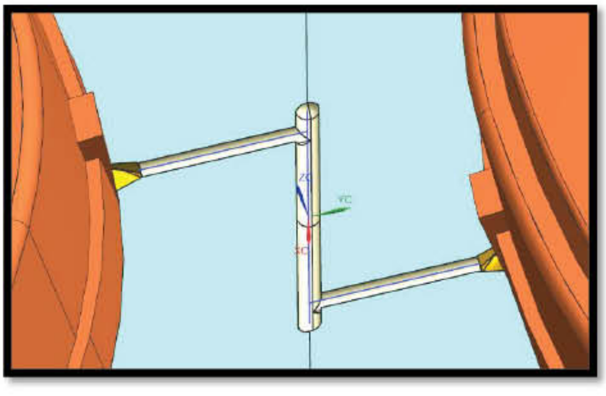
In the Details box, you can edit the channel diameter to suit the nozzle.

Click Apply.

On the Mold Wizard toolbar, select Pocket  in the Main tab.



Click on the Core sheet in the Select Body section > click on the newly created plastic channel to groove in the Select Object section > select Apply.

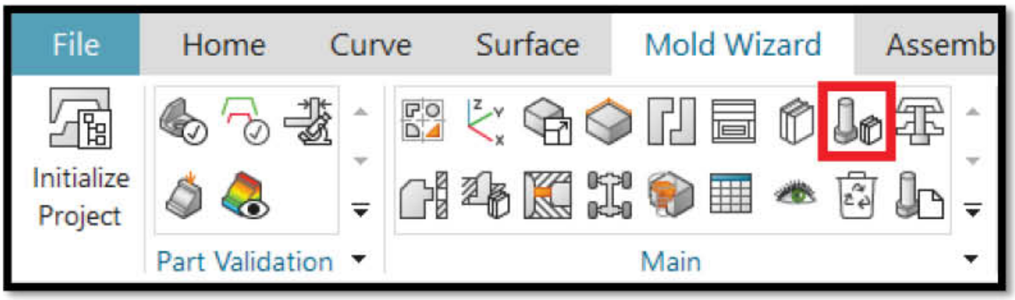


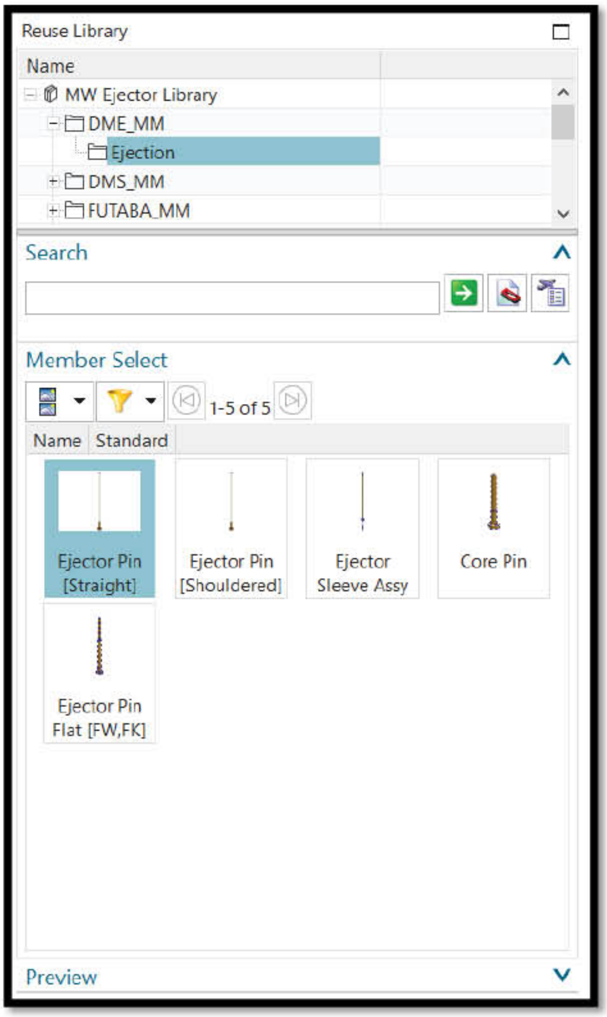
The spray system has been designed.

3.6.2. Ejection system design

To be able to arrange the propulsion system appropriately, we need to rely on the structure of the product.

On the Mold Wizard toolbar, select Design Ejection Pin  in the Main tab.

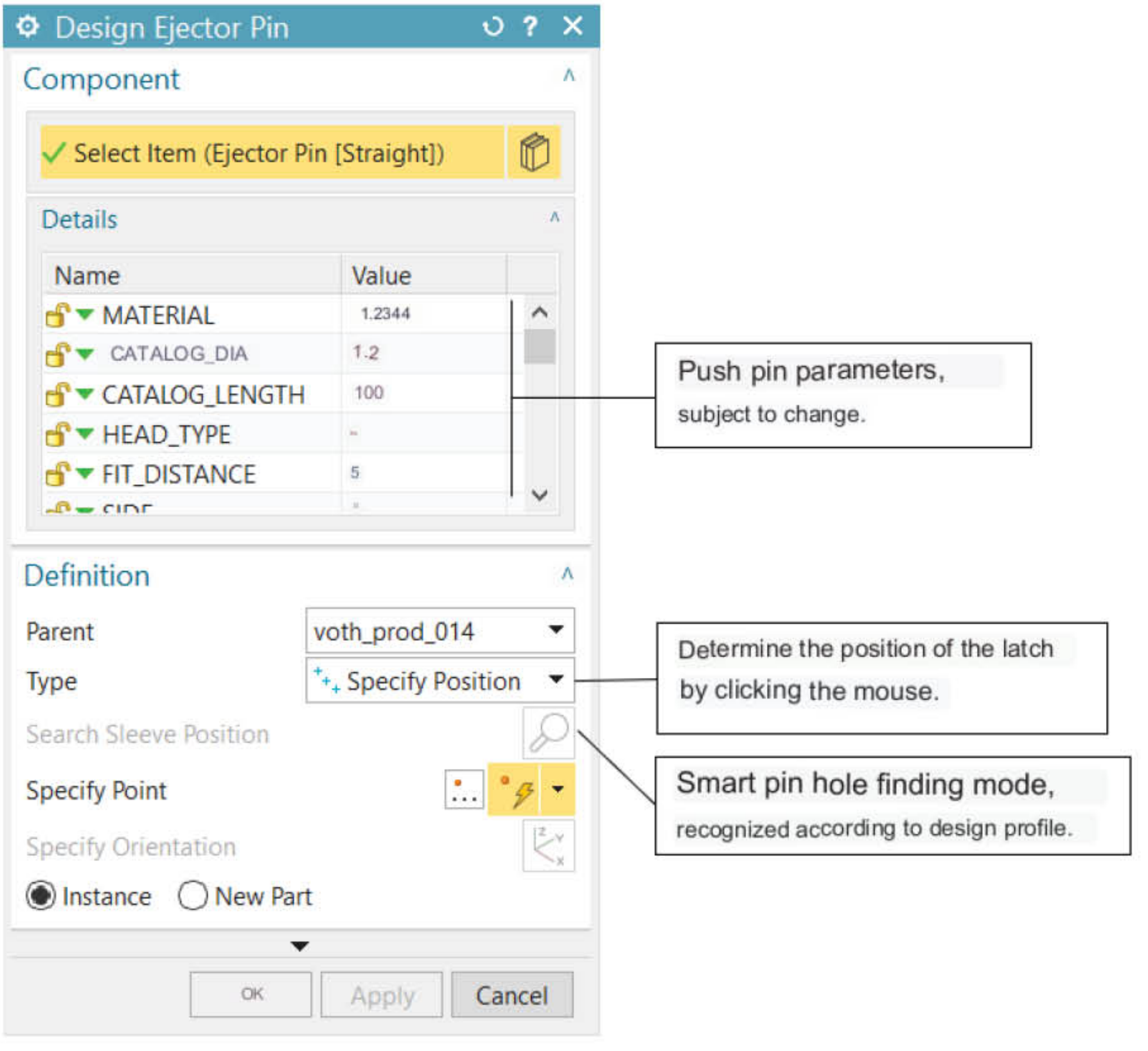


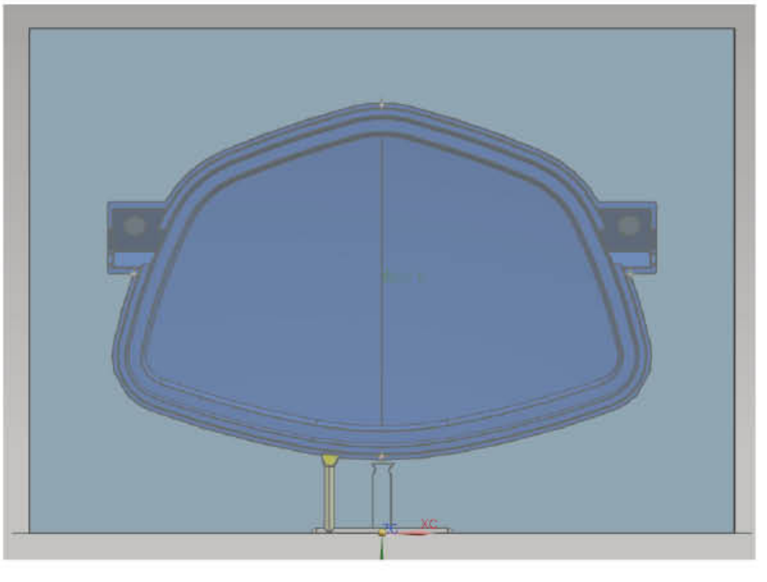


On the **Rouse Library** tab > in **MW Ejector Library** > **DME_MM** > Select **Ejection.**

Member Select selects **Ejector Pin** .

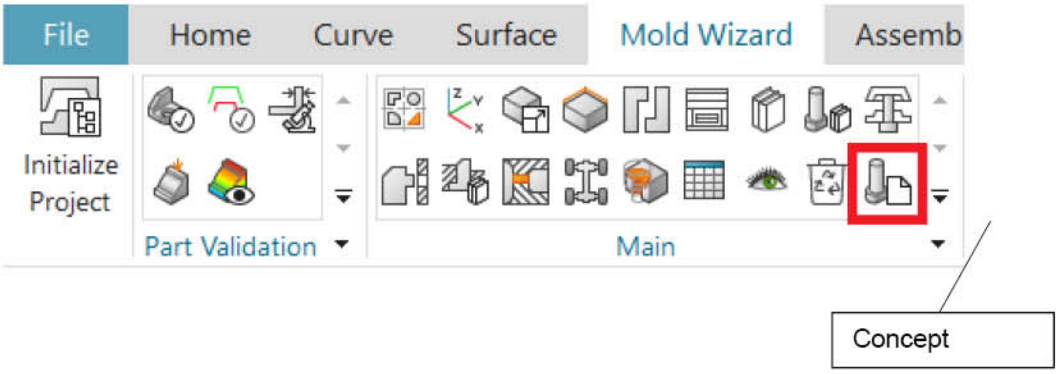
The Design Ejector Pin dialog box appears.



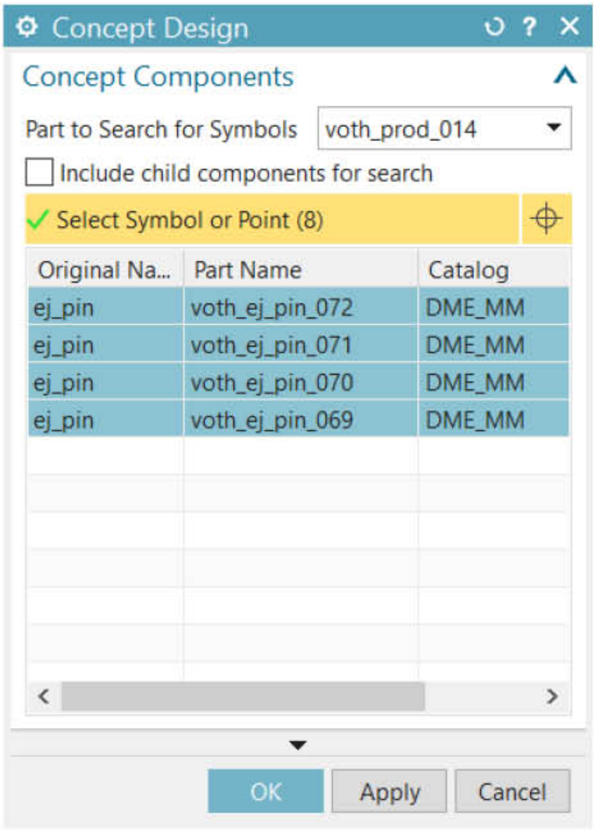


Click **Specify Point** > select the points you want to set the pin > Click **OK** .

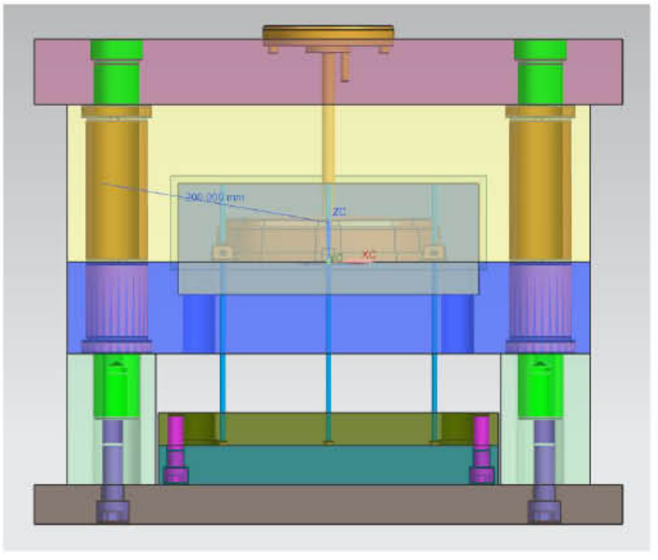
On the Mold Wizard toolbar, select Concept Design in the Main tab.



The Concept Design dialog box appears.

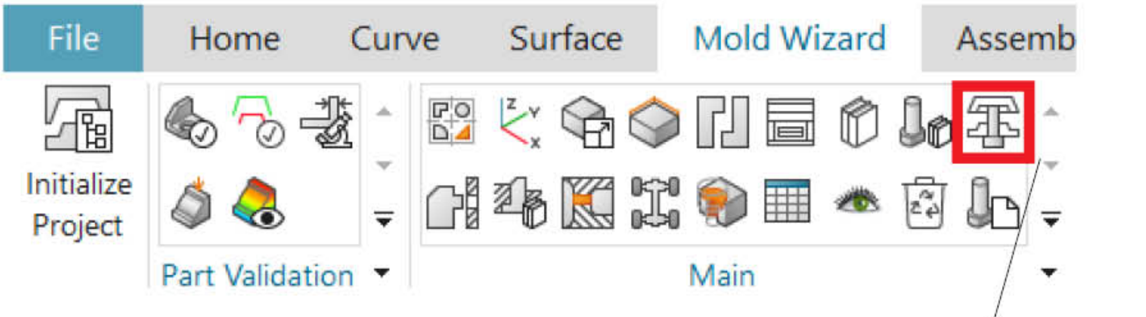


Press **CTRL** then select the lines below > click **OK** to create the handle.

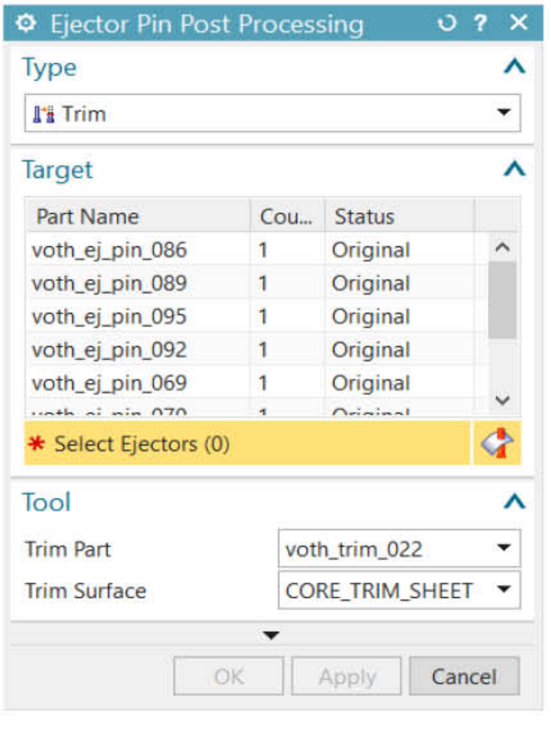


The push pins are longer than the product position > cut off the excess products of the pin.

On the Mold Wizard toolbar, select Ejector Pin Post Processing in the Main tab to cut the pin.



The Ejector Pin Post Processing dialog box appears.

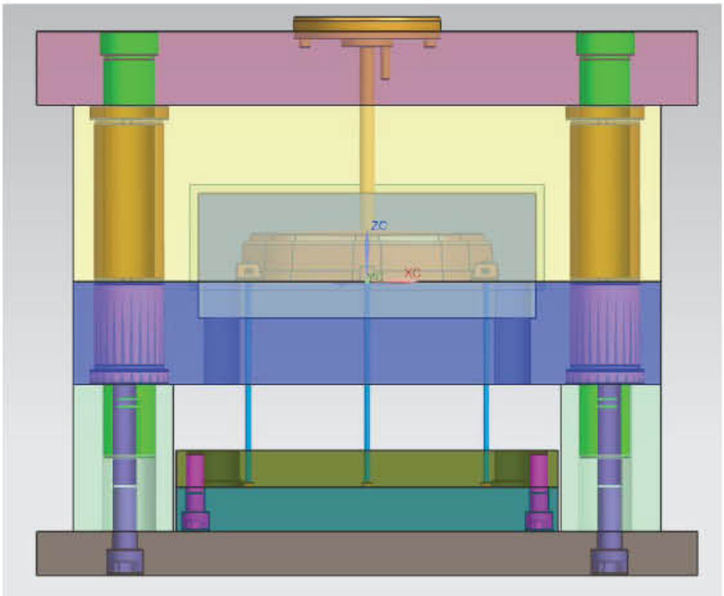


Press **CTRL** and then select the rows in the Target table.

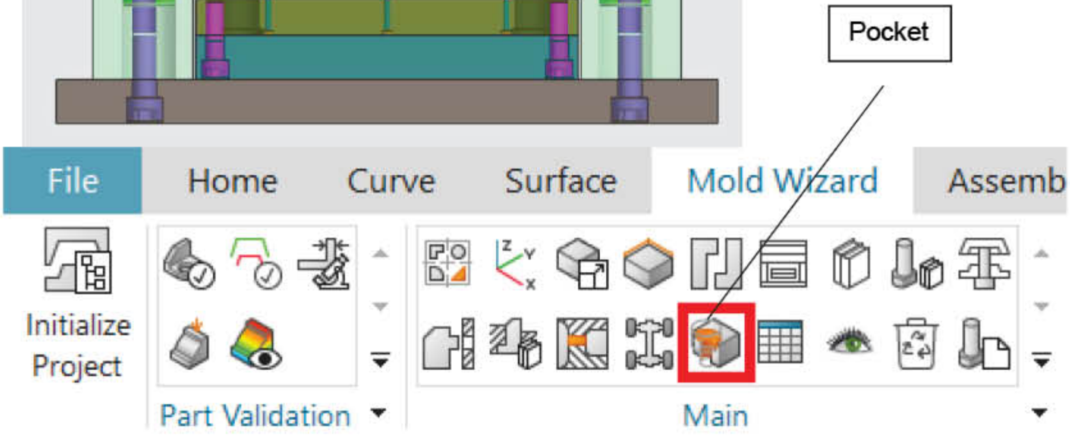
Trim Surface : select **CORE_TRIM_SHEET,** use the male mold surface to cut the pin.

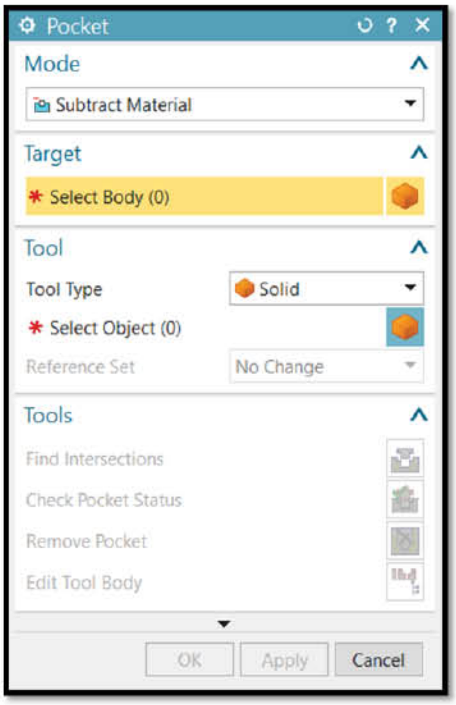
Click **OK** .

The mold has cut off all excess pins.



On the Mold Wizard toolbar, select Pocket  in the Main tab.



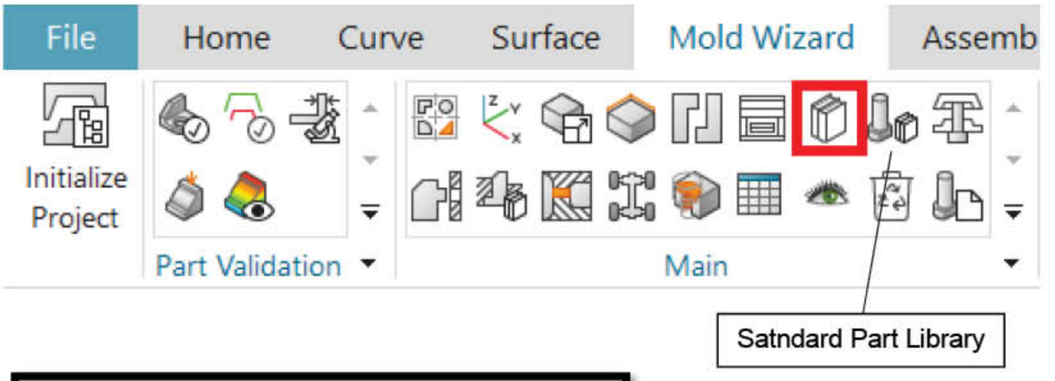


_Target: Main mold, main mold cover, return plate.
_Tool: are the pins created here.
_Click OK.

3. 6.3. Insert some sub details (Sub Insert).

The spring is mounted coaxially with the return pin to push the return plate back to its original position.

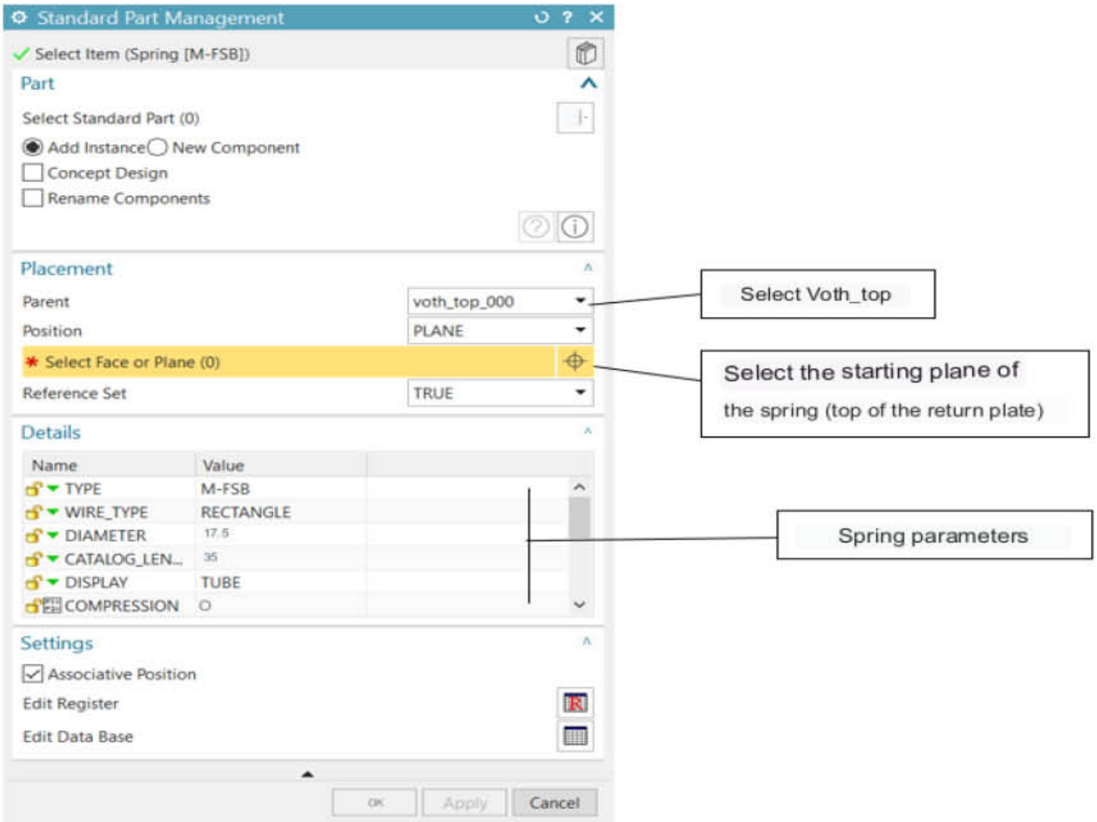
On the Mold Wizard toolbar, select Standard Part Library  in the Main tab.



Select the **Rouse Library tab** > in **MW Standard Part Library** > Find the mold row > **FUTABA _MM** > Select **Springs**

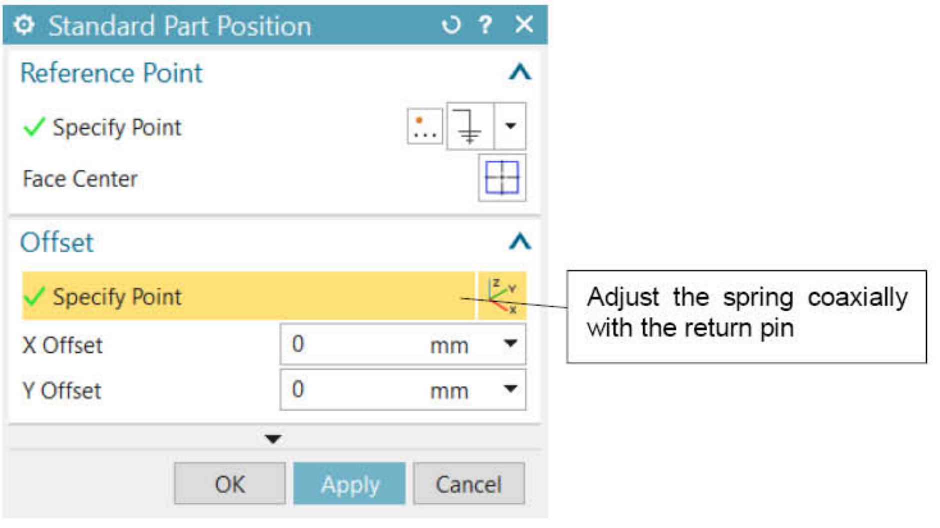
Member Select section : select **Springs [M-FSR]:** spring type

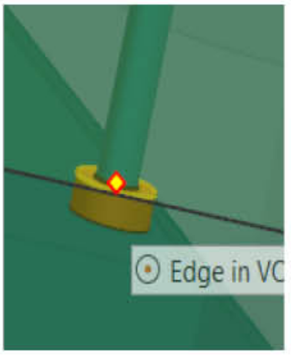
The Standard Part Management dialog box appears.



Click Apply

The Standard Part Position dialog box appears.



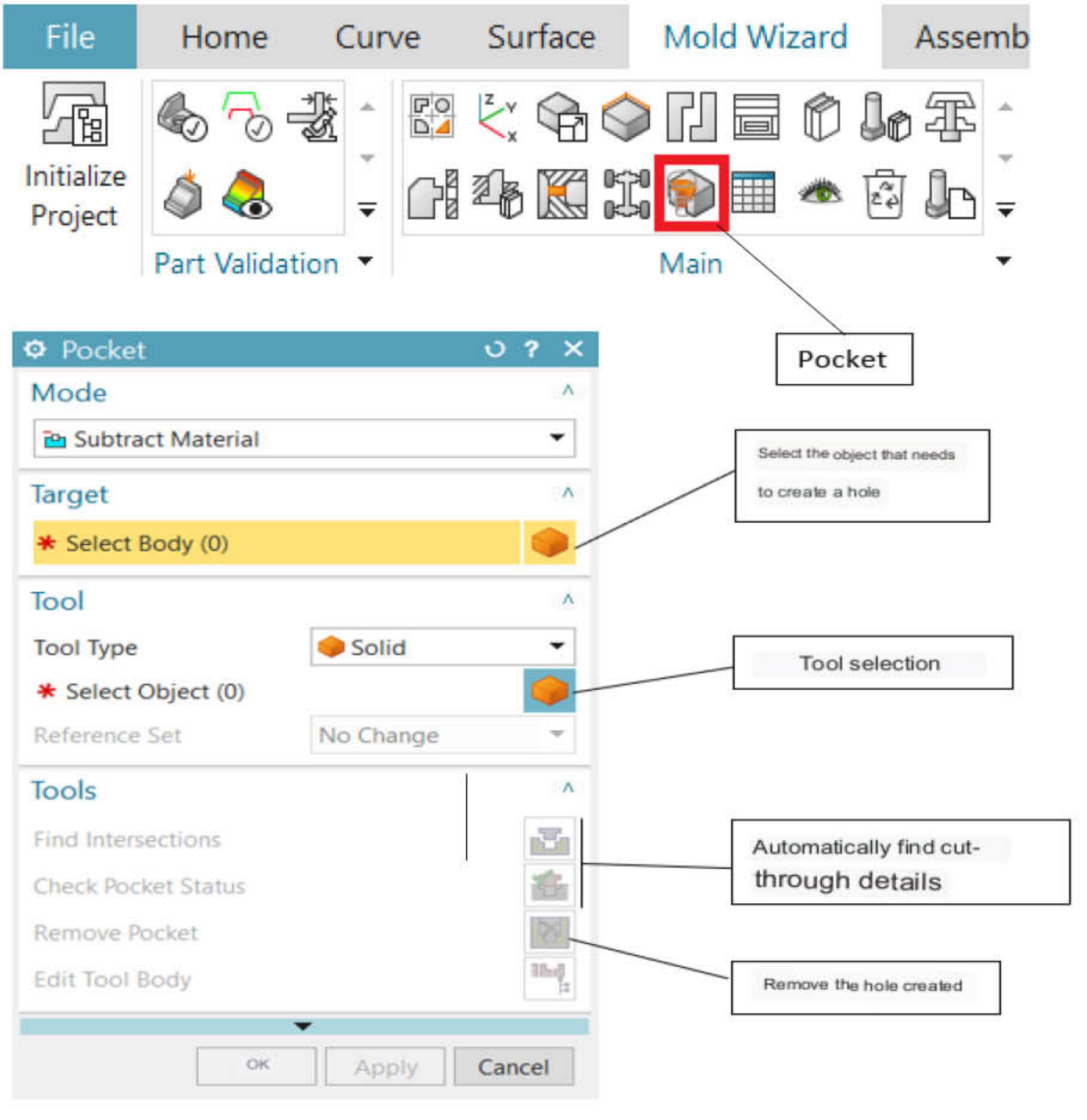


3.7. Complete design.

3.7.1. Create holes (Pockets).

The hole making stage is the final stage of the mold design process. Intersection locations between pipes and support plates or screw holes... need to be created. The Pocket tool helps create holes quickly with a few taps.

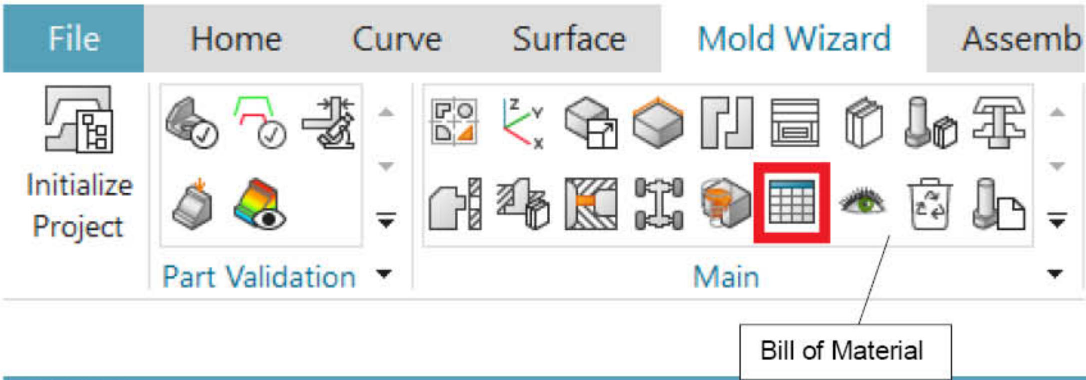
On the Mold Wizard toolbar, select Standard Part Library  in the Main tab.

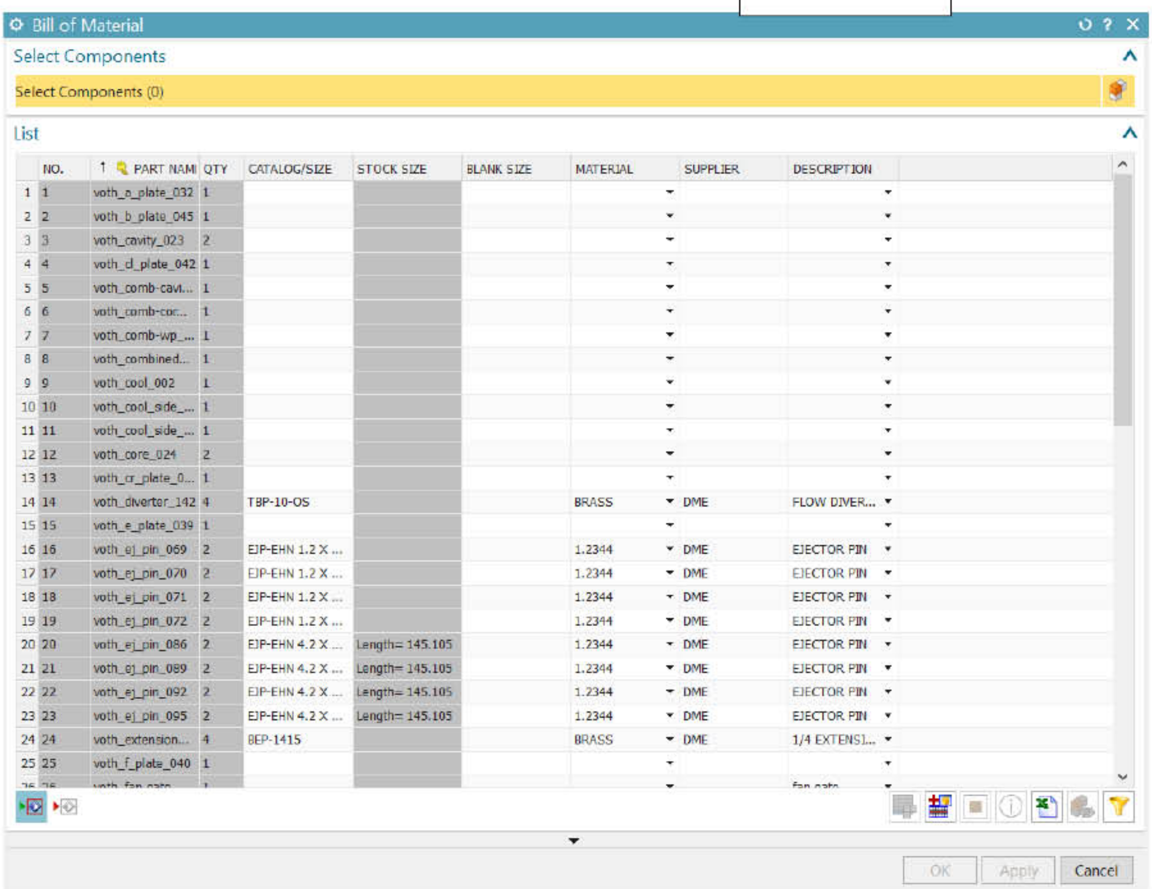


3.7.2. Create a bill of material (Bill of Material).

Molds are created with many details, systems, and sub-details. Each detail or system consists of different materials and sizes. Bill of Material allows creating a list of materials, dimensions, etc. so that users can optimally manage materials.

On the Mold Wizard toolbar, select Bill of Material  in the Main tab.





Materials table

UNIT 4: DESIGN OF PRESS DIES WITH NX PROGRESSIVE DIE WIZARD: STREAMLINING WORKFLOW

Detailed Content:

Objective: By the end of this training unit, the trainees will be able to

- Present the concept of sheet presswork technology and classify the types of this technology

- Analyze the functions of the components that make up a common stamping die

- Explain the use of support commands of the PDW tool

- Use the commands of the PDW tool to design a complete set of progressive die

- Solve application assignments with the help of the PDW tool.

Content:

4.1. Overview of press dies

4.1.1. Sheet stamping technology

Sheet forming technology is a part of pressure metal processing technology to deform sheet metal to obtain details of the desired shape and size. This is a type of technology that is being widely applied in many different industries, especially in the fields of electrical engineering, electronics, automobile manufacturing industry, aviation industry, and manufacturing industry, consumer goods manufacturing industry, defense industry, food, chemicals, healthcare...

a. Concept of sheet stamping technology: Sheet stamping is a part of a technological process that includes many different technological operations to deform metal (tape or strip blanks) to obtain details of the required shape and size, with negligible change in the thickness S of the material and no scrap in the form of chips.

Sheet stamping is often done with the workpiece in a cold state, so it is also called cold stamping when the thickness S of the workpiece is small (usually $S \leq 4$mm) or may require stamping in a hot state when the thickness of the material is large. Products of die technology may have to go through many different operations, which can be performed on the same machine or multiple machines. For example: cutting shapes, punching holes, stamping, bending...

b. Classification of sheet stamping technology: According to the deformation characteristics of the sheet stamping process, people are divided into 2 main groups:

- Material cutting operations group: when shaping details, operations in this group often have to deform and destroy the material.

- Material plastic deformation operations group: detailed shaping is based on the plastic deformation of the material and in most cases there is metal redistribution. In most cases, the workpiece material thickness barely changes or changes slightly but unintentionally.

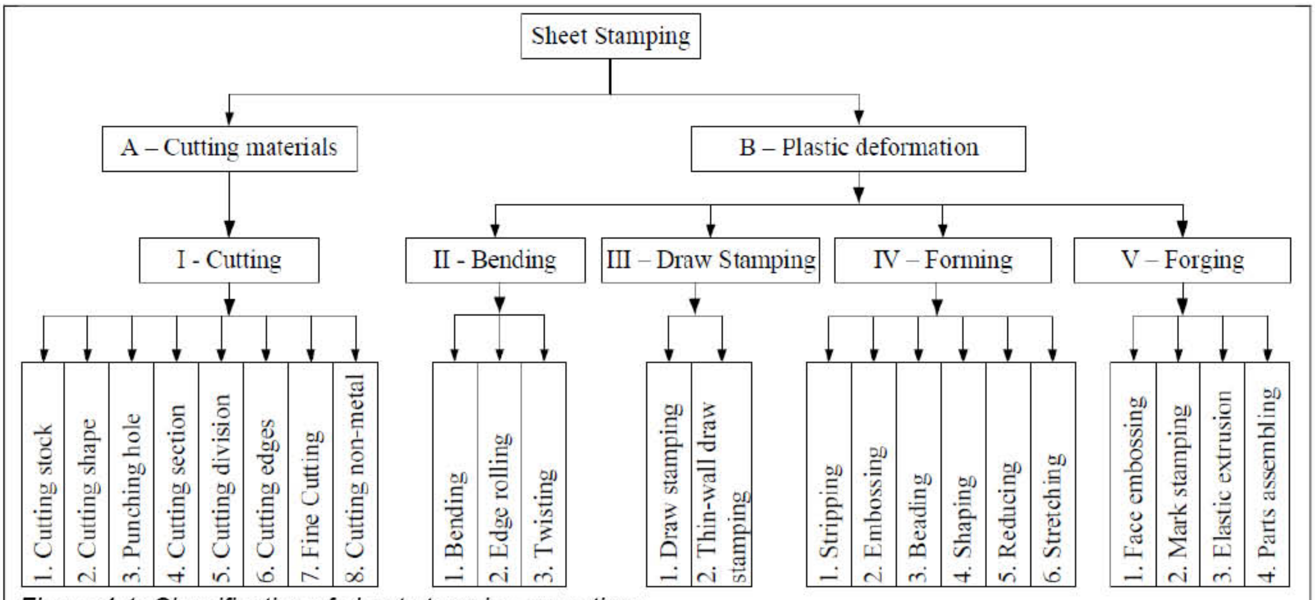


Figure 4.1: Classification of sheet stamping operations

In the die process, people can stamp each operation separately or combine two or more operations on the same die. When stamping each operation separately, the productivity is generally low, the manufacturing time is long, it will have to be disassembled many times, and many workers are used.

b1. Advantages of die technology:

- Can perform complex tasks with simple movements of equipment and dies

- Can produce complex parts that are impossible or very difficult with other mechanical processing methods.

- The precision of the die parts is relatively high, ensuring good fit, no need for mechanical processing

- The structure of the die part is sturdy, durable and lightweight, the level of material wastage is not large, saving materials

- Saves raw materials, facilitates mechanical processes and automation, thus high labor productivity and lower product prices.

- The operation process is simple, so there is no need for high-level workers, reducing salary training costs

- Production form suitable for large batch and mass production

- Take advantage of raw materials, high material utilization coefficient

- Stamping dies can only process metal materials but can also process non-metallic materials and flexible materials.

b2. Disadvantages of die technology:

- Relatively large initial investment (dies and equipment) so only suitable for large batch and mass production

- Requires a team of highly qualified and skilled engineers

- Complicated technological calculations.

c. Stamping sheet metal on a progressive die

c1. Concepts, characteristics and applications of continuous stamping dies

Continuous stamping is a unique technology in which a progressive die integrates multiple die operations on the same press machine journey. Designing and manufacturing continuous stamping dies is a complex and meticulous process, requiring the designer to master all technological processes in die. At the same time, master current advanced manufacturing technology.

The application of modern simulation design and machining technologies has increasingly increased the accuracy, speed, and efficiency of continuous stamping dies to meet the needs of producing large series of parts with high interchangeability, low cost for industries.

Progressive stamping die (referred to as progressive die) is a die consisting of many pairs of punches and dies arranged on the same die base to perform different sheet stamping operations (stamping, bending, embossing, cutting. ..) after one journey of the press. Each working position (a pair of punch and die) performs one or more separate technological steps, with support of the automatic bar feeding structure, the workpiece is continuously and sequentially moved through positions to complete the part to be manufactured. The product is formed after the final stamping in the cavity of the die. Each stamping will produce a product. Each journey of the sliding head will perform many different technological operations. The sequential execution of operations makes continuous dies highly automated.

Always accompanying the progressive die is the automatic bar feeding system. It plays a decisive role in the automation ability of each factory. Progressive stamping requires absolute precision in moving the workpiece to produce high quality products.

Progressive die s can be classified as follows:

- The die continuously performs stamping operations
- The die continuously performs bending operations
- The die continuously presses and cuts
- Progressive stamping includes many different operations (bending, drawing, punching, cutting...)

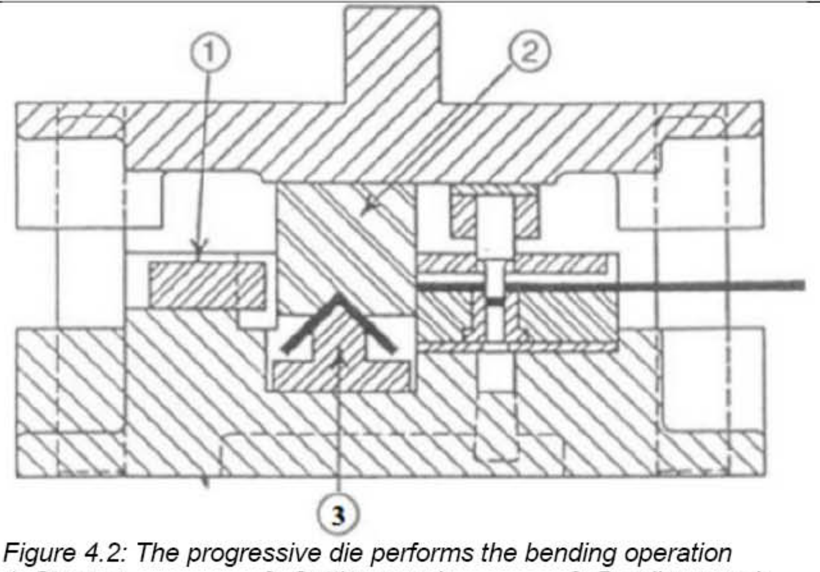


Figure 4.2: The progressive die performs the bending operation
1. Stopper 2. Cutting punch 3. Bending punch

Progressive die application selection criteria:

- Manufacturing parts need to go through many different sheet stamping operations
- High required production capacity (mass and large batch production)
- Products require high uniformity and accuracy

- Machinery and equipment capable of meeting high requirements: high-precision stamping machines, unrolling equipment, billet straightening and precise automatic bar feeding system.

Around the world, stamping dies are continuously applied in many manufacturing industries: electricity, electronics, aircraft, cars, mechanical parts, household appliances... In Vietnam, this technology is initially being applied to production in the field of processing details in motorbikes, electric motors...

The reason why continuous stamping dies are so widely used is because this type of die has many outstanding advantages compared to other types of technology:

- Smooth metal organization, high product mechanical properties

- The gloss and precision are higher than parts made by other stamping methods because the stamping process occurs continuously

- Save time and minimize errors by not wasting time setting and setting errors

- Easy to mechanize and automate, so productivity is high and costs are low

- Process complex parts efficiently and effectively

In the future, the application of continuous sheet stamping technology will continue to grow strongly because of the great efficiency and benefits it brings to industrial manufacturing industries not only in Vietnam but around the world.

c2. Basis of technological processes on progressive die

Stamping operations in continuous stamping technology can be divided into 2 groups:

- Separation operations: based on the principle of material destruction to separate one part from another (division cutting, punching cutting, injection cutting...)

- Shaping operations: based on the plastic deformation process of metal (bending, stamping, drawing, chipping...)

c3. Cutting operations

Cutting is a chipless machining operation that separates part or all of a metal from a sheet of material. The cutting method is arranged according to the shape of the blade. The material is cut by two cutting edges that join together at an angle of the β knife.

When the punch contacts the workpiece, the cutting edge presses against the material until the stress exceeds the allowable shear stress value. At the cutting edge of the die, cracks form in the material, the cracks grow into the plate and separate the material. In the cutting die, the punch plays the role of the moving upper blade while the die plays the role of the non-moving lower blade of the sheet cutting machine.

The die cutting process is carried out on a crankshaft press or a hydraulic press.

Sectioning is the operation of completely separating material along a curve whose two ends do not meet.

Extraction cutting is the operation of cutting a part of material along a curve whose two ends do not meet.

Shape cutting is the operation of separating a part of metal along a closed contour, the separated metal part is the detail.

Hole punching is the process of cutting material along a closed contour to form a smooth hole in the part or plate. The cut material is scrap.

Edge cutting is the operation of cutting the edge rim when stamping block parts on an open die.

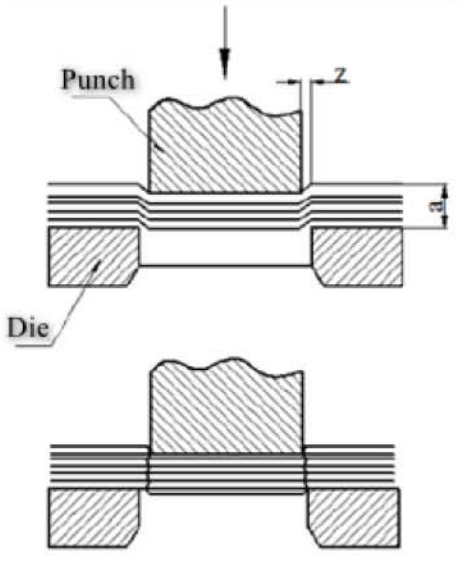


Figure 4.3: Stamping and cutting process

c4. Shaping operations

Draw stamping is an operation to transform flat or hollow blanks to create hollow parts of the required shape and size. There are two methods of stamping as follows:

Draw stamping without thinning: unintentional stamping reduces the thickness of the material

Draw stamping with thinning: stroke stamping intentionally reduces the thickness of the workpiece (mainly the thickness of the part wall). Most cases of debossing with thinning are performed with workpieces that have been first deformed without thinning from flat workpieces.

During the non-thin stamping process, the rim edge of the workpiece may not be pulled all the way into the die, causing wrinkles at the rim. To prevent wrinkles, people often use a material blocking plate to press the rim of the workpiece to the die surface.

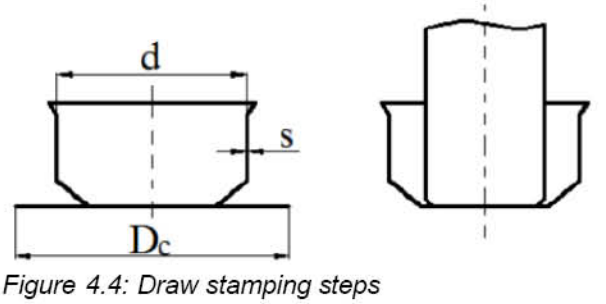


Figure 4.4: Draw stamping steps
a) Draw stamping for the first time *b) Draw stamping for the next times*

Bending is a method of turning a straight workpiece into a curved or folded workpiece at a certain angle.

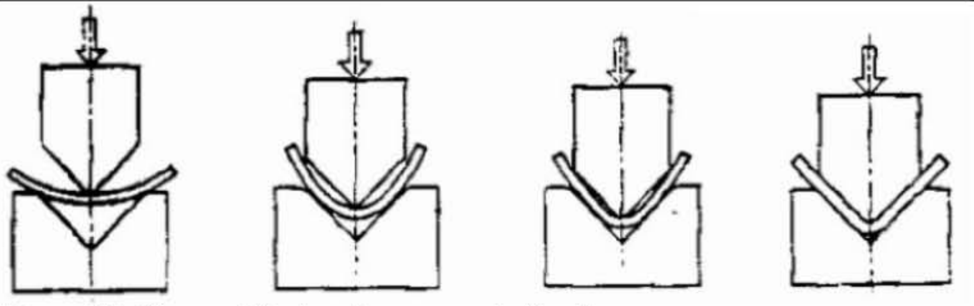


Figure 4.5: Stages of the bending process in the die

4.1.2. Structure of a progressive stamping die

a. Structure of a progressive stamping die

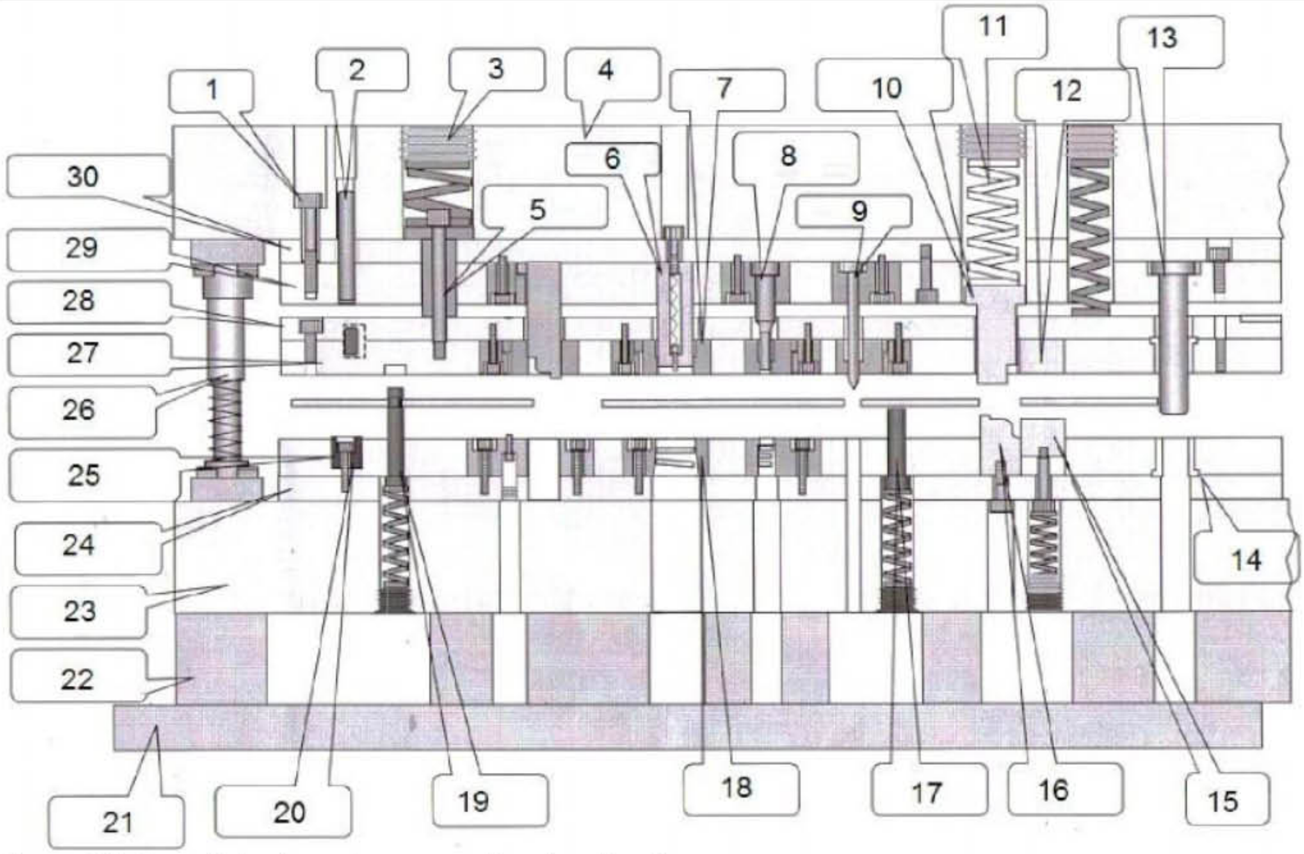


Figure 4.6: Overall structure of a progressive stamping die

1. Bolt	2. Locator pin	3. Screw
4. Upper die base	5. Suspension bolt	6. Punch
7. Local wiper	8. Shoulder punch	9. Centering punch
10. Upper pusher	11. Spring	12. Bending die
13. Auxiliary guide pin	14. Auxiliary guide bush	15. Workpiece clamp (Pad)
16. Bending punch	17. Workpiece push pin	18. Punching die
19. Workpiece lifting gauge	20. Balancer	21. Lower die base
22, 23. Bedding plate	24. Die bedding plate	25. Die shell
26. Guide pillar, guide bush	27. Stripper plate	28. Workpiece stripper plate
29. Punch shell	30. Punch pad	

b. Uses of main components of progressive stamping dies

A set of progressive stamping die includes many parts and components depending on the complexity of the product to be stamped as well as the cost of making the die. In addition, in the stamping dies we will get acquainted with standard parts from famous brands such as Misumi, Daton...

Important and indispensable parts in the design of sheet metal stamping dies are: Punch and die assembly, guide assembly, springs, billet ejection pins, basic metal plates of the die... These assemblies are also same as single stamping die.

The difference between a progressive die and a single die is the automatic blank feeding system into the die, the blank lifting system during the blank moving process, balancer, the centering punches and the blank step, stop block, height adjustment assembly for storage.

b1. Centering punches and guide pins to position the workpiece

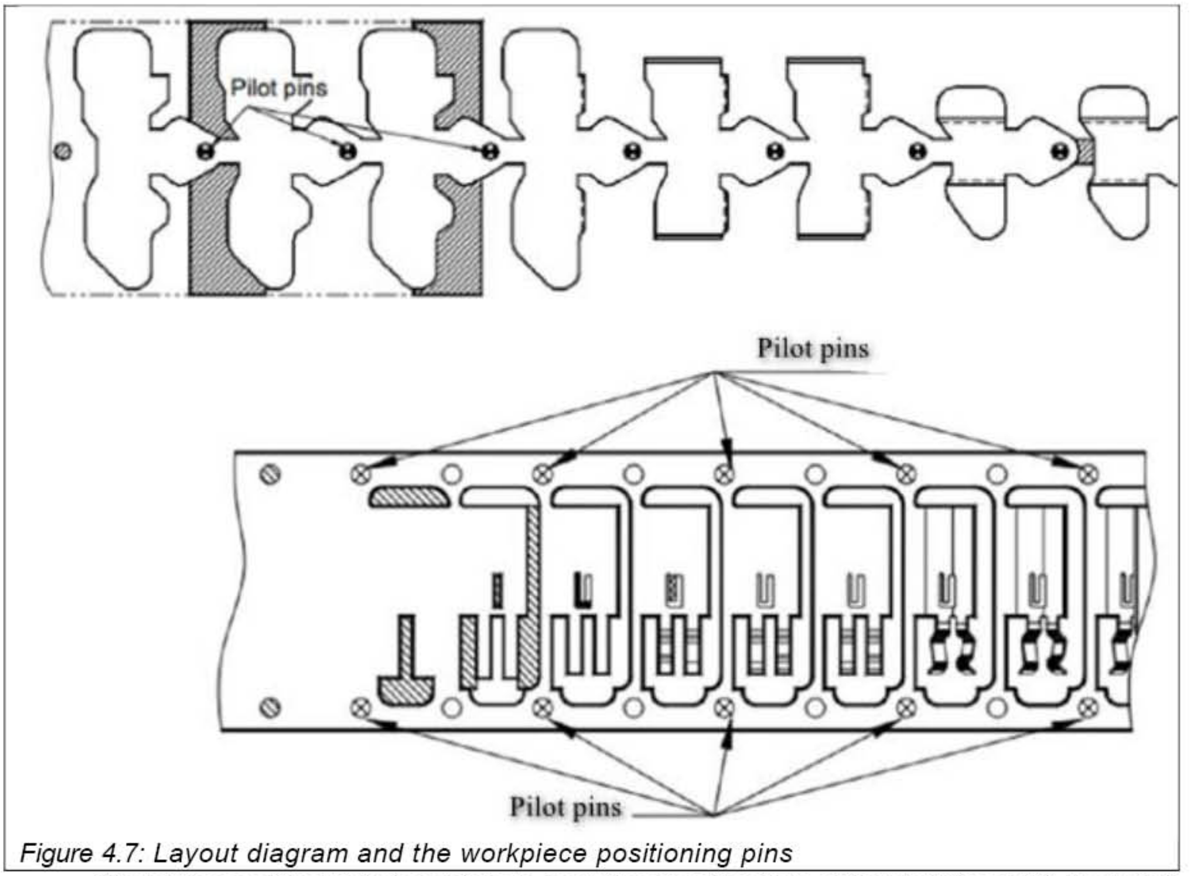
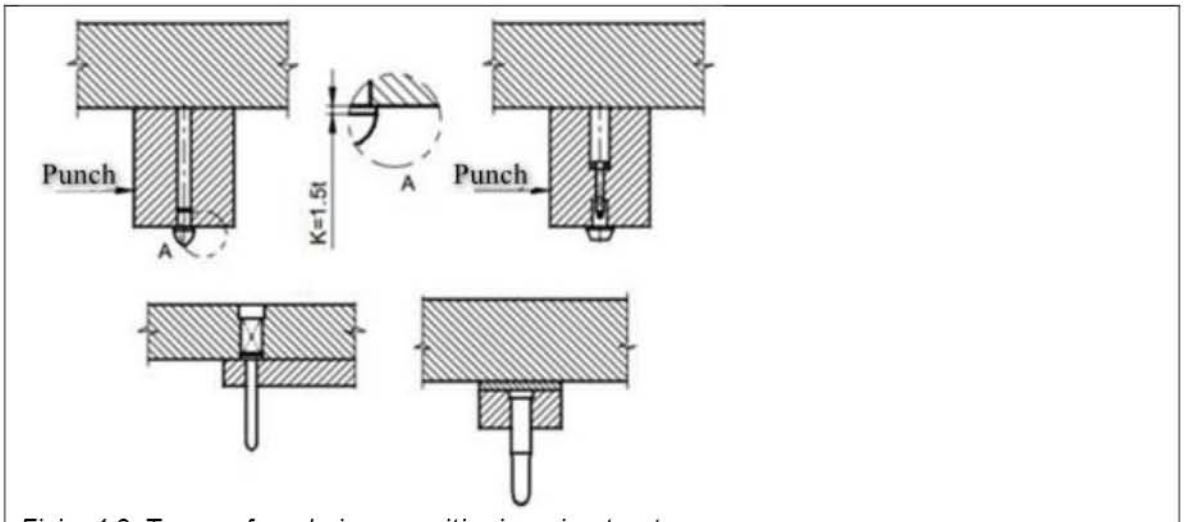


Figure 4.7: Layout diagram and the workpiece positioning pins

The purpose of the workpiece pin is to keep the exact position of the workpiece with the punch and die during the workpiece transportation process. Each workpiece step is arranged with one or two workpiece pins depending on the technological layout. The distance and position between the punches and dies participating in the machining process need to be arranged appropriately. Therefore, we need to add waiting steps of the workpiece to increase the distance between the punch and the working die live the step adjacent near.



Figire 4.8: Types of workpiece positioning pin structures

To create holes and slots for the centering punch to position the workpiece, step-slot punches are needed. The structural types of centering punches can be round punches or step-shaped punches.

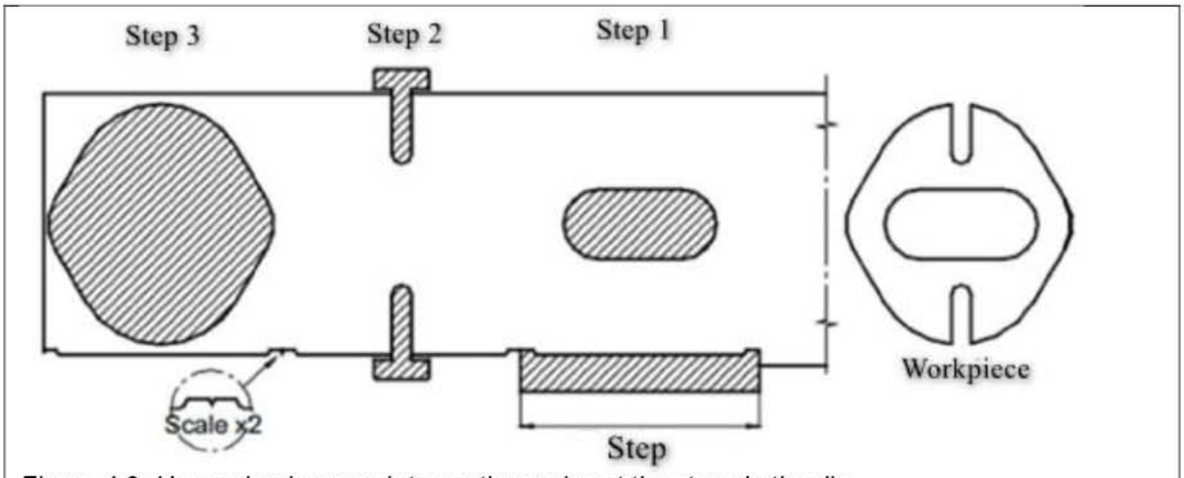


Figure 4.9: Use a shaping punch to continuously set the steps in the die

b2. Balancer

Balancer ensures that the material thickness is within the elastic limit and does not overcompress and turn into plastic deformation. After deformation, the thickness of the workpiece will be reduced.

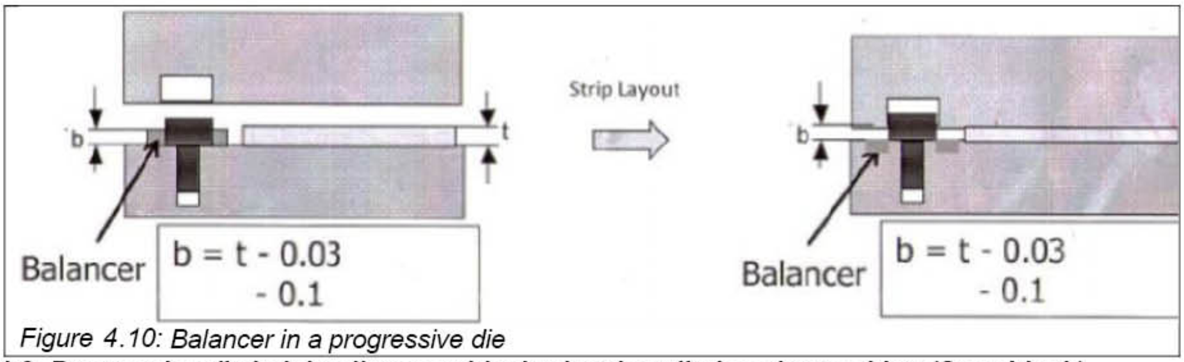


Figure 4.10: Balancer in a progressive die

b3. Progressive die height alignment block when installed on the machine (Stop block)

Unlike single dies, when putting the die on the machine, the height must be adjusted to ensure product retrieval. The progressive die has a base to ensure the tight height of the die. This part is made last after aligning the die on the machine for the first time. The next time the die is put on the machine, just make sure the stop block touches each other.

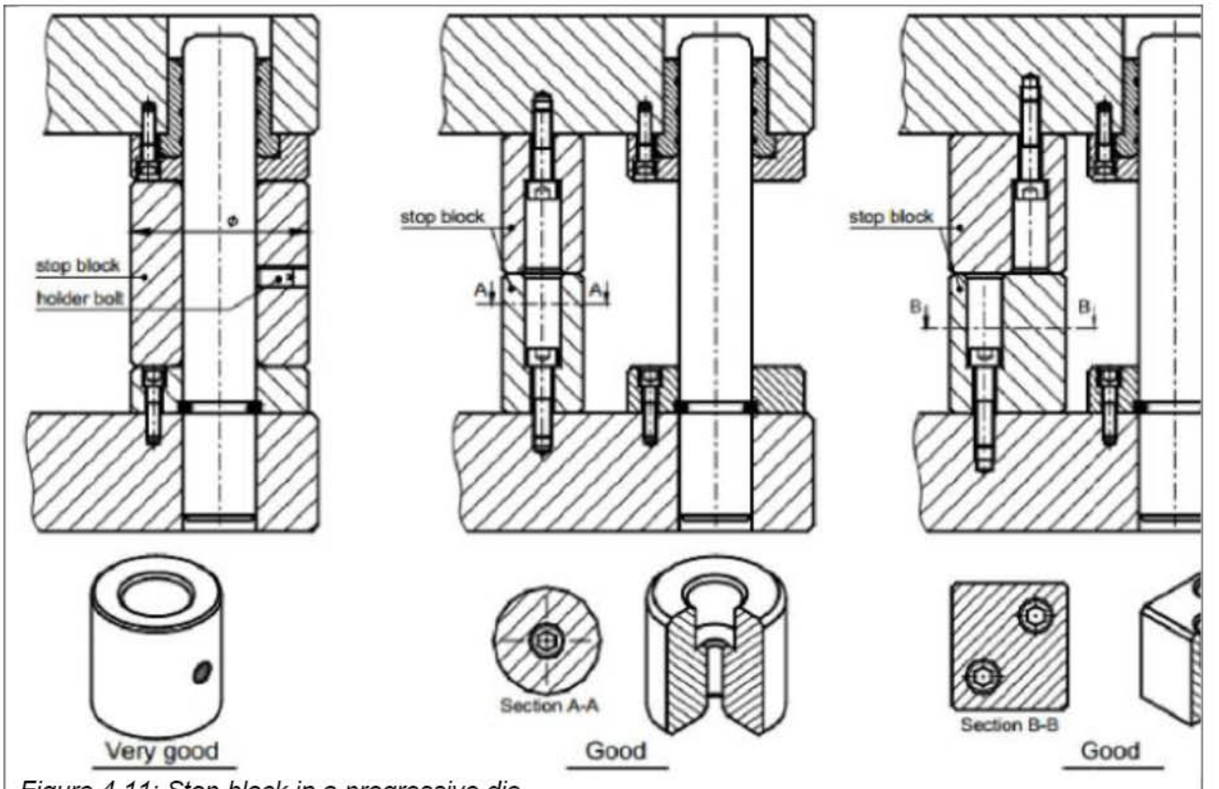


Figure 4.11: Stop block in a progressive die

b4. Stock block

When storing progressive dies in warehouse, to ensure that the punch and die do not collide during transportation and storage, stock block is used. Structures are plastic blocks inserted into the corners of the die to support the two halves of the die to ensure that the punch and die do not collide.

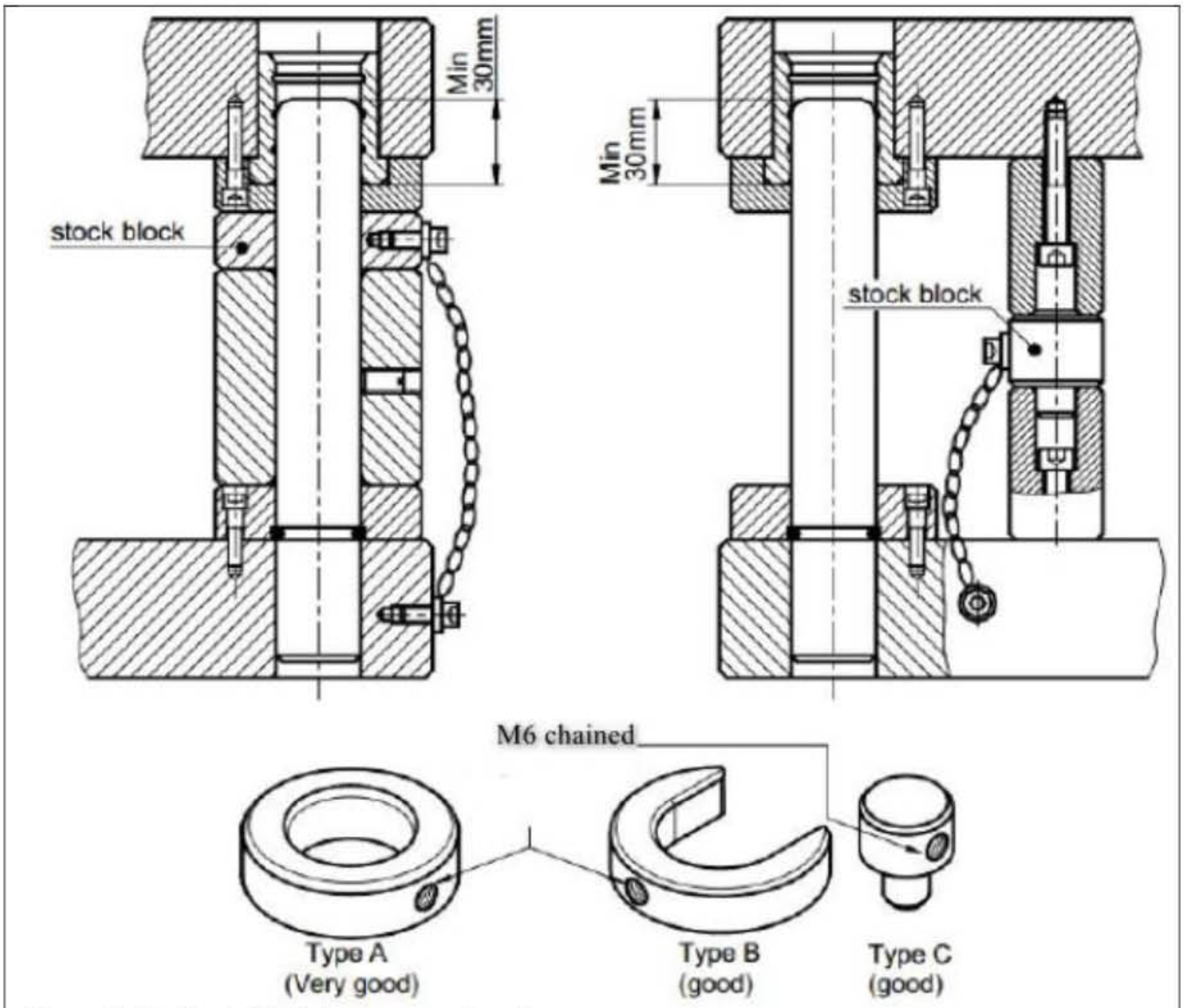


Figure 4.12: Stock block in a progressive die

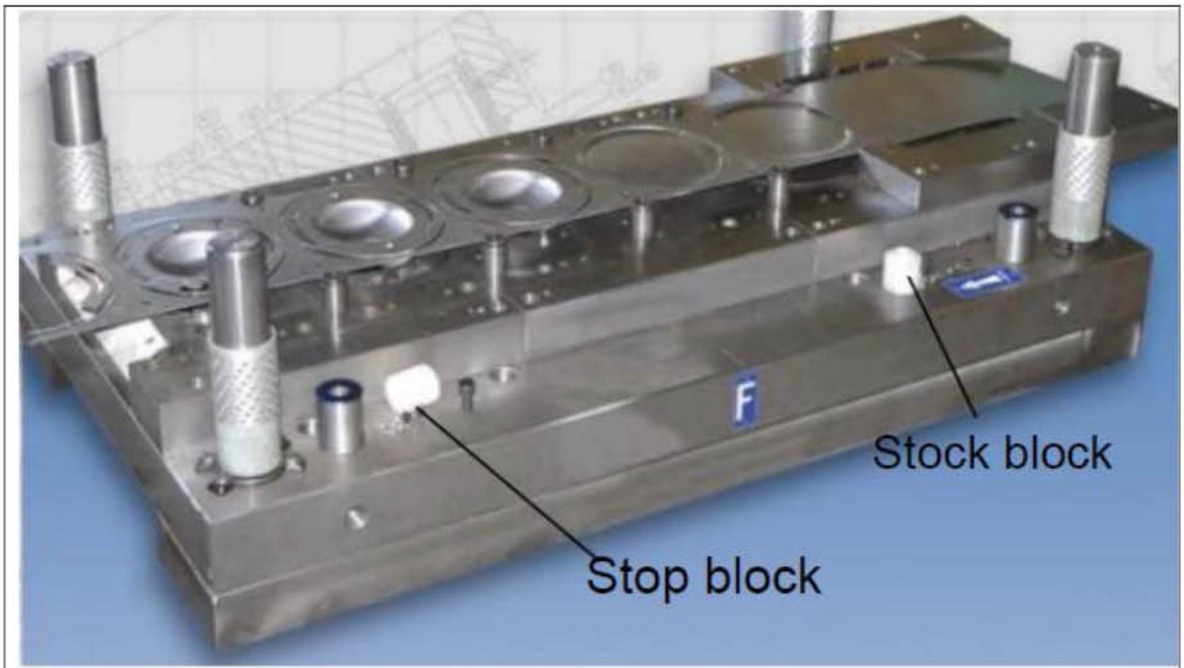


Figure 4.13: Stop block and stock block in a progressive die

b5. Coordinate pin/hole

Determining the pressure center of the die is to bring the die pressure center to coincide with the pressure center of the machine. These pins also have the effect of taking the center to serve the assembly and repair of the die.

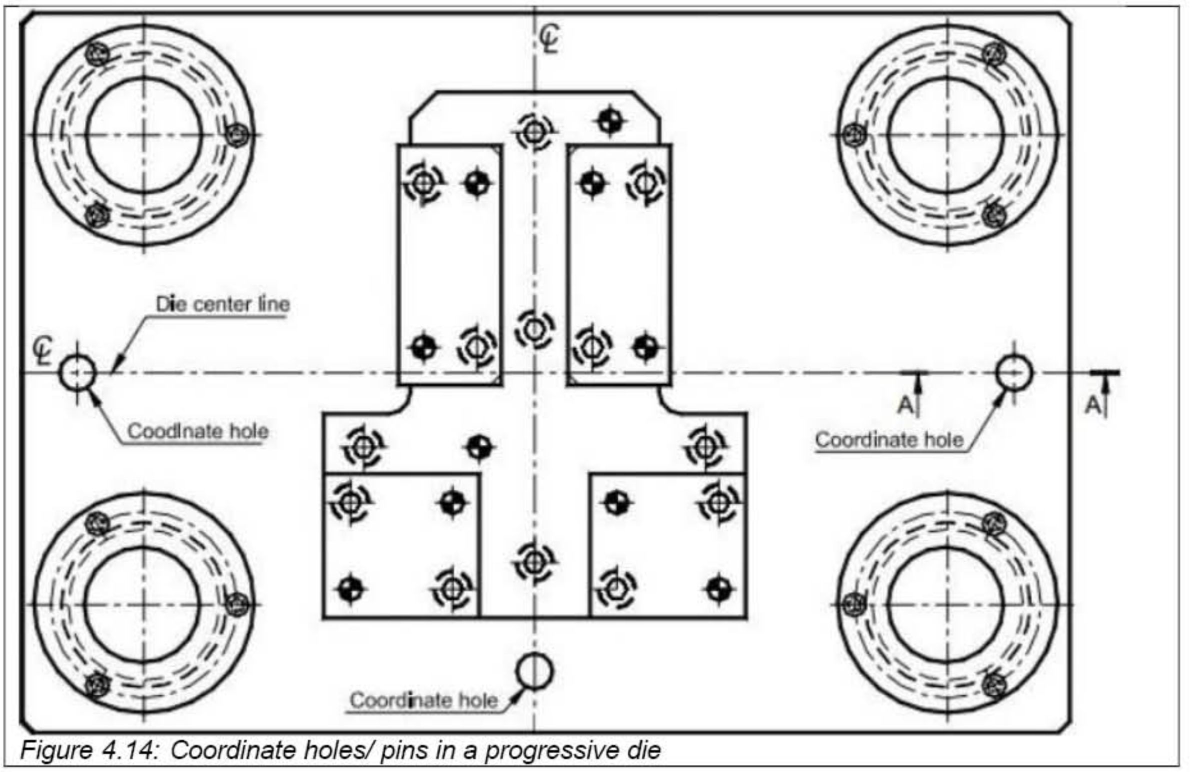
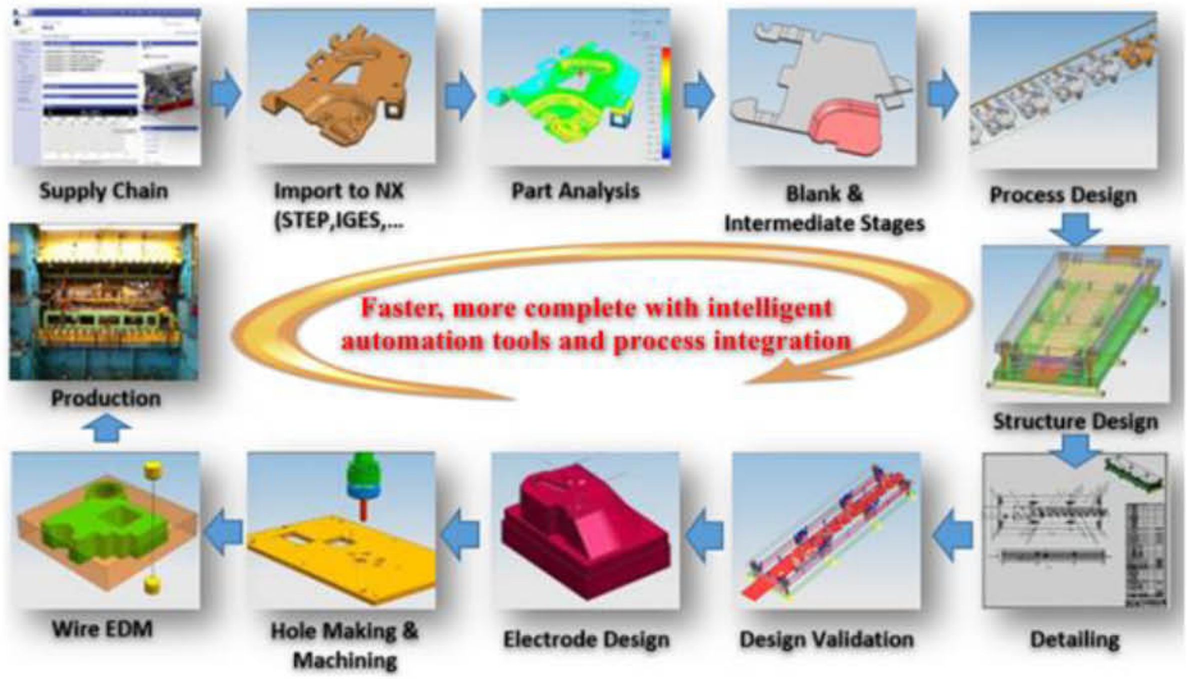


Figure 4.14: Coordinate holes/ pins in a progressive die

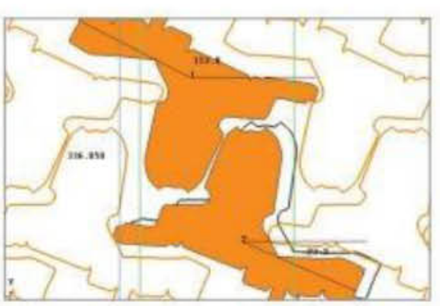
4.2. Overview of NX PDW (Progressive Die Wizard)

4.2.1. Introduction

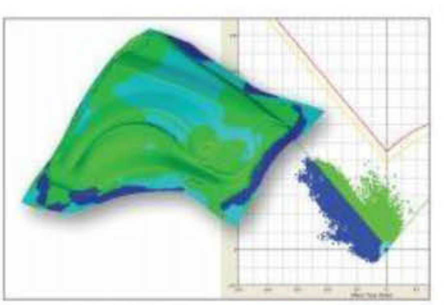
NX progressive die design is a comprehensive solution for both edge and free surface part design processes. You can design a complete structure with relationships between components and stamping operations.



Product analysis: Appraise the design with analysis of the stamping process (product thickness distribution, stress concentration, wrinkles) and flattening the workpiece. Apply Unform and Formability analysis tools for stamping operations and create models for stamping operations with complex profiles.



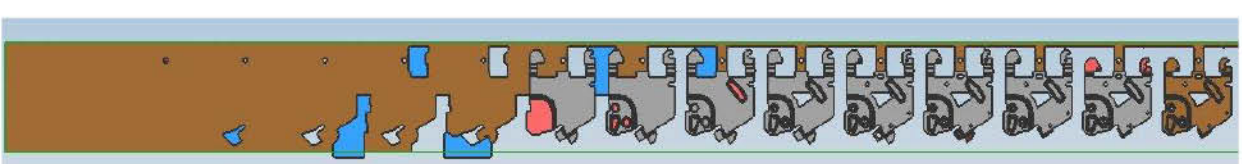
Blank nesting to maximize material utilization



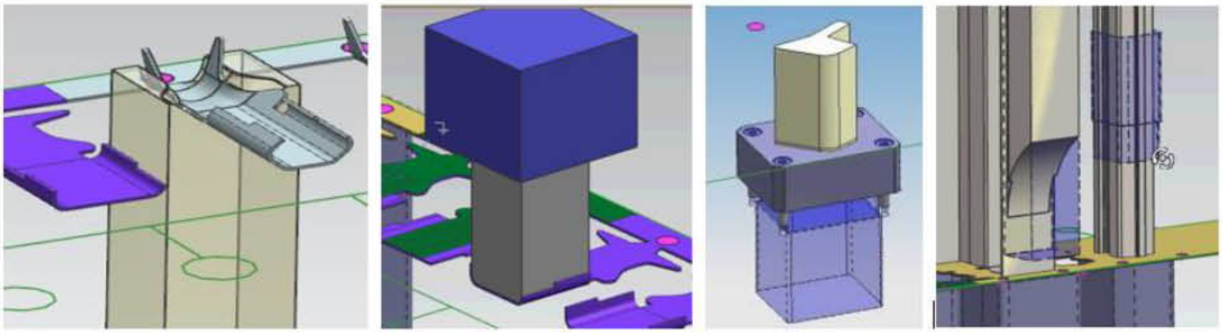
Optimization based on formability

Mesoscopic scale composite forming simulation

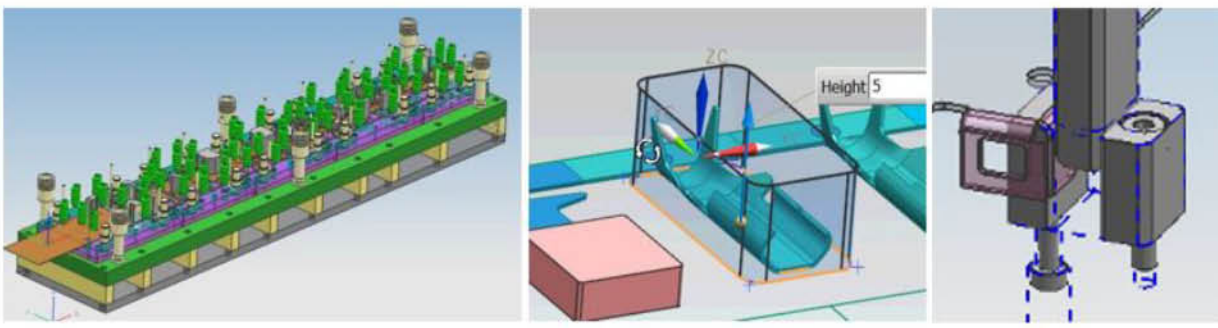
Design of stamping operations: Tool to design layout of stamping operations and design punching and cutting areas. Definition of stamping operations, reasonable arrangement of stamping operations. Simulation of the stamping process to ensure that the design operations are reasonable.



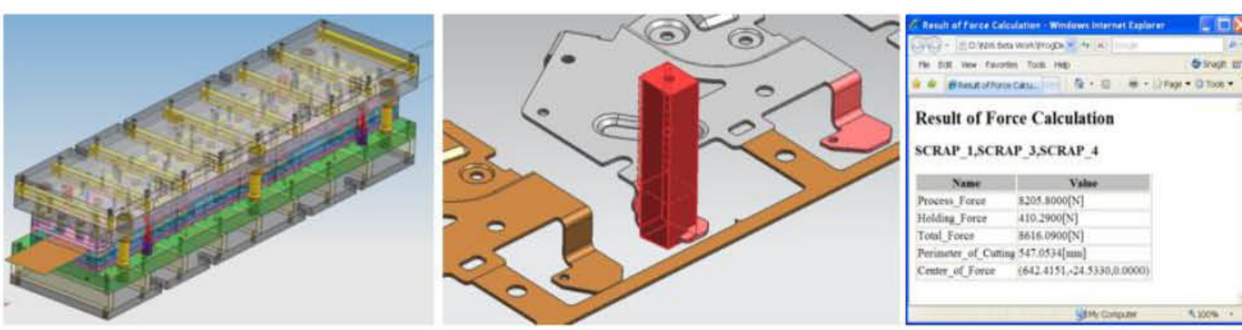
Structural design: Automatically insert all standard die components, design models of punches, dies, punching and cutting tools, design additional mold components. Easily change the design with ease.



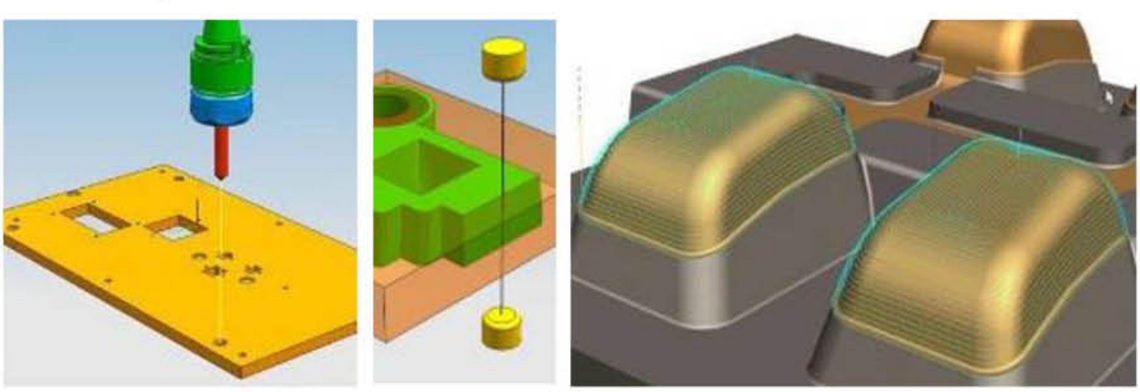
Detailed design: Tool to automatically export 2D technical drawings, create hole lists, record 3D dimensions, record hole tolerances, 3D dimension recording tool allows easier production connection.



Design appraisal: Appraise the entire design in the assembly environment of the completed die system, including tolerances and exit grooves. Analyze materials and force balance, simulate the entire stamping process, calculate and create reports on stamping force and force application points.



Integrated solution: Integrated solution with CAM for CNC machining and CAE for analytical simulation helps reduce time and costs.



4.2.2. The relevant commands

Command	Description
Initialize Project	Lets you select a part, specify project name and path, select material, and create a **Progressive Die** project.
Blank Generator	Lets you create blanks in your project once you have completed project initialization.
Blank Layout	Lets you set the flat pattern orientation, the pitch, and the strip width.
Scrap Design	Lets you determine scrap profiles, split the strip layout, create scrap features, and define overlap and over-cut.
Strip Layout	Lets you generate the strip layout boundary, retrieve design information to generate process features, and do Process Planning for **Progressive Die** design.
Force Calculation	Lets you calculate force for each process feature, calculate total force, and display the force center.
Die Base	Lets you load a die base from a catalog, then split the die plate.
Die Design Setting	Lets you specify the values of progressive die parameters.
Piercing Insert Design	Provides you with functions for creating and modifying piercing punches and dies.
Bending Insert Design	Lets you create a bending punch and die.
	Lets you create a forming punch and die.

Forming Insert Design	
Burring Insert Design	Lets you create a burring punch and die.
Insert Auxiliary Design	Lets you add shanks to strength punch or die inserts, create and edit punch mounts, or copy or delete inserts.
Standard Parts	Lets you add to the die base from a library of standard or custom parts and group sets.
Relief Design	Lets you select the forming feature and relief type to create user-defined and automatic relief pockets.
Pocket Design	Lets you select one of five methods to create pockets.
Bill of Material	Lets you create and export a bill of material to a spreadsheet (.xls) file.
View Manager	Lets you change the color and visibility of components, and show tools in 3 positions.
Tooling Validation	Provides tools to check interferences, run motion simulations, and get advice on the effects of design changes.
Workflow Management	Gives you access to the following functions: **Changeover Management**: Lets you create one or more changeovers in your project, which lets you store similar sheet metal products in the same die design. **Concurrent Design Management**: Lets multiple designers work on the same project concurrently.
	Lets you calculate a quick cost estimation of your progressive die based on your conceptual design.

Quick Quotation	
NX Generic Tools	Lets you access the **NX Generic Tools** gallery. The **NX Generic Tools** gallery is populated with a collection of often-used NX tools, such as **Extrude**, **Edge Blend**, **Sew**, and synchronous modeling commands.

a. Initialize Project

a1. Function and command call

Use the **Initialize Project** command to either open a project or create a new project for a sheet metal part.

To design a die using Progressive Die Wizard, you must create a project. A project is uniquely defined by a project name (or number) and path, which you define. A project contains all design information related to your tool: sheet metal parts, part material, and all subsequent parameters and components (such as strip, die bases, punches, dies, or screws) that you add to your tool design.

You can use the **Initialize Project** dialog box options to:

Insert or remove sheet metal parts or a strip layout.

Specify a project path and name.

Specify the part material.

Edit the material data base.

Stamp multiple parts in a die.

The units, thickness, and material of the inserted part or strip layout are automatically evaluated and displayed in the **Initialize Project** dialog box.

Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Initialize Project**

a2. Initialize Project dialog box

Project	
Sheet Metal Parts	Lists the sheet metal parts open in your session. **Add** lets you browse for additional parts to add to the **Sheet Metal Parts** list. **Remove** deletes the selected part from the list.
Project Path and Name	Shows the name and path of your current project.

 **Browse Project Path**	Lets you browse for an existing project.
Part Unit	Shows whether your part's units are Metric or English.
Part Thickness	Shows the thickness of your sheet metal part.
Part Material	Shows the part's material, and lets you change it from a list of eligible materials.
Project Template	Lets you select a template to use with your project.
Settings	
 **Edit Material Data Base**	Lets you edit the material parameters spreadsheet.
 **Edit Project Template Configuration**	Lets you edit the project template configuration spreadsheet. This is useful when you want to add customized project templates.
Rename Components	Lets you rename each component of the design assembly template.
Insert Strip	Lets you insert an existing strip layout into your project. You must select the top face as the stationary face in the strip. Keeps the design association related to the inserted strip by directly adding it to the project. The strip part may be a subassembly.

b. Blank Generator

b1. Function and command call

Use the **Blank Generator** command to create blanks in your project once you complete project initialization. You can:

Insert a blank from a file.

Select a blank body from the sheet metal part.

Update a blank.

Remove a blank.

Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Blank Generator** 

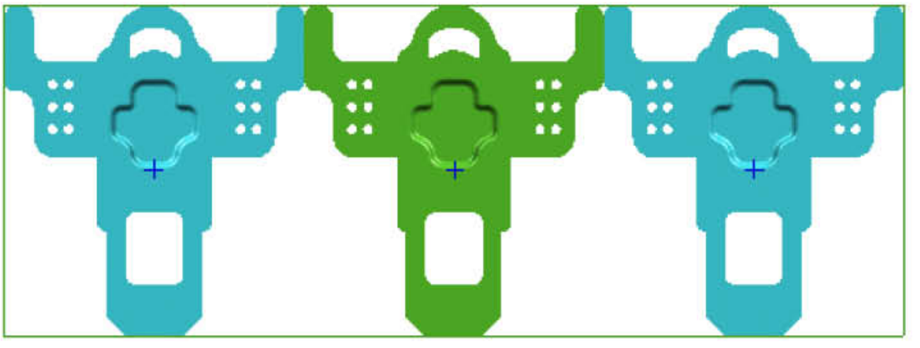
b2. Blank Generator dialog box

Type	
list	Specifies whether to create or edit a blank.
Create	
Appears when **Type** is set to **Create**.	
Sheet Metal Part	Lets you select a sheet metal part from a list of available sheet metal parts.
 **Import Blank Part**	Imports a sheet metal part as a blank.
 **Select Blank Body**	Lets you select a sheet metal part to create the blank.
 **Planar Face**	Lets you select a planar stationary face to define the blank.
Edit	
Appears when **Type** is set to **Edit**.	
Blank Part	Lets you specify the blank to edit.
 **Update Blank**	Updates the blank after you make changes.
 **Delete Blank**	Deletes the blank.

c. Blank Layout

c1. Function and command call

Use the **Blank Layout** command to orient parts in a flat pattern relative to each other, to decide what strip width to use, and to determine the distance between process stations. The sheet metal blank must be complete before you can layout the strip and design the scrap.



You can use the **Blank Layout** command to:

Insert blanks and put new instances of a blank under *_*nest* part.

Design the orientation of the blank.

Copy blanks.

Remove blanks.

Set a base point for rotation origin.

Modify burr direction by flipping the blank.

Set the pitch, strip width, rotation angle, and number of rows.

View the percent of material utilization and the minimum space between blanks.

Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Blank Layout**

c2. Blank Layout dialog box

Type	
Type menu	Lists the operations you can perform.

	Create Layout Creates the initial blank layout. (This ia a required initial step.) **Add Blank** Adds the blank from the list. **Copy Blank** Copies the selected blank to a second row. **Remove Blank** Deletes the selected blank. **Set Base Point** Sets the base point for reference to move, copy, and rotate blanks. The software uses this point as the “0” station’s start point in **Strip Layout**. It is better to set this point to a pilot hole or another meaningful point. **Flip Blank** Turns the blank over, reversing the action direction of punch and die tools in the die assembly.
Blank	
 **Select a Blank**	Lets you select a blank for the layout.
 **Select the Base Point**	Appears when **Type** is set to **Set Base Point** Lets you select a base point to place the blank.
Blank List	
Appears when **Type** is set to **Add Blank**	
List	Lists the blanks you added.

The following options appear only when **Type** is set to **Create Layout**.

Placement	
Shift X	Moves the selected blank along the X axis. Type a new value, or use the slider to move the blank in snap-size increments.
Shift Y	Moves the selected blank along the Y axis. Type a new value, or use the slider to move the blank in snap-size increments.

Rotate	Turns the selected blank around the base point. Type a new value, or use the slider to move the blank in snap-size increments
Pitch-Width	
Pitch	Specifies the distance between parts in the strip layout. The software automatically computes the minimum pitch as a starting point.
Width	Specifies the width of the strip. The software automatically computes a minimum strip width on the blank layout.
Side Webs	
side webs options	Assigns side webs to the layout. You can specify one of the following: **Average** Available when you select the **Average** radio button. Assigns equal webs to each side of the layout. **Bottom** Available when you select the **Bottom** radio button. Specifies the bottom distance for the side web. You can either type values in the boxes, or move the sliders to specify the distance. **Top** Available when you select the **Top** radio button. Specifies the top distance for the side web. You can either type values in the boxes, or move the sliders to specify the distance.
Optimization Data	
Material Utilization	Displays the percent of material the current layout uses.
Minimum Space Size	Displays the minimum area necessary for the specified layout, excluding side webs.
Settings	
Snap Size	Specifies the increments of the **Placement**, **Pitch-Width**, and **Side Webs** sliders.
Show Three Blanks	Appears only in the Engineering Die Wizard application. Lets you specify whether a single blank or three blanks are displayed.

d. Scrap Design

d1. Function and command call

A scrap is a sheet body created from any of the following:

A blank's boundary plus additional curves

A hole boundary

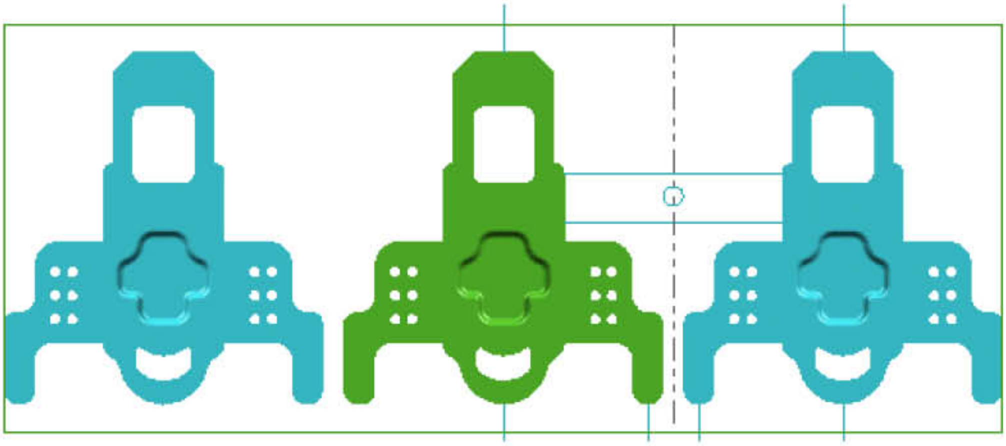
A closed loop of curves

An existing sheet body

Use the **Scrap Design** command to specify how to punch out scrap piercings and, if needed, create pilot holes to aid in transporting the metal strip through the die. Pilot holes can be any shape, but are often circles.

All scraps are sheet bodies and put into a *"*_nest"* part.

After you finish using **Blank Layout**, use **Scrap Design** to pierce the strip and form the profile shape of the sheet metal part. **Scrap Design** determines scrap profiles based on the blank layout result.



You can use **Scrap Design** to:

Design all scraps and assign station numbers.

Design individual scrap you profiled by using, for example, sketching, existing sheet bodies, or holes.

Extract internal holes as scraps automatically.

Specify multi-hit, precision-punched scrap trims.

Split scrap areas into smaller pieces. The scrap can be regular/piercing scrap or pilot scrap.

Merge scraps.

Design and edit overlaps and overcuts, which you can add to any scrap, including pilot scrap. You can also add overcuts to overlaps.

Apply a minimum radius to all sharp corners of a scrap.

Create pilot holes for metal strip transport.

Set the display color for scraps, holes, overlaps and over-cuts.

Create a group of scraps that are the same shape and size.

Change regular (piercing) scrap to pilot scrap, and vice versa.

Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Scrap Design**

d2. Scrap Design — Create options

Note See Common dialog box options for common options not discussed here.

The options in the **Scrap Design** dialog box vary, depending on the specified operation in the **Type** group. The following options appear when **Type** is set to **Create**.

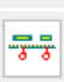
Method	
Method icons	Specifies the method to use in creating the scrap: **Blank Boundary + Sketch** Creates scrap based on curves that remains associative to the blank. **Hole Boundary** Extracts holes you select in the blank and defines them as hole scrap automatically. **Closed Curves** Creates scrap based on a closed loop of edges and curves you select as the boundary. If an edge or curve is adjacent to another edge or curve, they must be joined together. If an edge is adjacent to a curve that is not an edge, they must intersect. **Existing Sheet Body** Creates scrap based on an existing sheet or group of sheets you select. **Change Type** Lets you change piloting scrap to piercing scrap, or vice versa.
Select Curve	Appears only when the method icon is **Blank Boundary + Sketch** or **Closed Curves**. Lets you select a closed loop of curves in the graphics window, or you can click **Sketch Section** to create curves that, by themselves or along with a blank boundary, complete a closed loop.
	Appears only when the method icon is **Existing Sheet Body**. Lets you select one or more sheet bodies to define the scrap or a group of scraps.

Select Sheet Body	
Station Number	Lets you assign a station number to the scrap.
Setting	
Scrap Type	Lets you specify whether the scrap is piercing or piloting. Note The **Piloting** setting is ignored unless the method is **Closed Curves**.
Position	Lets you specify the position of the scrap in a strip layout: **Project to Strip** **Keep Origin**
Tolerance	Lets you specify the tolerance for cutting off scrap.
Scrap Color	Lets you define colors to distinguish different types of scrap.

d3. Scrap Design — Edit options

Note See Common dialog box options for common options not discussed here.

The options in the **Scrap Design** dialog box vary, depending on the specified operation in the **Type** group. The following options appear when **Type** is set to **Edit**.

Edit	
You can choose any of the following six icons to define the edit operation.	
 **Split**	Divides large scrap into two smaller pieces. The pieces are split along curves you select or create with the Sketch task environment.
 **Merge**	Joins adjacent pieces of scrap together.
 **Apply Minimum Radius**	Lets you specify a minimum radius to round sharp corners in the scrap.
	Lets you change the scrap station number.

Change Station	
Delete	Deletes scrap pieces you select from the blank. You can use the **Filter** options to help you select the scrap.
Update	Updates all defined scraps after you change the blank profile.
Select Scrap	Appears for all edit operations except **Update**. Lets you select a scrap for an operation.
Select Split Curve	Appears only for the **Split** edit operation. Lets you select existing curves for a split operation. If the curves you want do not exist, you can click **Sketch Section** to open the sketch environment, where you can create the curves. See the *Sketching* Help for more information.
Minimum Radius	Appears only for the **Apply Minimum Radius** edit operation. Lets you specify a minimum radius that is used to round sharp corners.
Station Number	Appears only for the **Change Station** edit operation. Shows the current station number of the scrap, which you can modify.
Filter	Appears only for the **Delete** edit operation. Lets you specify the scrap type, which helps you select it in the graphics window.

d4. Scrap Design — Add-on options

Note See Common dialog box options for common options not discussed here.

The options in the **Scrap Design** dialog box vary, depending on the specified operation in the **Type** group. The following options appear when **Type** is set to **Add-on**.

Add-on	
You can choose any of the following four icons to define the add-on operation.	
Overlap	Lets you define the overlap between two adjacent scraps.

Overcut	Lets you add an overcut on to a scrap overlap corner.
Trimming	Lets you define the scrap precision trimming processes.
User Defined	Lets you create a custom add-on for scrap.
Select Scrap	Appears for every add-on operation except **Overcut**. Lets you select a scrap.
Select Edge of Scrap	Appears only when **Overlap** is selected. Lets you select a scrap edge for the overlap.
Overlap Width	Appears only when **Overlap** is selected. Lets you specifies the overlap width.
Overcut Type	Appears only when **Overcut** is selected. Lets you specify the type of overcut.
Select Side Edge	Appears only when **Overcut** is selected. Lets you select a side edge of the overlap to place the overcut.
A **B** **H** **R**	Appears only when **Overcut** is selected. Lets you define the parameters of the overcut.
Number of Trimming	Appears only when **Trimming** is selected. Lets you specify the number of trimming processes.
Process	Appears only when **Trimming** is selected. Lets you specify the size for each trimming process.

Select Curve	Appears only when **User Defined** is selected. Lets you select existing curves to define your custom add-on. If the curves you want do not exist, **Sketch Section** opens the sketch environment, where you can create curves. See the *Sketching* Help for more information.

d5. Scrap Design — Grouping options

Note See Common dialog box options for common options not discussed here.

The options in the **Scrap Design** dialog box vary, depending on the specified operation in the **Type** group. The following options appear when **Type** is set to **Grouping**.

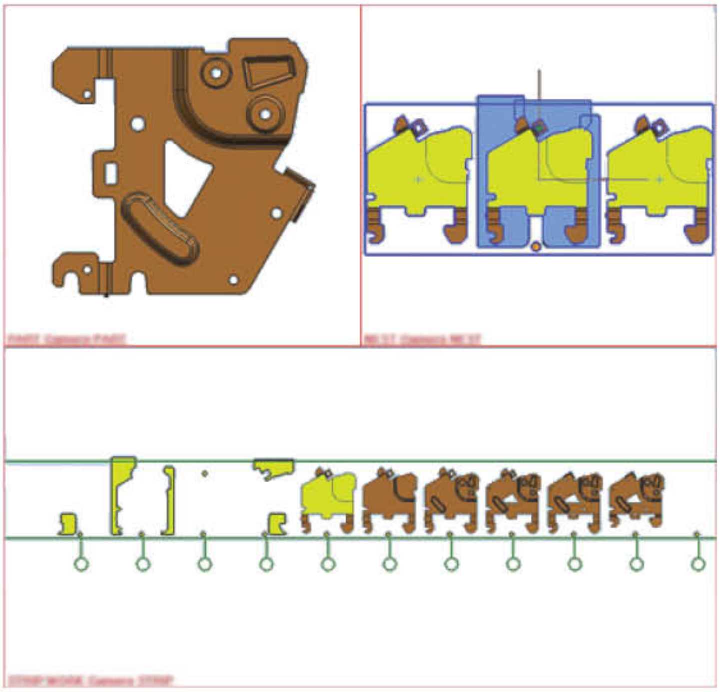
Assign Color to Same Scrap	
Select Scrap	Lets you select scrap in the graphics window.
Select Color	Lets you select a color to apply to the selected scrap.
Apply Color to Scraps of Same Size	Controls whether the selected color is also applied to all scraps of the same size.

e. Strip Layout

e1. Function and command call

A die strip layout is required for every progressive die.

Use the **Strip Layout** command to create that layout by setting the number of stations, assigning scrap and processes to specific stations, then simulating the forming of the sheet metal part.



Strip Layout enables the relationships between associative intermediate stages to be maintained with the original sheet metal part, so you can make changes to a die design by modifying its associated parts. Even clearing a simulation will not remove associative intermediate stage associations.

When you lay out all the processes in **Strip Layout**:

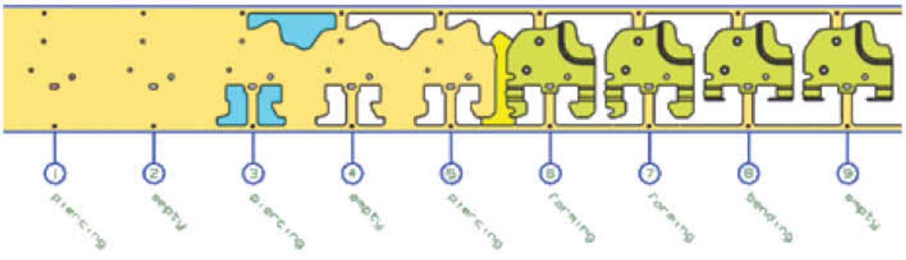
Each scrap process in the strip layout is represented by a sheet body.

Each intermediate stage is represented by a solid body.

Your strip layout view can have either one or three views, depending on your **Views in Strip Layout** customer default setting.

To add a description of the process performed at each station, right-click a station name in the **Strip Layout Navigator** and choose **Add Process Description**. You can set the font, scale, and color of the description text if desired.

NX also displays the station number inside the circle of each station.



Tip To find a customer default, choose **File→Utilities→Customer Defaults**, and click **Find Default**.

Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Strip Layout**

e2. Create a strip layout

Choose **Progressive Die Wizard** gallery→**Strip Layout** .

The **Strip Layout Navigator** appears. Its **Strip Layout Definition** node has the following child nodes:

Pitch

Width

Feeding Direction

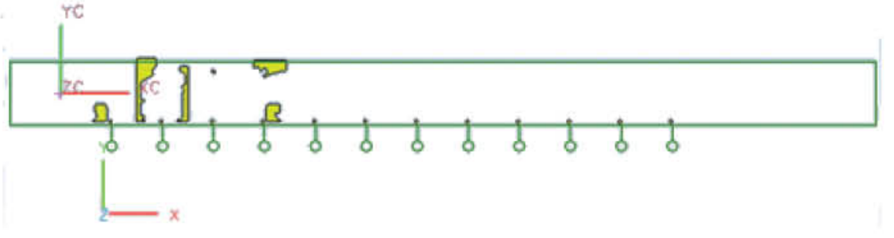
Station Number

In the **Strip Layout Navigator**, double-click the **Station Number** node and enter the number of stations you want.

Right-click the **Strip Layout Definition** node and choose **Create**.

Scraps that were not assigned to stations during scrap design appear in the **Strip Layout Navigator** under the **Unprocessed** node.

The strip layout appears, as shown in the following figure.



Strip showing scrap with preassigned stations

The **Strip Layout Navigator** now shows a node for each station. Other nodes appear if appropriate, such as an **Unprocessed** node, and nodes for any existing scrap, intermediate bodies, or intermediate parts.

If you specified station numbers during scrap design, those scraps appear in the graphics window at their strip layout station nodes, and also under their corresponding station nodes in the **Strip Layout Navigator**.

If you want to move a scrap to a station, drag it in the **Strip Layout Navigator**.

You can drag multiple scraps to a station.

An alternate way to move a scrap is to right-click the scrap, choose **Move Up** or **Move Down**, and choose the station number you want.

If you want to change the strip feed direction, do the following:

Double-click the **Feeding Direction** node.

Specify **0** to feed the strip from left to right, or **1** to feed the strip from right to left.

Right-click the **Strip Layout Definition** node.

Choose **Update** to update the strip.

If you want to edit the strip pitch or width, do the following:

Double-click the **Pitch** or **Width** node.

Specify the new pitch or width.

Right-click the **Strip Layout Definition** node.

Choose **Update** to update the strip.

You can insert an intermediate stage as either a body or a part, depending on the procedure used to design it:

Intermediate Body — Use this option when the intermediate stage was defined as a separate body within the original sheet metal part added to the project.

To insert an intermediate body, drag it from the **Unprocessed** node to the station where you want to place it.

Intermediate Part — Use this option when the intermediate stage was defined within an intermediate staged assembly by the **Direct Unfolding** command, or another method.

To insert an intermediate part, do the following:

Right-click the **Intermediate Part** node.

Choose **Open**.

In the **Select Part** dialog box, specify the appropriate parent part of the intermediate staged assembly.

Click **OK**.

If the intermediate part has a preassigned station number, it automatically appears at that station. Otherwise, you can drag the intermediate part to the station where you want it.

To insert an idle station and create intermediate stages in the **Strip Layout Navigator**:

Right-click the station before the desired location of the idle station.

Choose **Insert Idle Station** to insert an empty station.

Right-click the intermediate stage body or part before the empty station, and choose **Copy**.

Right-click the empty station, and choose **Paste**.

The station is now an idle station.

Optional - You can choose **Insert Process Station** in step 2 to create a new station after the current station. This will also copy the intermediate stages from the current station into the new station.

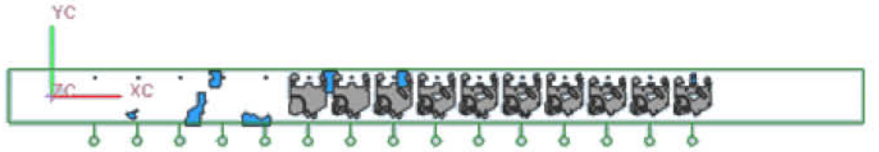
To delete a station, in the **Strip Layout Navigator**, right-click it and choose **Delete**.

When you finish defining the stations, you can simulate the piercing process.

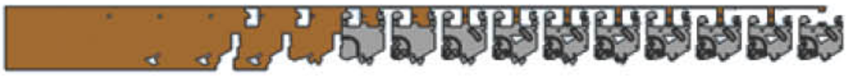
In the **Strip Layout Navigator**, right-click the **Strip Layout Definition** node.

Choose **Simulate Piercing**.

The software generates a solid body representation of the strip based on the scraps, intermediate stages, and station definition, as shown in the following figures.



A completed strip layout



A simulated piercing

The simulation's resemblance to the actual physical strip is improved if you remove the blank material from stations with intermediate stages. To do this, right-click the **Strip Layout Definition** node, choose **Remove Blank Material**, and define the range of stations in the **Strip Layout Design** dialog box.

To delete the simulation, right-click the **Strip Layout Definition** node again, and choose **Clear Simulation**.

Optional - You can create a description at each station.

Right-click a station name in the **Strip Layout Navigator** and choose **Add Product Description**.

Specify a description for the station. You can also specify the font, scale, and color of the tex

e3. Strip Layout Navigator options

Node	Description
Strip Layout Design Center	Provides the top-level node for all the other nodes in the **Strip Layout Navigator**.
Strip Layout Definition	Specifies the parameters of the strip. Its child nodes are: **Pitch** — Specifies the distance between strip stations. **Width** — Specifies the width of the strip. **Feeding Direction** — Sets the direction in which your strip feeds through the die. **Station Number** — Specifies the number of stations in the die.
Unprocessed	Appears after you create a strip layout. Shows any existing scrap or intermediate stages (bodies or parts) that are available to be inserted in a strip station.
Station nodes	Appears after you create a strip layout. Shows the scrap and intermediate stages at each station. You can right-click a scrap node to move it to another station. A station node appears for each station.

Strip Layout Definition node shortcut menu

The following table lists the shortcut menu commands that appear when you right-click the **Strip Layout Definition** node.

Command	Description
Create	Creates the initial strip layout. Scrap with preassigned stations appear at the assigned stations.
Update	Updates the strip layout after a blank change, scrap change, pitch change, or width change.
Simulate Piercing	Simulates the piercing operation, so you can preview the physical appearance of a strip.
Clear Simulation	Deletes the piercing simulation.
Update Simulation	Updates the piercing simulation.
Remove Blank Material	Removes blank material from stations with intermediate stages, to make a piercing simulation more accurately resemble the physical strip.
Update Blank Orientation	Updates the orientation of the blank.
Drag/Drop Intermediate	Lets you lock or unlock the ability to drag intermediate stages.

Station node shortcut menu

The following table lists the shortcut menu commands that appear when you right-click the **Station** node.

Command	Description
Insert Process Station After	Adds a new process station after the selected station. The software clones the intermediate stage from the selected station to the new station.
Insert Idle Station After	Adds an idle station after the selected station. The idle station is empty, so you can copy the intermediate stage you want to this station.
Delete Station	Deletes the current station.
Add Process Description	Lets you add a description of the process that is performed in the station. You can set the font, scale and color for the description text. Note You can set the default process in the **Customer Defaults**. You can display the **Station Number** in the station circle permanently.
Paste	Lets you paste an intermediate stage you copied to the selected station.

Process node shortcut menus

The following table lists the shortcut menu commands that appear when you right-click the **Process** node for a scrap.

Command	Description
Move Up	Lets you move the scrap up to the station you select from this list.

Move Down	Lets you move the scrap down to the station you select from this list.

The following table lists the shortcut menu commands that appear when you right-click the **Process** node for an intermediate stage.

Command	Description
Close	Closes the current intermediate stage.
Copy	Copies the current intermediate stage.
Move Up	Lets you move the intermediate stage up to the station you select from this list.
Move Down	Lets you move the intermediate stage down to the station you select from this list

f. Force Calculation

f1. Function and command call

Use the **Force Calculation** command to calculate the forces for processes. Scraps created in the previous **Scrap Design** step are automatically included in the process list.

To calculate the force of a process, you can do any of the following:

Select a process from the list and quickly calculate its forces. The software chooses the formula for the calculation.

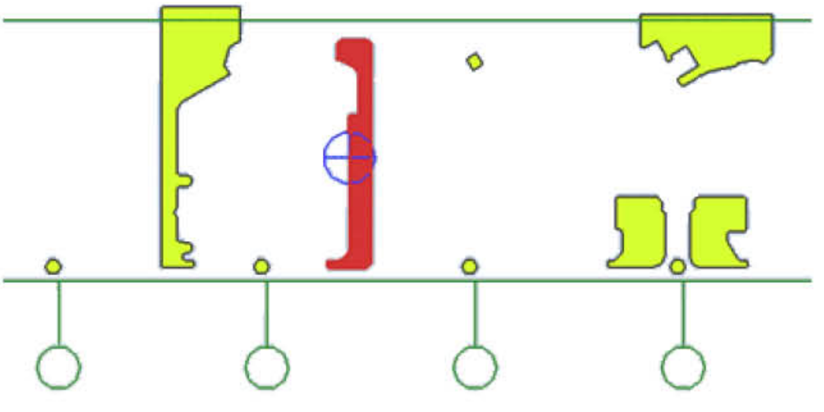
Edit the formulas used for calculation, then select a process and calculate the forces.

Create a user defined process. You specify the type of process (such as piercing or bending) and select faces for the process. You can then calculate the force.

Calculate the total force for all processes with current force calculations.

> Note An uncalculated process has an asterisk * before its name in the process list. To include it in the total force calculation, you first need to select the process and calculate its forces.

The results appear in the **Force Calculation** dialog box, or you can create an .xml report that can be displayed in your browser.



Select the piercing for a force calculation

Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Force Calculation**

f2. Force Calculation dialog box

Note See Common dialog box options for common options not discussed here.

Process List	
list	Lists the currently defined processes for selected faces in your model. Scraps from the previous **Scrap Design** step are automatically listed. You can select additional faces and specify processes for them. When you calculate the force for a process, the * character at its beginning is removed.
Define New Process options	Lets you select a face and define a new process, which you can then add to the process list. NX displays a graphical representation labeled with the listed values for the selected process.
Calculation	
Select Process	Lets you select the process that you want to calculate.
parameters	Shows the results of a force calculation for the following parameters: **Process Force** **Holding Force** **Perimeter of Cutting** **Center of Gravity**
Calculate	Calculates the selected process in the process list.
Create Report	Generates an .xml report that can be displayed in a browser.

 **Calculate Total Force**	Calculates the total force for all processes in the **Process List**.
Settings	
Calculation Type	Lets you specify the type of calculation: **Overlapped Cutting** The software calculates the force after removing from consideration the common edges of adjacent scrap and the length of scrap edges that extend beyond the strip boundary. **No Overlapped Cutting** The software calculates the force according to the full boundary of each scrap. This method is faster than the **Overlapped Cutting** method, and is commonly used when calculating the force for holes.
Decimal Places	Specifies the number of decimal places for the calculation.
Edit Formulas	Lets you customize formulas used to interactively calculate force

f3. Calculate forces in the Progressive Die Wizard

Choose **Progressive Die Wizard** tab→**Main** gallery→**Force Calculation**  , or choose **Menu**→**Tools**→**Process Specific**→**Progressive Die Wizard**→**Force Calculation**.

The **Force Calculation** dialog box appears, with the scraps listed in the **Process List**.

In the **Process List**, arrange the scrap processes as desired.

You can select a scrap process and remove it, or move it up or down in the list.

(Optional) To define a new process, do the following:

In the **Define New Process** subgroup, from the **Process Type** list, select the type of process.

In the graphics window, select one or more faces from the strip layout, simulation part, or intermediate part.

In the **Define New Process** subgroup, in the **Process Name** box, specify a name for the process.

Click **Add** .

The new process is added to the list. The asterisk * at the front of its name shows that the force of the process is not calculated yet.

(Optional) In the **Settings** group, click **Edit Formulas** to open a spreadsheet where you can edit the formula used for the calculation.

In the **Process List** group, from the list, select one or more processes.

In the **Calculation** group, click **Calculate** .

The results appear in the **Calculation** group in the boxes for the following parameters:

Process Force

Holding Force

Perimeter of Cutting

Center of Gravity

(Optional) Click **Create Report** .

The results appear in the browser, where you can save them for further analysis.

(Optional) After you calculate all the processes you want, click **Calculate Total Force**  to calculate the total force for all currently calculated processes.

The results appear in the **Calculation** group and in your browser.

Click **OK** or **Apply**.

The results are automatically written onto the features' sheet bodies as the following attributes:

FORCE_CENTER_X

FORCE_CENTER_Y

FORCE_CENTER_Z

CUTTING_LENGTH

PRO_FORCE

HOLD_FORCE

g. Die Base

g1. Function and command call

Use the **Die Base** command to lay out a die base in your progressive die design. A die base consists of an assembly of standard parts, including plates, guide pins, guide bushings, and screws.

You can:

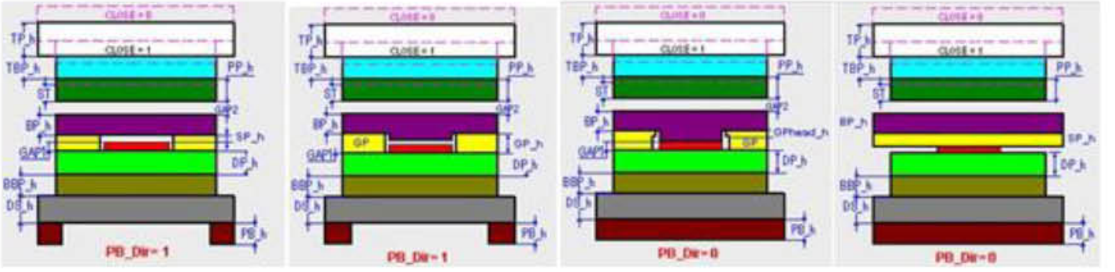
Add a die base from standard catalogs to your die design.

Edit an existing die base.

Split, merge, or delete die plates.

Modify the length of a die plate.

Add a die base with part family components. A sample template, **<DB_UNIVERSAL_PartFamily>**, is provided to create a die base with part family components.



UNIVERSAL_SIMPLE die bases

When you add a die base, the die base NX provides for your strip layout is based on your inputs in the **Manage Die Base** dialog box. NX may switch to a different default die base when you modify any of the following inputs:

The **Catalog**

The **Plate Numbers**

The **Value** of a parameter in the **Details** table

Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Die Base**

g2. Manage Die Base dialog box

Create Die Base	
Appears when **Type** is set to **Design Die Base**. You can also use these options to select and edit an existing die base.	
Select Die Base to Edit	Lets you select an existing die base for editing.
Parent	Lets you specify the parent subassembly node where the die base will be installed. Tip You can review the assembly structure of your die design in the **Assembly Navigator**.
Catalog	Lets you specify a catalog from which you can select a standard die base. The catalog list is dependent on the project units and the die base register file. You can edit this file to customize your installation. For example, when the

	project units are in inches, only die base catalog names that are defined in the English units sheet of the register file appear in the **Catalog** list.
Plate Numbers	Lets you specify the number of die base plates for the installation.
Pick Work Area	Lets you specify the work area that includes defined processes by selecting two points on the strip. Note Because die base choices are limited to those offered in the catalog you specified, only die bases from the catalog that meet both the length and width values of the work area are available. For example, the available die bases may all be longer than the length of your defined work area.
Specify Reference Point	Appears only in the Progressive Die Wizard application. Lets you select a reference point in the strip layout to position the die base.
 **Edit Register File**	Opens a spreadsheet that lists all the registered die bases.
 **Edit Data Base**	Opens a spreadsheet that contains all parameter data for the selected die base.
Distance to Die Base Edge	Lets you define the distance from the selected reference point to the die base edge. If the feeding direction for the strip layout is from left to right, the **Distance to Die Base Edge** value is measured from the reference point you select to the left edge of the die base. Otherwise, it is measured from the reference point to the right edge of the die base.
Details	
Appears when **Type** is set to **Design Die Base**.	
Table	Lets you review the parameters of the current default die base. The default die base is available in the specified catalog. If you need a different die base, you can click the **Value** column to modify a parameter. You can select an alternate value, if available, from a list or, for some parameters, you can enter a new value.
Design Tools	
Appears when **Type** is set to **Design Tools**.	

Split Die Plates	Lets you split one or more selected die plates or the whole sub-diebase.
Merge Die Plates	Lets you merge either selected die plates or a whole sub-diebase into a single plate.
Align Die Plates	Lets you align selected die plates or adjust the gap between two plates. It is useful to align the die plates or adjust the gap between plates after you change the length of the die plates.
Adjust Length of Die Plates	Lets you change the length of selected die plates by dragging an edge or entering a new value.
Save as Template	Lets you save the current die base as a template for a new catalog.
Delete Die Base	Deletes the selected die base.
Standard or User Defined	Lets you choose a split method. **Standard** Lets you split die plates from a point. You can specify the direction in the **Direction** subgroup. **User Defined** Lets you split die plates with a curve you create.
Whole Sub-Diebase or Plates	Lets you specify whether the operation occurs for the whole sub-diebase or a selected plate. For a split operation, you can select more than one plate with the **Single Plate** option, but the selected plates must be the same length.
Direction	Lets you specify the direction for the operation.
Select Curve	Appears when you select **User Defined**. Lets you select existing curves or edges.

	Tip To filter curve selection, use the **Curve Rule** options on the Top Border bar.
Sketch Section 	Appears when you select **User Defined**. Opens Sketch task environment to enable you to create a new sketch. The Cue line states that you can also select a planar face to create a sketch. Tip To select all the edges of a face to define the section, set the **Curve Rule** option on the Top Border bar to **Face Edges** before selecting the face.
Pick Split Location	Appears when you select **Split Die Plates**. Lets you select a location on one or more die plates for a split operation. An alternate method is to specify the gap and the plate lengths or widths.
Plate Length/Width	Appears when you select **Split Die Plates** or **Adjust Length of Die Plates**. Lets you specify the length or width of the selected plates.
Gap	Appears when you select **Split Die Plates** or **Align Die Plates**. Lets you specify the gap value between plates.
 **Reverse gap Direction**	Appears when **Align Die Plates** or **User Defined** is selected. Lets you reverse the direction of the gap between plates.
Pick Location	Appears when you select **Adjust Length of Die Plates**. Lets you select one or more die plates.
Settings	
Appears when **Type** is set to **Design Die Base**.	
Load Die Base Only (Don't Update Parameters)	Loads the die base without updating any parameters. You can use this option to reuse a die base by saving it as a die base template.
Rename Components	Opens the **Part Name Management** dialog box where you can rename components before you complete a **Split Die Plates** operation. Lets you rename a component before loading the die base. You can also use this option to specify the components referenced within Teamcenter.

h. Die Design Setting

h1. Function and command call

Use the **Die Design Setting** command to specify the values of die parameters, which may be used later in downstream design.

Examples of die design parameters include the following:

STRIP_LIFT_HEIGHT	Specifies the metal-strip-lifted height when the die set is open.
PUNCH_PENETRATION	Specifies the depth of the piercing punch that goes into the piercing die.

The **Read Default Value From** customer default specifies whether the default values for die parameters come from their customer default values or from the PDW_VAR part.

Tip To find a customer default, choose **File→Utilities→Customer Defaults**, and click **Find Default** .

Downstream design gets the default values from the PDW_VAR part.

Where do I find it?

Application	Progressive Die Wizard
Command FInder	**Die Design Setting**

h2. Modify the value of a die design setting

Choose **Progressive Die Wizard** tab→**Main** gallery→**Die Design Setting** .

In the **Die Design Setting** dialog box, in the parameters table, double-click the parameter that you want to modify.

The image in the dialog box provides an illustration of the selected parameter.

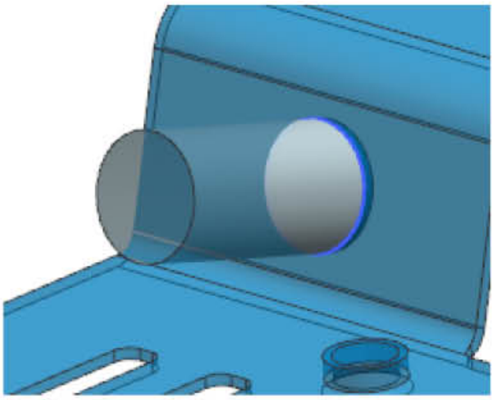
In the **Value** column, enter the new value.

Click **OK** or **Apply**.

i. Special Piercing Insert

i1. Function and command call

Use the **Special Piercing** command to create and edit a special punch for angular and side piercing within your die design.



Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Special Piercing**

i2. Special Piercing Insert dialog box

Type	
Type list	Lets you specify the type of operation you want to perform. **Create** Lets you to create a new insert in the current assembly. **Edit** Lets you edit an existing insert in the current assembly. **Delete** Lets you delete an insert from the current assembly.
The following options depend on the **Type** selected.	
Select Edge	
Select Edge	Lets you to select one or more edges to be used to create the insert.
Select Hole Face	Lets you to select a face in the hole of the selected edges to be used to create the insert.
Select Insert	

Select Insert	Lets you to select the insert to edit, when **Type** is set to **Edit**. Lets you to select the insert to delete, when **Type** is set to **Delete**.
Specify Vector	
Specify Vector	Lets you specify the vector of the piercing insert.
Specify Orientation	Lets you specify the orientation of the piercing insert.
Special Piercing Insert	
Parent	Lets you select a parent part for the insert.
Insert Type	Lets you select the type of insert to create. **Punch** **Die**
Select Standard Blank	Lets you select a standard blank for your insert.
Edit Standard Blank	
Edit Standard Blank	Lets you select a standard blank for editing.
Limits	
Start	Lets you specify the start distance of the insert travel.
End	Lets you specify the end distance of the insert travel.
Clearance	
Lets you specify individual clearance for each plate.	**Punch Plate (PP)** **Bottoming Plate (BP)** **Stripper Plate (SP)**

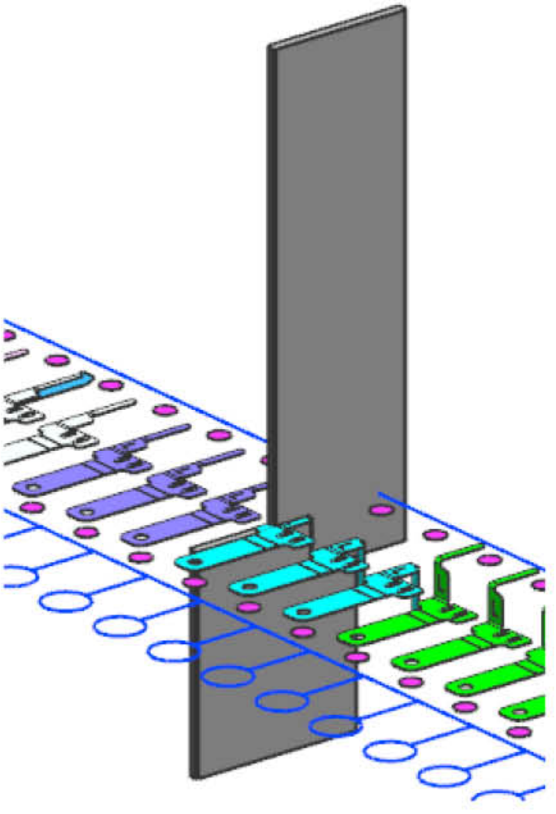
Cavity and Slug Hole	
Appears when the **Insert Type** is set to **Die**.	
Cavity Type	Lets you specify the type of cavity hole in a die base.  **Taper Angle** Creates a **Taper Angle** cavity.  **Step** Creates a **Step** cavity. H - Depth of hole A - Diameter of the die clearance (Step, Round Step 1) Angle of the die clearance (Taper Angle) C1 - Diameter of the slug hole in the top backing plate C2 - Diameter of the slug hole in the bottom backing plate
Clearance	
Appears when the **Insert Type** is set to **Die**.	
Die Plate (DP)	Lets you specify a clearance value for the die in the die plate.
Setting	
Clearance	Appears when **Insert Type** is set to **Die**. Lets you specify a clearance value.
Offset Side	Lets you specify whether the offset side is the die side or the punch side.

j. Bending inserts

j1. Function and command call

Use the **Bending Insert Design** command to add a bending insert group that forms one of the sheet metal folds in your product. As the progressive die moves each blank to the station where the insert is located, the bending punch folds the sheet metal of the blank against the bending die. If your product requires more sheet metal folds, you can add bending inserts at other stations.

You can design your own bending punch and die. In the Progressive Die Wizard application, you can also select a standard part from a catalog.

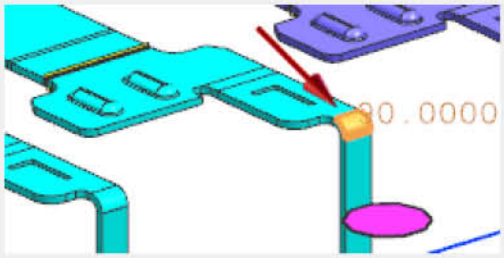


Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Bending Insert Design**

j2. Bending Insert Design dialog box

Type		
Type list	Standard Insert	Lets you select a bending punch or die from a catalog and add it to a station in your die design.
	User Defined	Lets you design a customized bending punch or die for a station in your die design.
	Delete	Lets you select one or more bending insert groups and delete them from the die design.
Standard Insert		
Appears when **Type** is set to **Standard Insert**.		

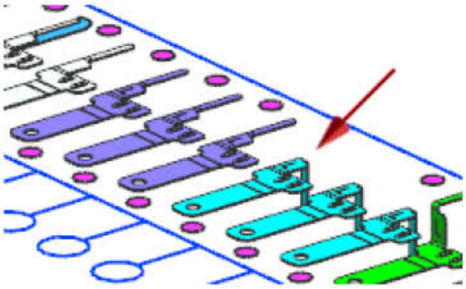
Select Bend Start Face	Lets you select one or more faces to be bent by the insert. 
Bending Type	Lets you specify the bending shape by clicking one of the following icons. The icons differ depending on whether the bending direction is up or down. **90 Degree Bending** Up, Down **Angle Bending** Up, Down **Z Bending** Up, Down **V Bending** Up, Down **Universal Z Bending** Up, Down
Insert Type	Lets you specify whether you are adding a bending punch or a bending die.
Standard Insert	Lets you select a standard bending punch or bending die to add to your design. You are offered default choices that can be used with your start face selection and your specifications for the **Bending Type** and **Insert Type** options.
User Defined	
Appears when **Type** is set to **User Defined**.	
Select Bend Region Faces	Lets you select one or more faces to be bent by the insert.
Insert Type	Specifies whether the insert is a bending punch or a bending die.
Parent	Lets you specify a parent node for the bending insert. You can review the assembly structure in the **Assembly Navigator**.

	In the Engineering Die Wizard application, the parent is the station where the insert is located.
Position	Lets you control the bending insert height by specifying which plate is associated to the insert height.
Sketch options	Lets you do one of the following: Select an existing loop of curves to define the shape and size of the insert. Use the sketch options to create a loop of curves (usually a rectangle). You can create a datum plane on which you can sketch the curves. If you want to create a composite insert, you should select or create two or more loops.
Extend Distance	Lets you control the trimming of the bending insert by specifying how far NX extends the bend region faces to create the trim.
 **Select Insert to Edit**	Lets you edit an existing bending punch or die.
Clearance	
Appears when **Type** is set to **User Defined**.	
Plates	Lets you define the clearance between the insert and the pocket that is created for the insert in each plate.
Settings	
Appears when **Type** is set to **User Defined**.	
Rename	Lets you name the insert.
Concept Design	Specifies that NX should design the bending insert without creating any solid bodies.
Without FALSE Body	Specifies that NX should install the insert without creating a pocket in the die plates.
Composite Insert	Specifies that the bending insert consists of multiple inserts. When you sketch the insert, you should draw two or more loops to define it

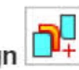
j3. Create a user defined bending insert

This example shows how to create a user defined bending insert, which consists of a bending punch and a bending die.

Together, the punch and die will produce the first bend in your strip layout.



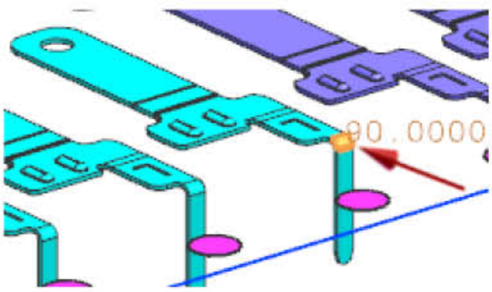
Create a user defined bending punch

Choose **Progressive Die Wizard** tab→**Main** gallery→**Bending Insert Design** .

In the **Bending Insert Design** dialog box, from the **Type** list, choose **User Defined**.

In the graphics window, select the bend region face.

NX displays the angle measurement beside the selected face. This example uses a 90 degree angle for the bend.



In the **Bending Insert Design** dialog box, in the **User Defined** group, define the following options.

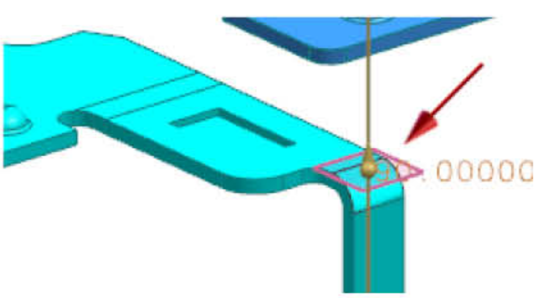
Insert Type = Punch

Parent = prj_db_008

Position = Punch Plate

Click **Create Datum Plane** .

The datum plane is created at the center of the selected bend region face. You will use this datum plane to sketch the bending punch.

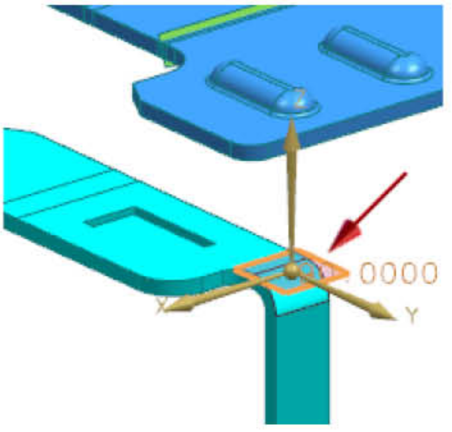


Sketch the curves to define the bending punch.

Click **Sketch Section** .

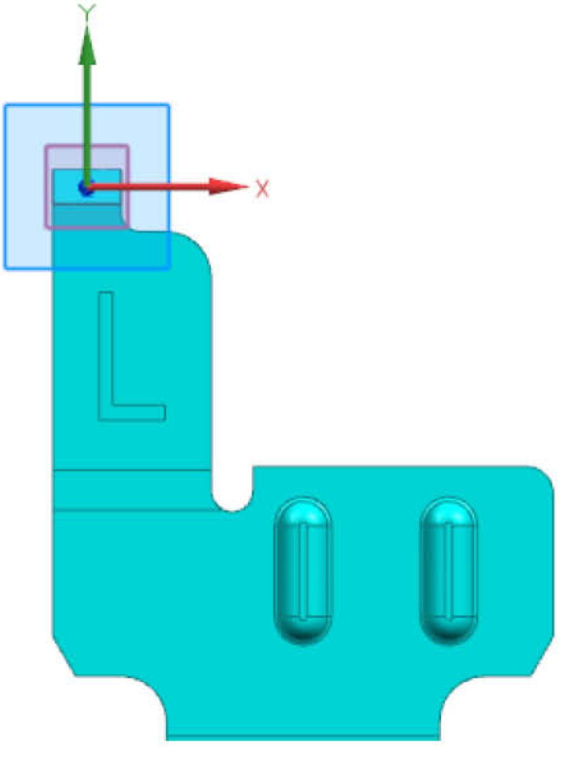
The **Create Sketch** dialog box is displayed, and NX prompts you to select the sketch plane.

In the graphics window, select the datum plane.

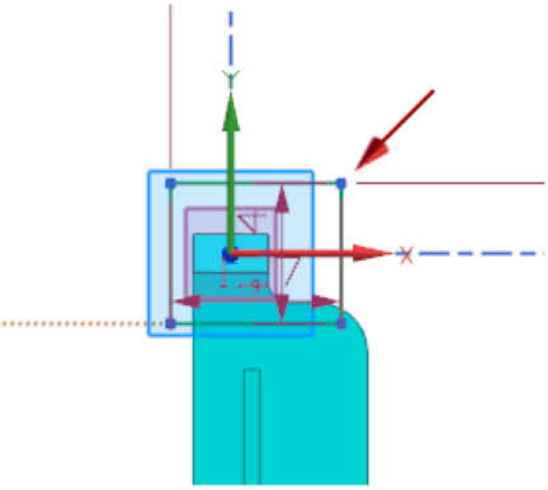


In the **Create Sketch** dialog box, click **OK**.

NX enters the sketch task environment.

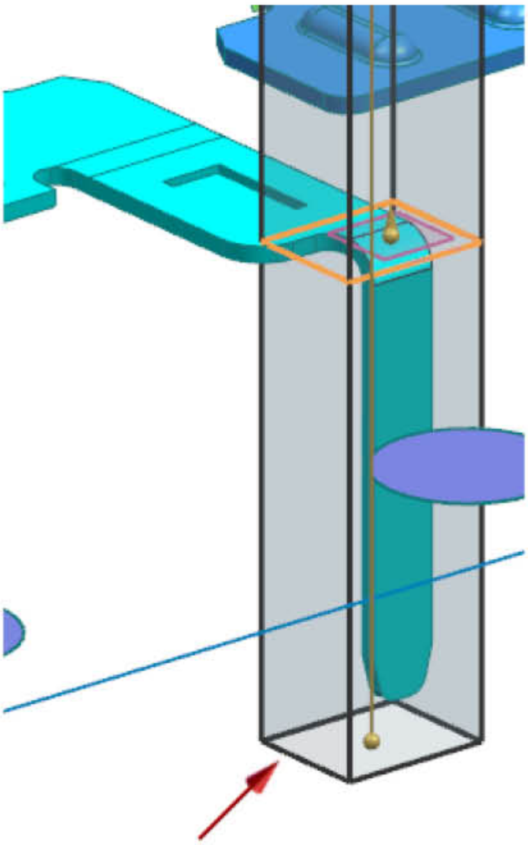


Sketch a rectangle to define the size and shape of the bending punch.



Click **Finish Sketch** .

NX exits the sketch task environment and displays a preview of the bending punch.



(Optional) Define other parameters for the bending punch.

In the **Bending Insert Design** dialog box, in the **User Defined** group, set **Extend Distance** to **1 mm**.

In the **Clearance** group, set the following clearances.

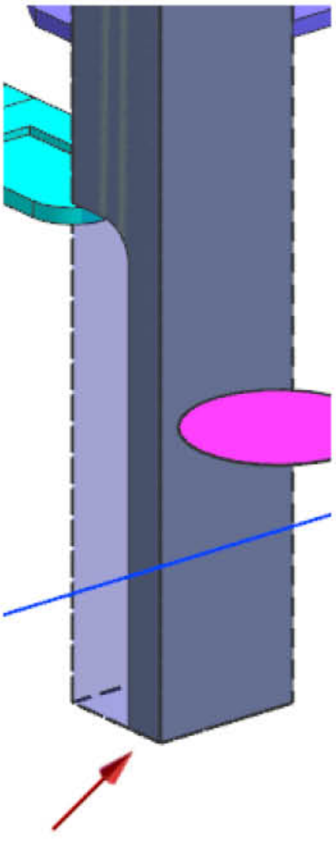
Punch Plate = 0.2

Bottoming Plate = 0.2

Stripper Plate = 0.2

Click **Apply**.

The bending punch is created.

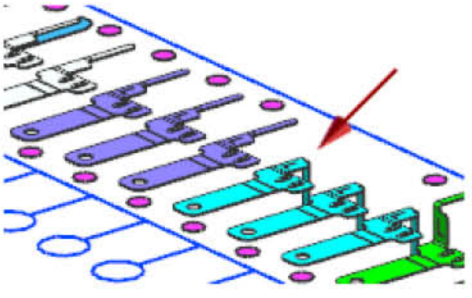


Now you want to create the corresponding bending die.

Add a standard bending insert

This example shows how to add a standard bending punch and bending die to a die design. A standard bending insert is an existing punch or die that you can order from a catalog.

Together, the standard punch and die will produce the first bend in your strip layout.



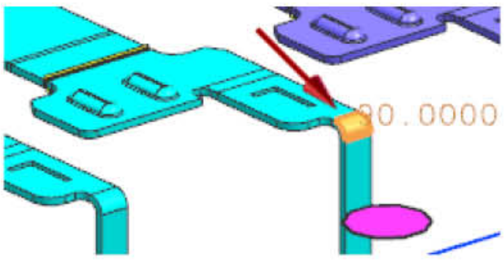
j4. Create a bending punch

Choose **Progressive Die Wizard** tab→**Main** gallery→**Bending Insert Design** , or choose **Menu**→**Tools**→**Process Specific**→**Progressive Die Wizard**→**Bending Insert Design**.

In the **Bending Insert Design** dialog box, from the **Type** list, choose **Standard Insert**.

In the graphics window, select the bend start face.

NX displays the angle measurement beside the selected face. This example uses a 90 degree angle for the bend.



In the **Bending Insert Design** dialog box, in the **Standard Insert** group, under **Bending Type**, click **90 Degree Bending** .

Under **Insert Type**, select **Punch**.

Click **Standard Insert** .

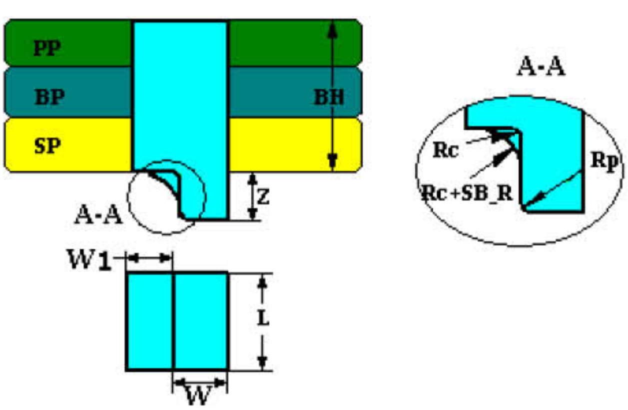
The **Standard Part Management** dialog box is displayed.

In the **Folder View** group, the **PDW Bending Insert Library**→**Straight_Down**→**punch** directory is highlighted.

In the **Member View** group, a table lists the types of 90-degree bending punches that are available.

In the **Member View** group, in the table, select the **Bend Down Punch[Without Screw, With Springback Radius]** row.

The **Information** window shows a drawing of the selected punch type.

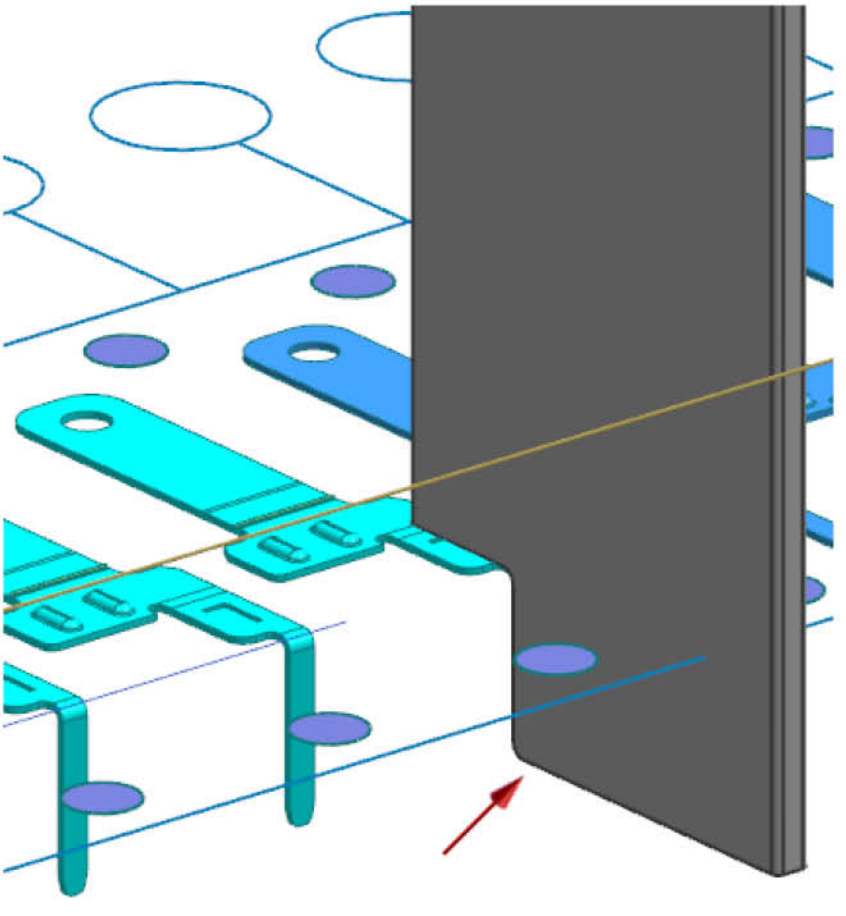


Based on your inputs, NX selects a default punch from the directory of the punch type you selected. The parameters of the default punch appear in the **Standard Part Management** dialog box, in the **Details** group. You can select a different standard punch by clicking the **Value** column of the parameter that you want to change.

For this example, you want to accept the default punch.

Click **OK**.

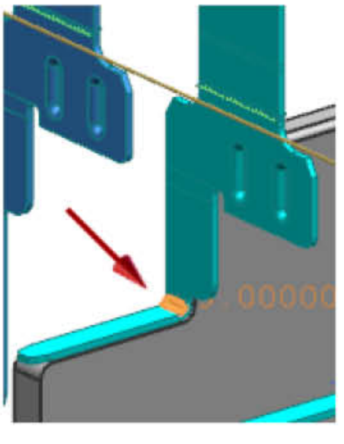
NX adds the bending punch to your progressive die design.



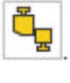
You have created the bending punch. Now you need to create the corresponding bending die.

Create a bending die

In the graphics window, select the bend start face for the die.



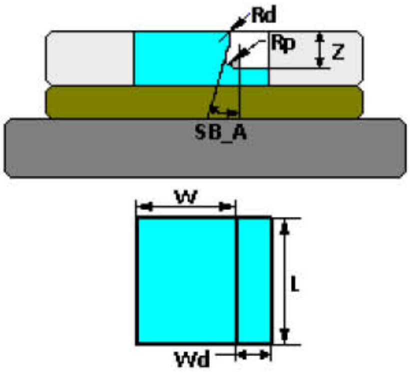
In the **Bending Insert Design** dialog box, in the **Standard Insert** group, under **Insert Type**, select **Die**.

Click **Standard Insert** .

In the **Standard Part Management** dialog box, in the **Folder View** group, the **PDW Bending Insert Library→Straight_Down→die** directory is highlighted.

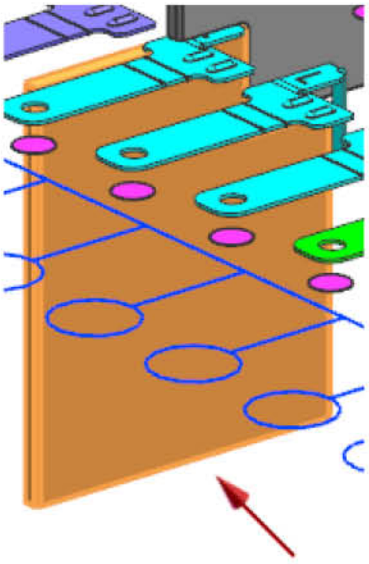
In the **Member View** group, in the table, select the **BDDIA[Bend down insert, Without Screw, With Springback Angle]** row.

The **Information** window shows the following drawing, and the parameters for the default die appear in the **Details** group.

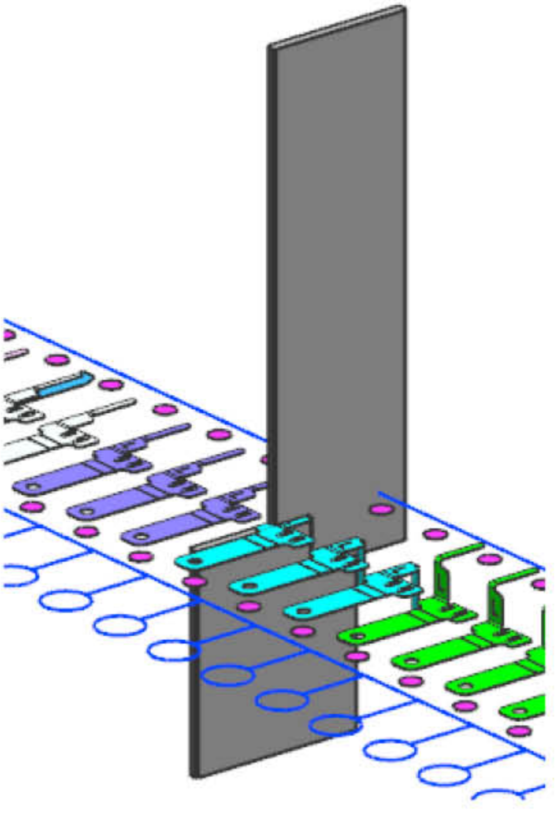


Click **OK**.

NX adds the bending die to your progressive die design.

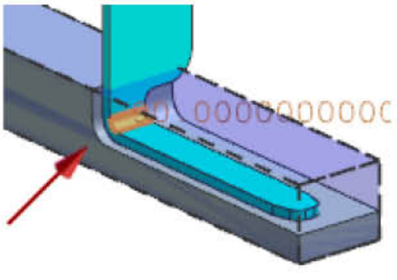


The following picture shows the complete bending insert.



Create a user defined bending die

In the graphics window, select the bend region face.



In the **Bending Insert Design** dialog box, in the **User Defined** group, under **Insert Type**, select **Die**.

Click **Create Datum Plane** .

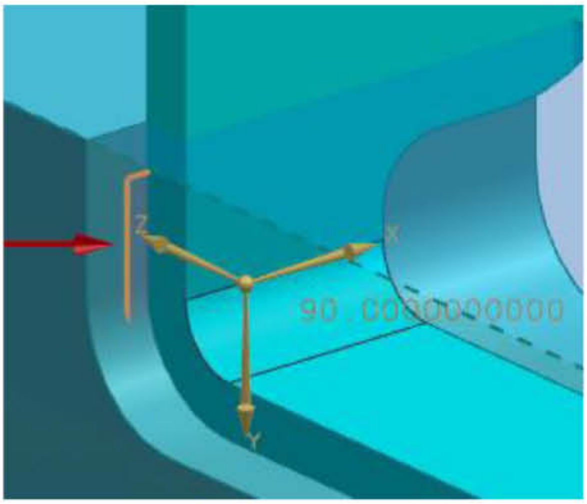
The datum plane is created.

Sketch the curves to define the bending die.

Click **Sketch Section** .

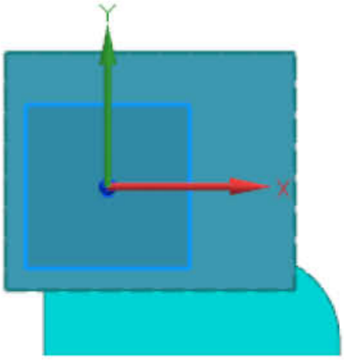
The **Create Sketch** dialog box appears, and NX prompts you to select the sketch plane.

In the graphics window, select the datum plane.

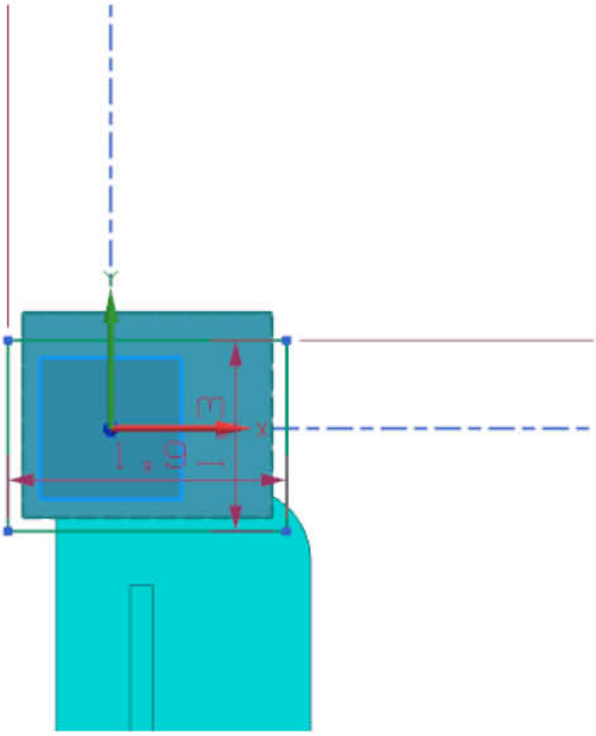


In the **Create Sketch** dialog box, click **OK**.

NX enters the sketch task environment.

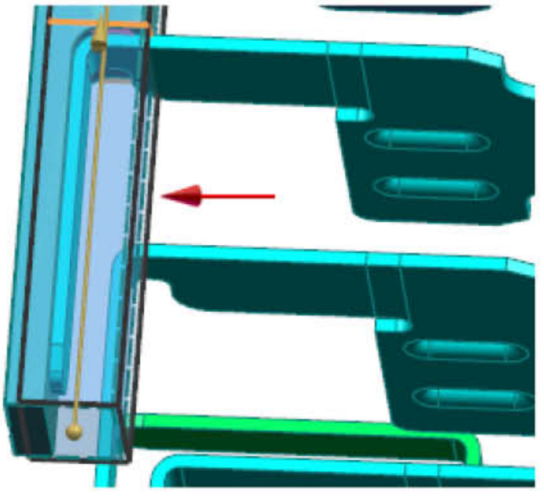


Sketch a rectangle to define the size and shape of the bending die.



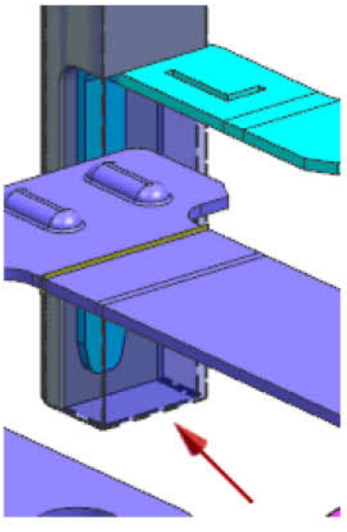
Click **Finish Sketch**.

NX exits the sketch task environment and displays a preview of the bending die.



In the **Bending Insert Design** dialog box, click **OK**.

The bending insert die is created.



k. Forming Insert Design

k1. Function and command call

Use the **Forming Insert Design** command to create a forming punch and die. You can:

Specify any forming region or free-form region for purposes such as bending or embossing.

Load a sketch to define any shape for a forming blank.

Specify a clearance for the punch plate, bottoming plate, stripper plate, or die plate.

Automatically create the forming head area and unit with the forming blank when you click **Apply**.

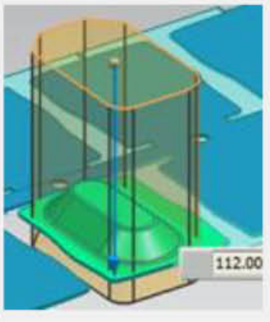
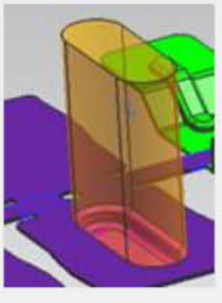
Where do I find it?

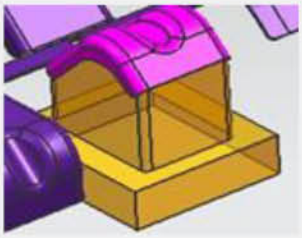
Application	Progressive Die Wizard
Command Finder	**Forming Insert Design** 

k2. Forming Insert Design dialog box

Note See Common dialog box options for common options not discussed here.

Select Forming Region	
You can select the forming region by two methods: selecting the faces directly, or defining the region by selecting seed faces and boundary faces.	
Use Seed Face and Boundary Faces options	Appears when you select the **Use Seed Face and Boundary Faces** check box. Lets you select forming faces by selecting only the faces necessary to define the region. NX determines the other faces based on your selections and **Region Options** settings. Boundary faces specify the ends of the forming region, and seed faces define the location of the region (that is, which side of the boundary face the region is on). Tip This method is useful when you have to select small faces or a large number of faces. **Preview Region** highlights the faces in the forming region as currently defined.
 **Select Forming Faces**	Appears when you clear the **Use Seed Face and Boundary Faces** check box. Lets you select forming faces in the graphics window.
Create Forming Blank	
Design Forming Punch	Specifies the type of blank.

Design Forming Die	
Parent	Lets you specify the parent part. You can accept the default or select a new parent from the list. In the Engineering Die Wizard application, the parent is the station where the insert is located.
Blank Type	Lets you select the type of blank to create.
 **Select Blank**	Appears when the **Standard** blank type is selected. Opens the **Standard Part Management** dialog box allowing you to select forming inserts for your die design. Note To make insert selection easier, only forming insert options from the **PDW Forming Insert Library** are displayed.
Position	Specifies where the punch or die is located: A forming punch can be in the punch plate or the stripper plate. A forming die must be in the die plate.
Trim by Forming Region Directly	Specifies whether the forming blank is trimmed directly by the forming region.  The forming region must be large enough to trim the extrude body, and the extrude body must be long enough to trim. If the **Trim by Forming Region Directly** check box is not selected, project the forming region to the datum plane if there is no extrude body.  If there is an extrude body, project the forming region to the extrude solid body.

	
profile options	Lets you sketch or select curves to define the punch or die profile. **Create Datum Plane** creates a datum plane where you can sketch curves. **Sketch Section** enters the sketch environment, where you can sketch the profile on the datum plane. **Curve** lets you select existing curves for the profile.
Height	Specifies the height of the punch or die.
Clearance options	Lets you specify clearances for the following plates: Punch plate (PP), bottoming plate (BP), and stripper plate (SP) when you select **Design Forming Punch**. Die plate (DP) when you select **Design Forming Die**.
Select Punch or Die to Edit	Lets you edit an existing punch or die.
Delete Punch or Die Component	Deletes the selected punch or die.

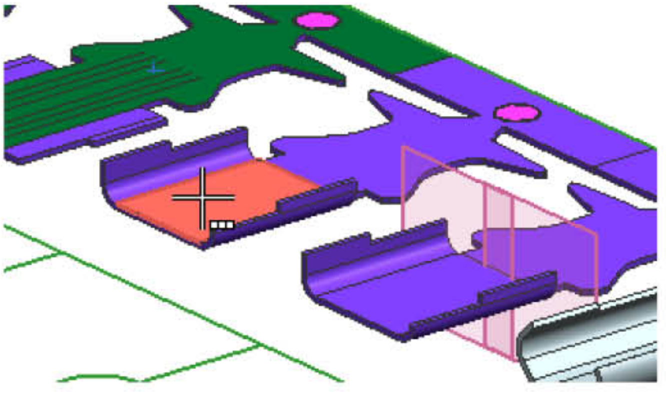
k3. Create a forming punch

This example shows how to create a forming insert in a die project.

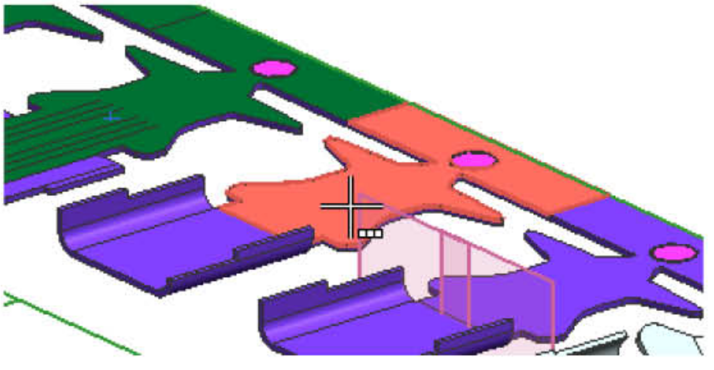
Choose **Progressive Die Wizard** tab→**Main** gallery→**Forming Insert Design** .

In the **Forming Insert Design** dialog box, in the **Select Forming Region** group, select the **Use Seed Face and Boundary Faces** check box.

In the graphics window, select a seed face.



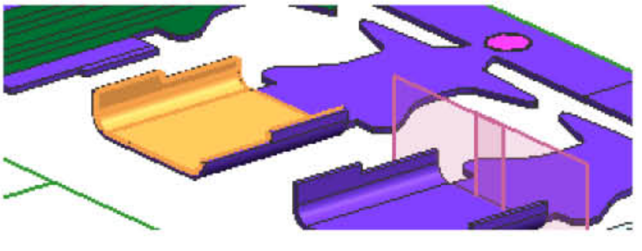
Select a boundary face.



In the **Region Options** group, select the **Traverse Interior Edges** and **Use Tangent Edge Angle** check boxes.

Click the **Preview Region** button.

The highlighted faces are the ones you want in the forming region.



Click the **Finished Preview** button.

In the **Create Forming Blank** group, select the **Design Forming Punch** option.

From the **Position** list, select **Stripper Plate**.

Sketch a profile for the forming punch by doing the following:

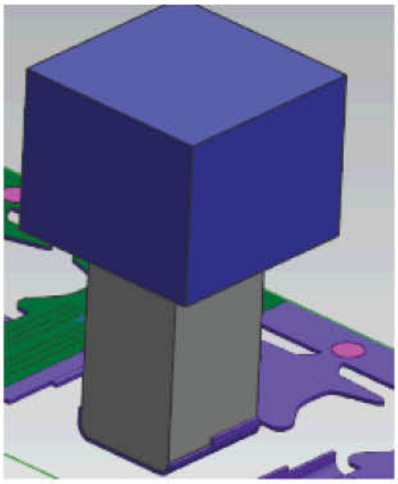
Click **Create Datum Plane** , and create a datum plane where you can sketch the profile for the forming punch.

Click **Sketch Section** .

In the sketch task environment, on the datum plane, sketch a rectangle around the selected faces.

Click **Finish Sketch**.

The initial forming punch appears.



(Optional) Modify the initial forming punch as needed. For example, you can do any of the following:

In the **Height** box, specify the height of the forming punch.

In the **Clearance** group, in the **Stripper Plate** box, specify the clearance between the forming punch and the stripper plate.

For a progressive die, you can use the **Replace Face** command to reshape the sides of the forming punch to match the shape of the forming region.

In the **Forming Insert Design** dialog box, click **OK**.

The forming punch is created.

You can now create a forming die to shape the following faces on the bottom of the part. Create the forming die in the same way as the forming punch, except in step 8, select the **Design Forming Die** option

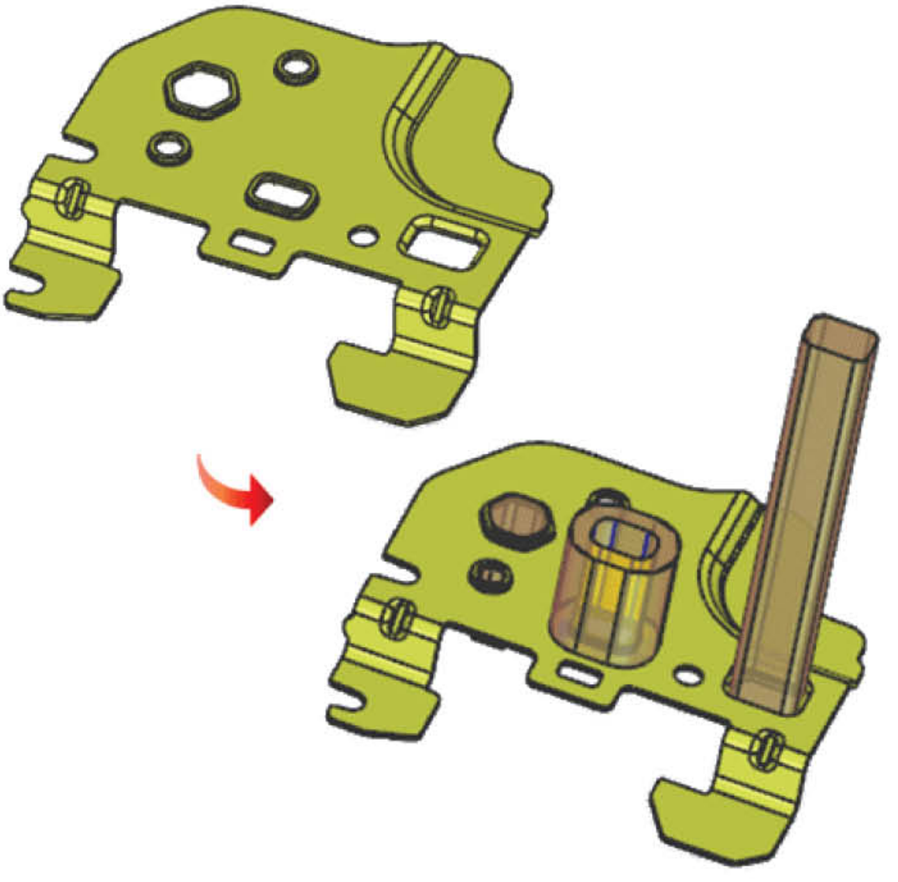
I. Burring Insert Design

I1. Function and command call

You can now design:

A punch and die to create non-circular burring holes in the product.

A punch for burring holes without defining a blank.



Non-circular burring holes are very common in electronic parts. Use this enhancement to generate such burring holes.

Where do I find it?

Application	Progressive Die Wizard and Engineering Die Wizard
Command Finder	**Burring Insert Design** 
Location in dialog box	**Burring Insert Design** dialog box→**Burring Insert** group→**Circular** or **Non-circular** **Burring Insert Design** dialog box→**Burring Insert** group→**Create without blank**

I2. Burring Insert Design dialog box

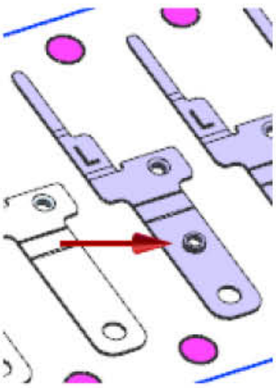
Burring Insert	
Select Burring Face	Lets you select a cylindrical face for the burring insert. For a burring punch, you should select an internal face. For a burring die, you should select an external face.
Burring Direction	Lets you specify whether the burring direction is up or down.
Insert Type	Lets you specify whether to add a burring punch or a burring die. The options in the dialog box vary, depending on which **Insert Type** you choose.
Circular or Non-circular	Lets you specify whether the burring hole is circular or non-circular.
Parent	Lets you specify the parent assembly for the burring insert. You can review the assembly structure in the **Assembly Navigator**.
The following options appear when you set **Insert Type** to **Punch**.	
Select Standard Blank	Opens the **Standard Part Management** dialog box, where you can select a standard burring punch to add to your design.
Punch head options	Lets you specify the shape and size of the punch head. The **Punch Head Type** list lets you choose a shape for the punch head. A drawing of the punch head type is displayed below this option in the dialog box. Depending on which type you choose, you can define other parameters such as the **Punch Head Arc Radius**.
The following options appear when **Insert Type** is set to **Die**.	
Select Curves	Lets you do one of the following: **Curve** — Select an existing loop of curves to define the shape and size of the insert. **Sketch Section** — Use the sketch options to create a loop of curves. You can create a datum plane on which you can sketch the curves (usually a rectangle).
The following options appear for both **Insert Type** settings.	
Parameters	Lets you specify other parameters for the burring punch or die.

	The start and end height of the insert The clearance between the insert and the plate that contains the pocket where the insert is installed. (Optional) If you are creating a burring die, you can specify that you want to provide a name by selecting the **Rename Component** check box in the **Settings** group

I3. Add a burring insert to a die design

This example shows how to add a burring insert in the up direction to a die design.

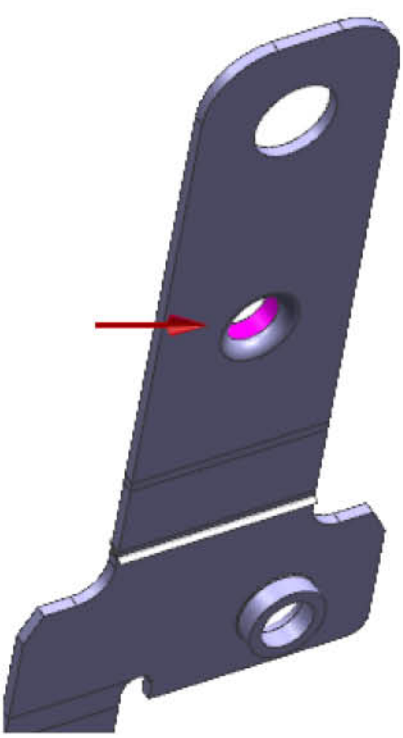
Because the burring is in the up direction, the punch will be below the strip, and the die will be above the strip.



Create a burring punch

Choose **Progressive Die Wizard** tab→**Main** gallery→**Burring Insert Design** .

In the graphics window, select the internal cylindrical face of the burr.



In the **Burring Insert Design** dialog box, under **Burring Direction**, select **Up**.

Under **Insert Type**, select **Punch**.

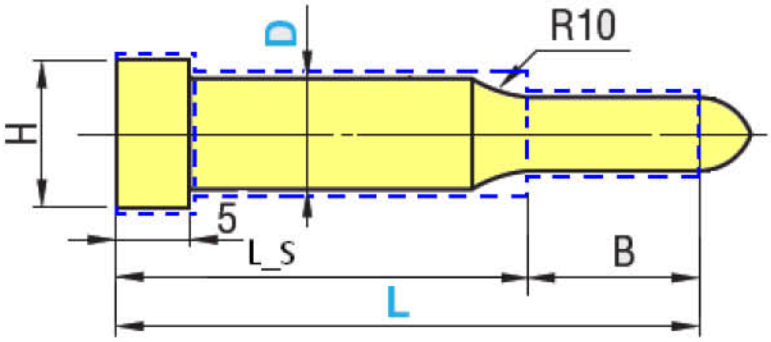
Add a standard burring punch.

Click **Select Standard Blank** .

The **Standard Part Management** dialog box is displayed.

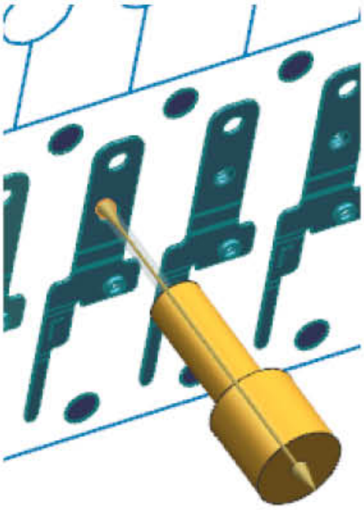
The parameters of the default burring punch that NX selects appear in the **Details** group, and the **Information** window displays a diagram of the punch and its parameters.

Note If the current standard punch does not meet your needs, change the value for a parameter by clicking its **Value** column. Select a new value from the list if values are provided, or enter a new value.

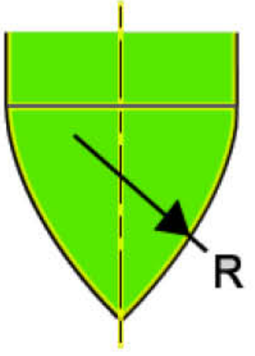


Click **OK**.

A preview of the burring punch is displayed in your die design.



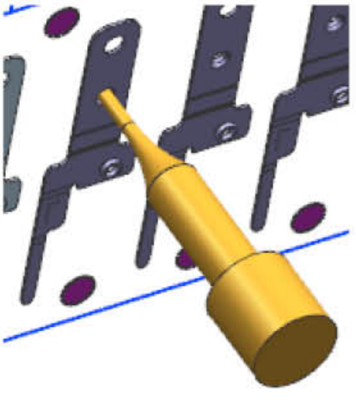
In the **Burring Insert Design** dialog box, from the **Punch Head Type** list, choose **Type 1**.



In the **Punch Head Arc Radius** box, enter **0.4**.

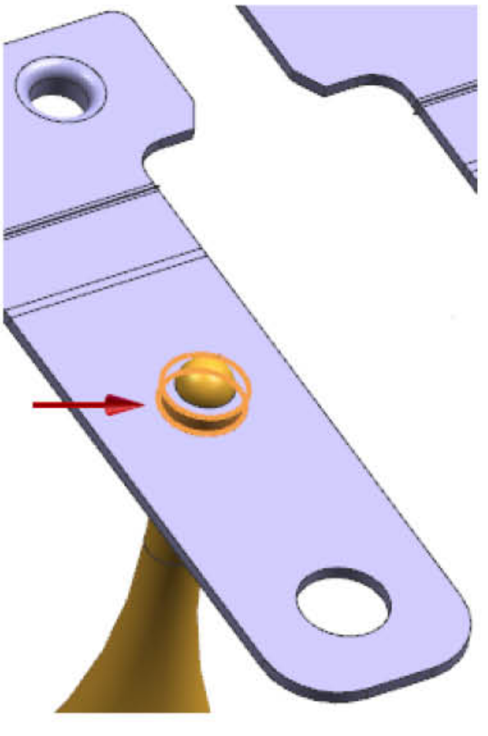
Click **Apply**.

The burring punch is created.



Create a burring die

In the graphics window, select the external cylindrical face of the burr.

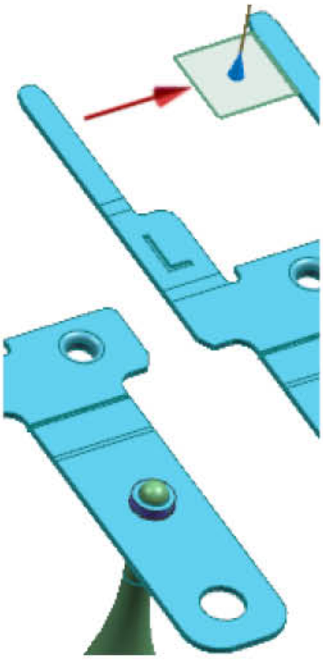


In the **Burring Insert Design** dialog box, under **Insert Type**, select **Die**.

Create sketch curves to define the boundaries of the burring die insert.

Click **Create Datum Plane** .

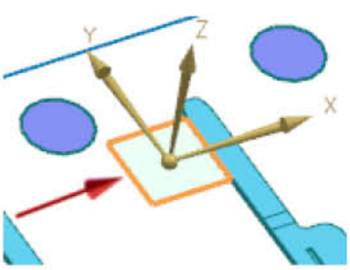
A datum plane that defines the sketch plane is created in the top plane of the strip layout.



Click **Sketch Section** .

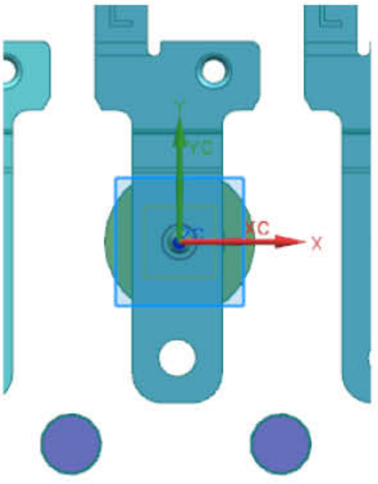
The **Create Sketch** dialog box is displayed.

In the graphics window, select the datum plane.

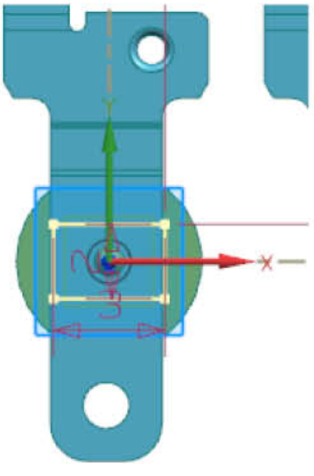


In the **Create Sketch** dialog box, click **OK**.

You are now in the sketch task environment.

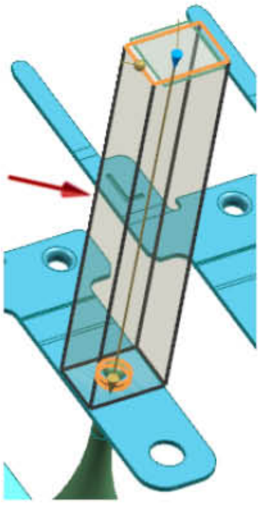


Sketch a rectangle to define the burring die.



Click **Finish Sketch** .

A preview of the die is created.



In the **Burring Insert Design** dialog box, click **OK**.

The burring die is created.

m. Insert Auxiliary Design

m1. Function and command call

Use the **Insert Auxiliary Design** command to:

Add a shank to strengthen any punch or die insert. The shank can have a heel, ramp, or flange shape.

Specify a clearance for the punch plate, bottoming plate, or stripper plate.

Create or edit a punch mount.

Copy or delete inserts.

Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Insert Auxiliary Design**

m2. Insert Auxiliary Design dialog box

Type	
Type list	Lets you specify the type of operation.
Insert Shank	
Appears when **Type** is set to **Insert Shank**.	
Shape options	Lets you specify whether the shank shape is a flange, ramp, or heel. You can create shanks as needed to strengthen punch and die inserts.
The following options appear when **Shape** is set to **Flange**.	
Flange shank options	Lets you create and edit a flange shank. The picture shows the parameters of the flange shank, which you can edit. Tip If you do not see all the parameters in the picture, drag the right edge of the dialog box to expand it. The **Clearance** check boxes let you define the clearance for the punch plate, bottoming plate, and stripper plate. **Select Punch Edge** lets you select a punch edge to place the shank. **Select Punch Shank Face** lets you select a shank face, to edit it, for example.
The following options appear when **Shape** is set to **Ramp**.	

ramp shank options	Lets you create or edit a ramp shank. **Select Punch** lets you select a punch where the ramp shank will be created. **Select Shank Profile** lets you select curves to define the shank profile. If the curves do not exist, lets you enter the sketch task environment to sketch the curves. The parameter boxes let you edit the dimensions of the shank. See the picture for definitions of each parameter. Tip If you do not see all the parameters in the picture, drag the right edge of the dialog box to expand it. **Select Face for Ramp** lets you select a face to place the ramp shank. The other ramp options let you select additional geometry or enter values to finish the ramp definition. **Select Punch Shank Face** lets you select a face of a ramp shank, fro example, to edit it.
The following options appear when **Shape** is set to **Heel**.	
heel shank options parameters	Let you create or edit a heel shank. The picture shows the parameters of the flange shank, which you can edit. Tip If you do not see all the parameters in the picture, drag the right edge of the dialog box to expand it. **Select Insert Edge** lets you select an insert edge to place the shank. **Select Insert Shank Face** lets you select a heel shank face, for example, to edit it.
Punch Mount	
Appears when **Type** is set to **Punch Mount**.	
Select Punch Edge	Lets you select an edge to specify the location of the punch mount.
Design Mount Head	Opens the **Standard Part Management** dialog box, where you can select a punch mount.

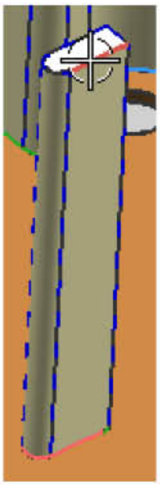
m3. Insert a flange shank

Choose **Progressive Die Wizard** tab→**Main** gallery→**Insert Auxiliary Design** .

In the **Insert Auxiliary Design** dialog box, set **Type** to **Insert Shank**.

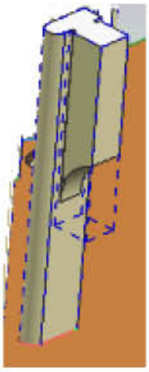
In the **Insert Shank** group, from the list, select **Flange**.

In the graphics window, select a punch edge.



Click **Apply**.

The flange shank is created.



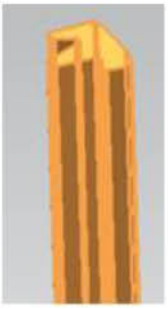
m4. Insert a ramp shank

Choose **Progressive Die Wizard** tab→**Main** gallery→**Insert Auxiliary Design** .

In the **Insert Auxiliary Design** dialog box, set **Type** to **Insert Shank**.

In the **Insert Shank** group, from the list, select **Ramp**.

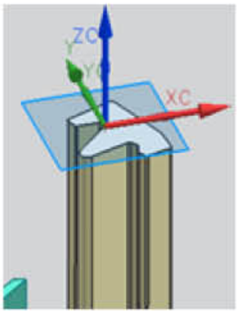
In the graphics window, select a punch.



Sketch the profile of the shank.

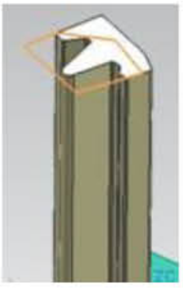
Click **Sketch Section** .

Select the top of the punch as the sketch plane.



In the **Create Sketch** dialog box, click **OK**.

In the sketch environment, sketch the following profile.



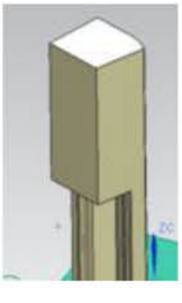
Click **Finish Sketch**.

In the **Insert Auxiliary Design** dialog box, in the **Insert Shank** group, in the **H** box, enter **-30**.

In the **F_H** box, enter **-32**.

Click **Apply**.

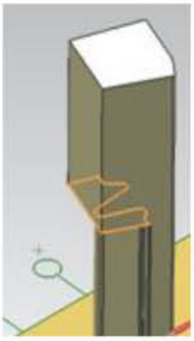
The initial ramp shank is created.



Create the ramp for the shank.

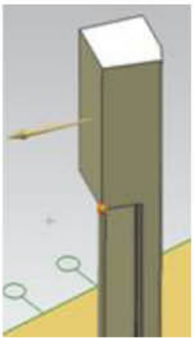
Click **Select Face for Ramp**.

Select the following face.



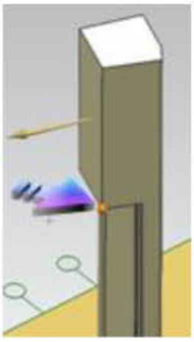
Click **Specify Vector for Ramp**.

Select the following vector.



Click **Specify Point for Ramp**.

Select the following point.



In the **Radius for Ramp** box, enter **1**.

Click **OK**.

The ramp is added to the ramp shank.

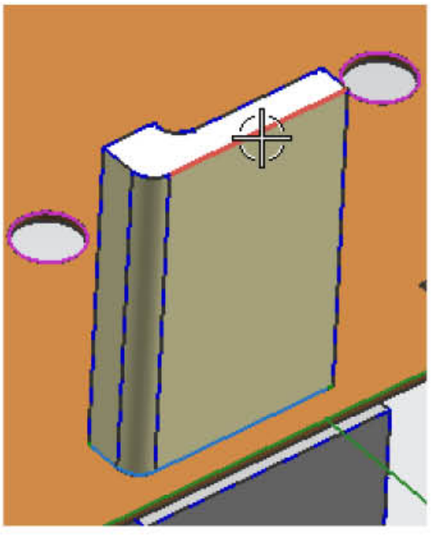
m5. Insert a heel shank

Choose **Progressive Die Wizard** tab→**Main** gallery→**Insert Auxiliary Design** .

In the **Insert Auxiliary Design** dialog box, set **Type** to **Insert Shank**.

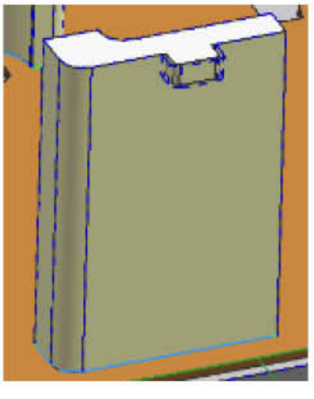
In the **Insert Shank** group, from the list, select **Heel**.

In the graphics window, select a punch edge.



Click **Apply**.

The heel shank is created.

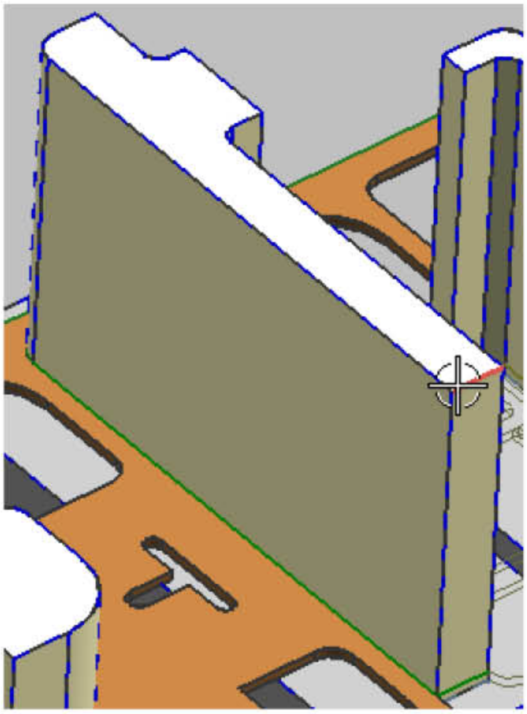


m6. Create a punch mount

Choose **Progressive Die Wizard** tab→**Main** gallery→**Insert Auxiliary Design** .

In the **Insert Auxiliary Design** dialog box, set **Type** to **Punch Mount**.

In the graphics window, select a punch edge.



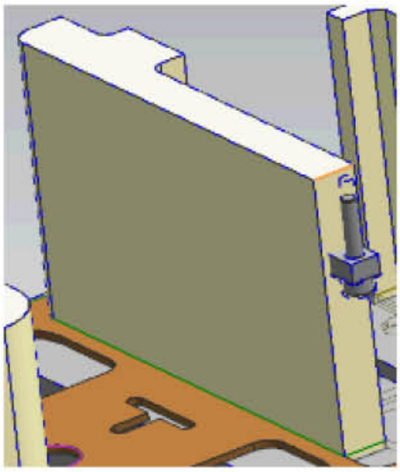
Click **Design Mount Head** .

In the **Standard Part Management** dialog box, from the parts list, select **Screwholder (Bottom)**.

Click **OK**.

In the **Insert Auxiliary Design** dialog box, click **OK**.

The punch mount is added to the punch.



n. Standard parts

n1. Function and command call

Mold Wizard, Progressive Die Wizard, Engineering Die Wizard, and Die Design provide several libraries of standard parts that you can use to create your design.

The standard part libraries available provide a wide selection of commonly used components that you can install and adjust, such as bushings, ejector pins, and inserts.

You can customize your libraries to match your company's design standards and expand the library to include any component or assembly.

You can add standard parts to your mold or die assembly in one of two ways:

Using the **Reuse Library** on the Resource bar. This is the preferred method.

Using the **Standard Part Library** command.

n2. Standard Part Management system

The Standard Part Management system consists of libraries of parts and supporting database spreadsheets.

You can use the Standard Part Management system to:

Register components in a new or existing catalog.

Install new components or instances of existing components.

Position standard parts in the mold or die assembly.

Edit database spreadsheets to control optional parameter choices and attributes for parts lists.

Use database spreadsheets to define interpart expressions to link components and mold bases.

Remove components.

Application	Die Design, Mold Wizard, Progressive Die Wizard, Engineering Die Wizard
Prerequisite	To work with Die Design commands, you must be in the Modeling application.
Resource Bar	**Reuse Library** →**MW Standard Part Library** or **PDW Standard Part Library**, or **EDW Standard Part Library**, or **Stamping Die Standard Part Library**.
Command Finder	**Standard Parts**

n3. Install a standard part

Prepare a library for use

Before you attempt to install a standard part, you should ensure that the library is set up:

Install the engineering library. It is supplied separately from NX.

In the Assemblies customer defaults, under **General**, on the **Interpart Modeling** page, under **Allow Associative Interpart Modeling**, select **Yes**.

On the same page, ensure that the **Delay interpart Update** check box is deselected.

(Optional) In the Mold Wizard, Progressive Die Wizard, or Engineering Die Wizard customer defaults, under **Standard Parts**, review the options and customize them. For Die Design, the **Standard Parts** customer defaults are in the **Standard Tools** section.

(Optional) Use the **Partially Shaded** rendering style. Typically, standard parts are shaded in this mode, and other objects are not shaded.

From Reuse Library, select **Reuse Library Management** , and click **OK** in the dialog box.

This step insures that the Reuse Library configuration file is properly loaded.

n4. Select and install a standard part

On the Resource Bar, click **Reuse Library** .

In the **Name** panel, select the library that contains the catalog you want, and expand it.

Navigate to the manufacturer's catalog or a custom catalog you created and expand it.

Select the folder for the type of reusable object you want to add.

Bitmaps of the parts in the folder are displayed in the **Member Select** panel.

If you do not see the parts displayed, select the **Thumbnails** from the view style list, and **View All** from the view filter list.

In the **Member Select** panel, select a part, right-click and choose **Insert**.

You can also select the part, hold down the left mouse button and drag the part into the graphics window. For this to work, you must have the Assemblies application open.

The **Standard Part Management** dialog box displays. An **Information** window that includes a bitmap of the selected part with dimensions and catalog information also displays. You can toggle the display of this window by clicking  in the **Part** group.

In the **Standard Part Management** dialog box, in the **Part** group, specify whether you want to add another instance or add a new component based on the selected instance.

In the **Placement** group, you can specify:

The parent for the standard part

A positioning method

A reference set

In the **Details** group, specify parameters for the part, such as length, diameter, radius, and taper.

Select the row for the parameter you want to modify.

In the **Value** column, click and select a value from the list.

Use options in the **Settings** group to open the register or database file for editing.

Click **OK** or **Apply**.

If the part requires positioning, specify the position information as required for the method you are using.

When you place a part as a child of a part that has two or more instances, you must specify a the placement point in the instance that is the work part

n5. Install a standard part using Point Pattern

To use **Point Pattern** with standard parts, you must first select the **Point Pattern** and modify it to fit your requirements. Then you select a standard part (such as a screw), and use the point pattern you created as your positioning method.

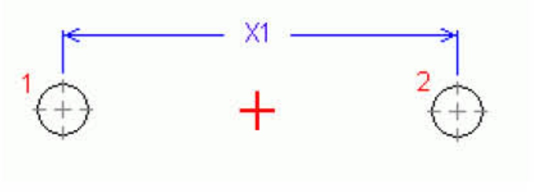
This example uses a simple part to illustrate the **Point Pattern** method.

On the Resource Bar, click **Reuse Library** .

In the **Name** panel, select and expand the **MW Standard Part Library**.

Expand the POINT PATTERN (MM or INCH) catalog. Select a Point Pattern, right-click and select **Insert**.

The **Standard Part Management** dialog box opens, and the **Information** window displays the default pattern (2A). Note the location of the red **Pattern Point** in the middle of the pattern.

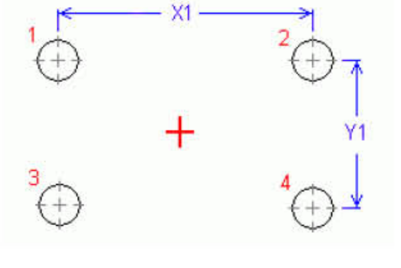


Change the pattern and modify some of its parameters.

This example uses the following settings:

Position = NULL

Pattern 4A



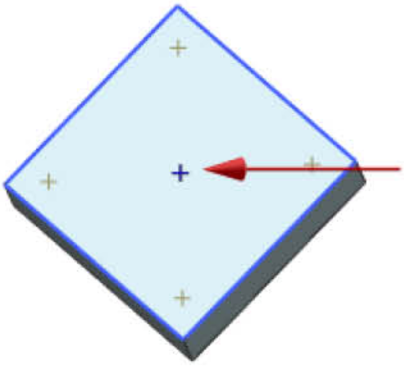
X1 = 75

Y1 = 75

Click **OK**.

The Point Pattern is now listed in the **Assembly Navigator**. It should also be visible in the graphics window. If you do not see it, make Layer 99 visible. It is the default layer for Point Pattern features.

The red arrow in the following graphic indicates the Pattern Point that you will need to select in a later step.



Repeat steps 1–2 and select the standard part you want to position with the pattern you just created.

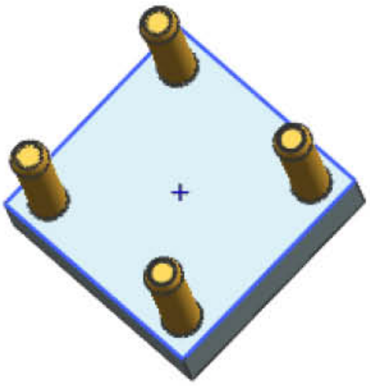
This example uses a PUNCH angle pin bolt.

Select the bolt and click **Insert**.

In the **Standard Part Management** dialog box in the **Placement** group, from the **Position** list, select **POINT PATTERN**.

Select the Pattern Point shown in step 5, then click **OK**.

The pattern of bolts is created.

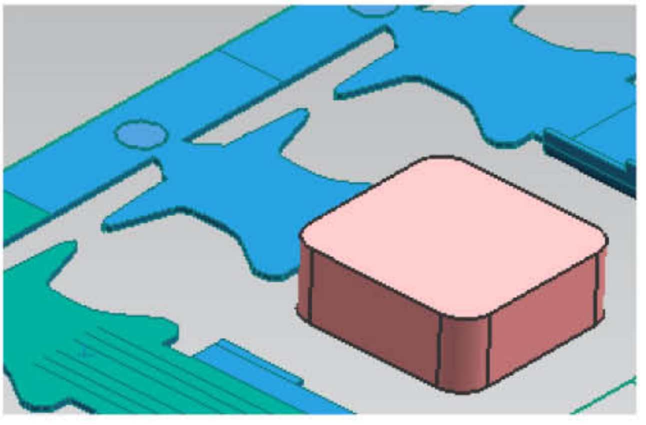


o. Relief Design

o1. Function and command call

Use the **Relief Design** command to create a solid body to cut out the pockets and holes on die plates. This prevents interference between the strip and the die plates.

You can also edit, copy, or delete a relief solid body.



Relief solid body — bounding-box type

Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Relief Design** .

o2. Relief Design dialog box

Note See Common dialog box options for common options not discussed here.

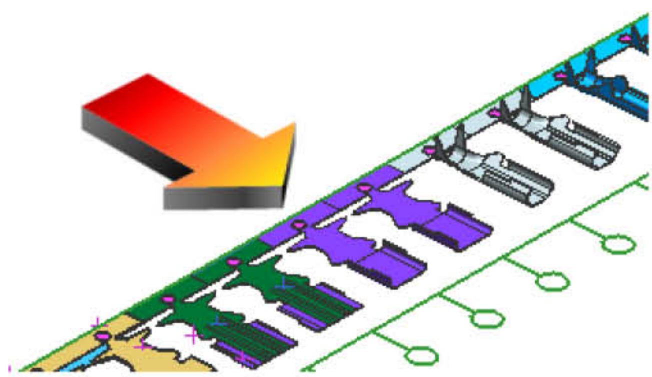
Type	
Type menu	Lets you select the operation to perform: **Create** **Edit** **Copy** **Delete**
Create	
The **Create** group appears when you set **Type** to **Create**.	
Bounding Box **User Defined**	Specifies whether the relief body shape is determined by a bounding box or by user defined inputs.

Select Face	Lets you select the extreme faces of the area where the relief is required.
bounding box options	Lets you specify the following when you select the **Bounding Box** option: **Clearance** The distance between the bounding box and the selected faces **Radius** The radius of the bounding box corners
user defined options	Lets you specify the user defined shape for the relief solid body. You can select curves to define the shape, and specify the start distance and height of the body.
Edit	
The **Edit** group appears when you set **Type** to **Edit**.	
Select Relief Solid	Lets you select a relief solid body to edit.
bounding box options	Lets you edit the bounding box that defines the relief solid body. You can change the radius, or you can use **Edit Bounding Box** to edit other parameters.
user defined options	Lets you edit the start distance or height of a user defined relief solid body.
Copy	
The **Copy** group appears when you set **Type** to **Copy**.	
Select Relief Solid	Lets you select a relief solid body to copy.
Number	Specifies the number of copies.
Delete	
The **Delete** group appears when you set **Type** to **Delete**.	

 **Select Relief Solid**	Lets you select a relief solid body to delete.
Delete Selected Instance **Delete All Instances**	Specifies how many relief bodies to delete.
Setting	
Create in Original Location	Creates the solid in the original station. Note Relief bodies are put in a *"*_relief"* part. If you want to generate a relief pocket, use the **Pocket Design** command in the "*Solid+Solid* " mode.
Hide Relief Solid Body	Hides the relief solid body

o3. Create a relief design using a bounding box

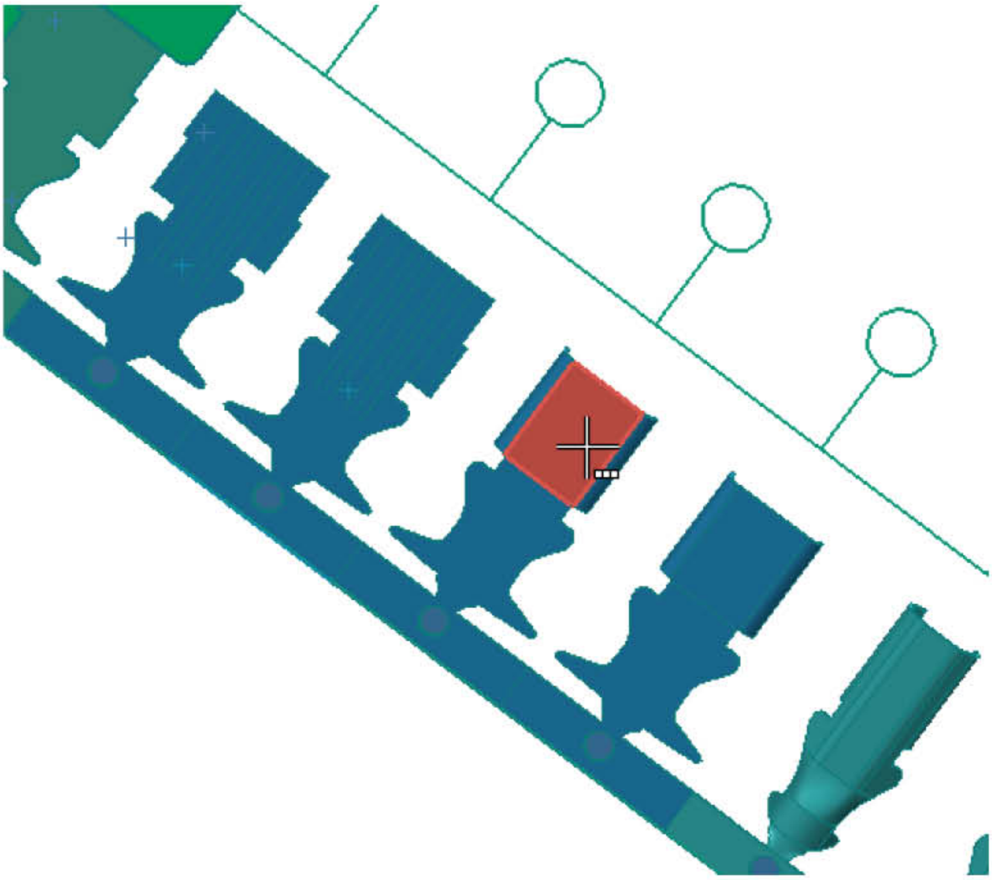
This example shows how to add a bounding-box type of relief solid body to a strip station.

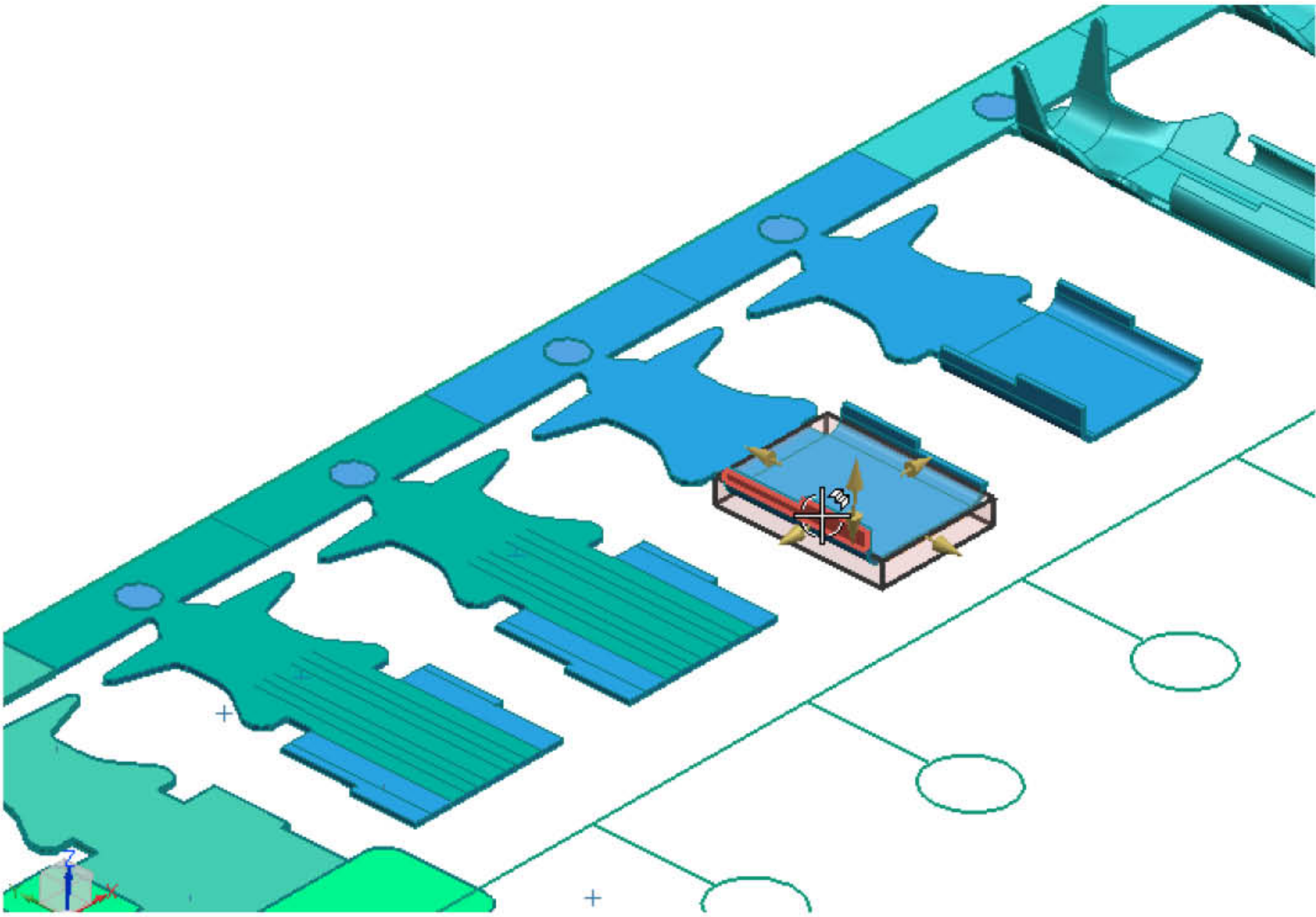


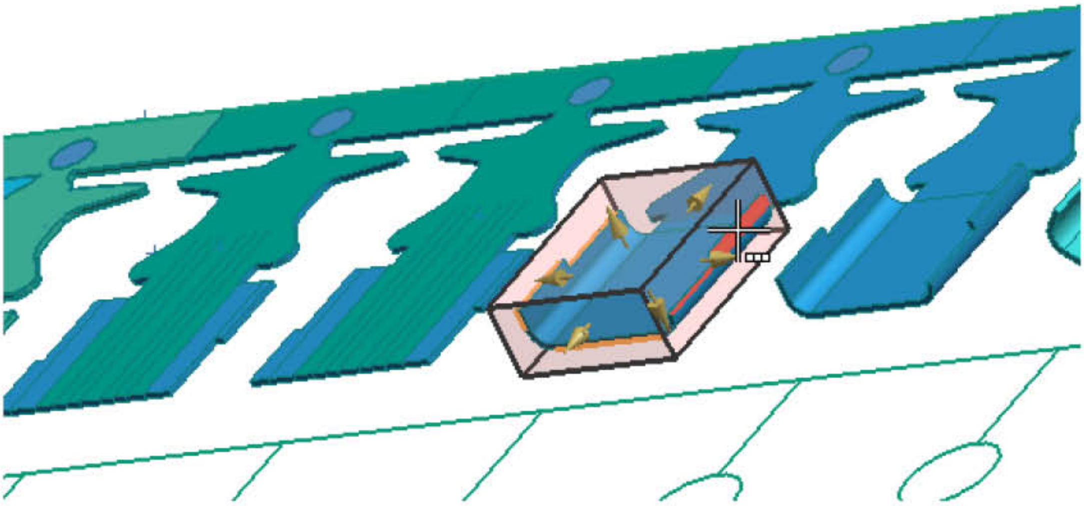
Choose **Progressive Die Wizard** tab→**Main** gallery→**Relief Design** .

In the **Relief Design** dialog box, in the **Create** group, select the **Bounding Box** option.

In the graphics window, select faces to define the boundaries of the relief area.

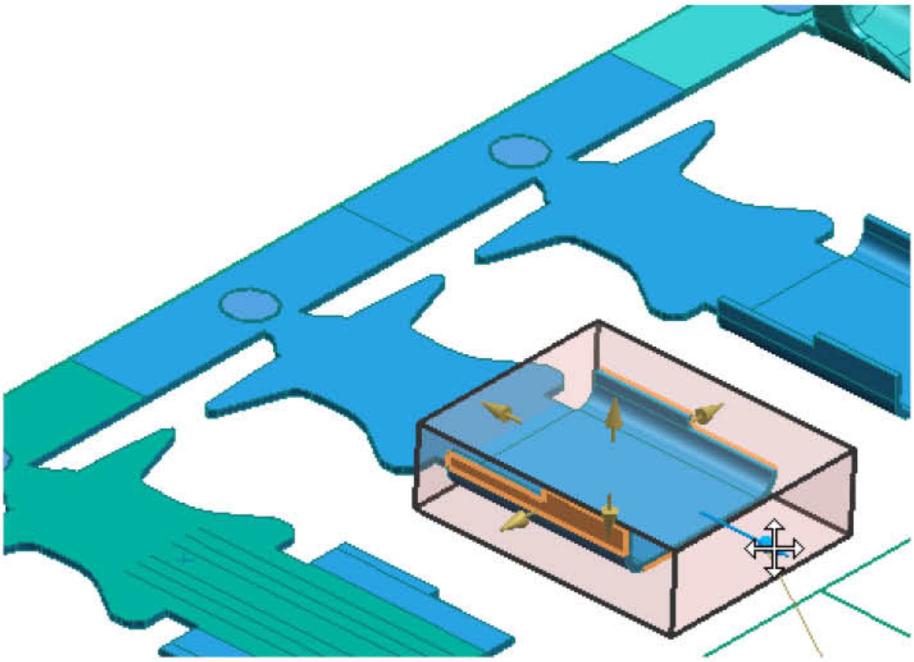






(Optional) Do any of the following:

In the graphics window, drag the handles of the bounding box to modify its size.



In the **Relief Design** dialog box, modify the values in the **Clearance** or **Radius** boxes.

Select or clear the **Create in Original Location** check box to specify the location of the bounding box.

Select or clear the **Hide Relief Solid Body** check box to specify the visibility of the relief solid body.

Click **OK**.

p. Pocket

p1. Function and command call

After you have finished adding standard parts and other components, you can use the **Pocket** command to model pockets in die or mold plates, inserts, and other solid bodies.

Pockets are associative to the tool body by default. You can choose to make them non-associative to improve performance.

When you create pockets, a tool body is geometry linked in the part containing the target body, and the linked body is subtracted from or united with the target body.

You can create pockets using part family members as tool components. Part family pocket bodies are defined in the **PDW Pocket Tool Body Library**.

You can specify a default reference set name for the pocket in **Customer Defaults→Progressive Die Wizard→Pocketing→Custom Reference Set of Tool Bodies**.

You can also:

Select a component pattern to be used as a tool component.

Edit the pocket tool body.

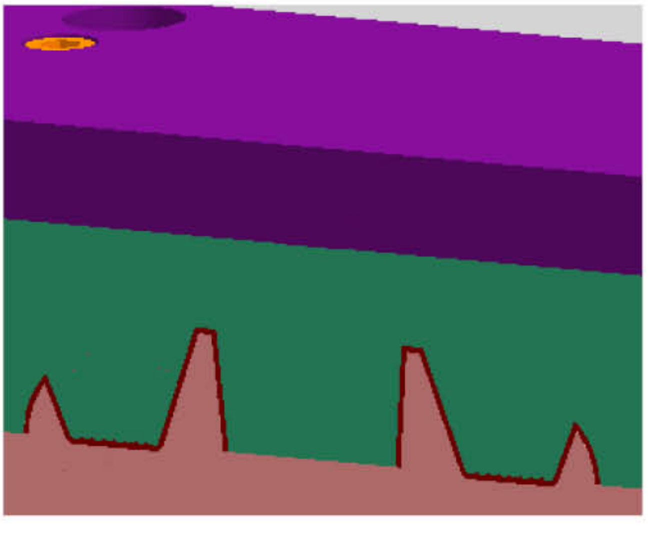
Preview part family tool bodies.

When to Create Pockets

To minimize the number of features that update as you design your tooling assembly, create pockets at the last stage of design:

After you cut pockets, the number of features in the assembly dramatically increases, which can affect system performance.

Some updates may fail if insert groups/standard parts are moved outside of their target body.



Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Pocket**

p2. Pocket dialog box

Mode	
List	**Subtract Material** — In body selection mode, subtract a copy of each tool body from the target(s). In part selection mode, subtract a copy of the FALSE reference set of each part from the target(s). **Add Material** — In body selection mode, unite a copy of each tool body with the target(s). In part selection mode, unite a copy of the ADD_MATERIAL reference set of each part with the target(s).
Target	
Select Object	Select one or more target bodies.
Tool	
Select Object	Select one or more tool parts, components, or bodies. If the selected component's parent part has **UM_STANDARD_PART_ROOT = 1** attribute, the parent part will be automatically selected.
Tool Type	**Solid** — Filters the **Select Object** command in the **Tool** group to select only solids. **Component** — Filters the **Select Object** command in the **Tool** group to select only components. **Show Shortcuts** — Replaces the **Select Types** list with the **Solid** and **Part** buttons. **Hide Shortcuts** — Replaces the **Select Types Solid** and **Part** buttons with the list.
Reference Set	With **Part** or **Component** selected as the **Tool Type**, you can specify one of the following reference sets: False True Entire Part No Change
Tools	
	Find all target bodies that intersect with selected tool bodies, or find all tool bodies that intersect with selected target bodies.

Find Intersecting Components	
Check Pocket Status	Highlight and show a total count of all standard parts that have no corresponding pockets in the Status line, or give a message that there are no unpocketed bodies.
Remove Pocket	Remove all pockets from the selected target body.
Edit Tool Body	Opens the **Edit Reusable Object** dialog to edit the pocket tool body.
Settings	
Associative	Makes the pocket associative to the tool body.
Show Target and Tool bodies Only	Hides all bodies other than those currently selected as targets and tools.
Create Interfering Solid	In cases where the pocket subtraction fails, creates a solid to highlight the problem area.
Always Save Results in Part	Automatically saves the results of failed pocket checks in the part file.
Show Check Results in HD3D	Displays the check results in the **HD3D** tab on the resource bar.
Preview Tool Body	Displays a preview of the generated pocket tool body for the family standard tool part.

p3. Pocket design procedures

Creating new pockets

Tip Before you create pockets, you can use the **View Manager** dialog box to hide components other than target and tool bodies. This simplifies selection and checking.

You must be in the **Mold Wizard** application. If you are in the **Progressive Die Wizard**, or **Engineering Die Wizard** application, use **Command Finder** to locate the command.

Choose **Mold Wizard** tab→**Main** gallery→**Pocket** .

In the **Mode** group, select **Add Material** or **Subtract Material**.

(Optional) In the **Target** group, click **Select Object** and select one or more target objects.

Caution If you skip this step, you must select tools and then click **Find Intersecting Components**.

(Optional) In the **Tool** group, from the **Select Types** list or shortcut buttons, specify whether to select parts or solid bodies.

Note You can select only parts that have the **UM_STANDARD_PART** attribute.

(Optional) In the **Tool** group, when a part family part is selected as the tool object, the entire component pattern containing the part is selected and added to the tool object list.

(Optional) If you are selecting parts, in the **Tool** group, from the **Reference Sets** list, select one of the available filters to make selection easier.

(Optional) In the **Tool** group, click **Select Object** and select one or more tool objects.

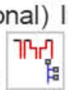
Caution If you skip this step, you must select targets and then click **Find Intersecting Components**.

(Optional) After you select one or more of either targets or tools, in the **Tools** group, click **Find Intersecting Components** .

Note If you selected one or more targets, all potential intersecting standard parts are selected.

If you selected one or more tools, all intersecting plates and other parts that can be pocketed are selected.

Standard parts can be targets, tools, or both, depending on the value of the **UM_STANDARD_PART** file attribute.

(Optional) In the **Tools** group, when selected part family parts are all the same size, the **Edit Tool Body**  command can open the **Edit Reusable Object** dialog to edit the pocket tool body.

(Optional) In the **Tools** group, select the **Show Target and Tool Bodies Only** check box to temporarily hide all bodies except the current selection of targets and tools.

(Optional) In the **Settings** group, select the **Preview Tool Body** option and a preview of the pocket tool bodies for selected family parts is displayed.

Click **OK** or **Apply**.

Deleting pockets

Choose **Mold Wizard** tab→**Main** gallery→**Pocket** .

Select one or more target bodies or tool bodies that are associated with pockets.

In the **Tools** group, click **Remove Pocket** .

Checking pocket status

Choose **Mold Wizard** tab→**Main** gallery→**Pocket** .

In the **Tools** group, click **Check Pocket Status** .

Note If there are no standard parts with no corresponding pocket, you will see an information dialog box advising you that no standard part is unpocketed.

If there are parts without corresponding pockets, they are highlighted. You can see a count of parts without pockets in the Status line

q. Bill of Material

q1. Function and command call

Use the **Bill of Material** command to manage part attributes. The **Bill of Material** command name is commonly referred to by the acronym *BOM.*

Using templates

You can use templates to define the related part information and the following parameters.

Display Name Specifies the display names of the BOM list items, which are displayed as column titles in the **Bill of Material** dialog box.

Attribute Name Specifies the attribute name of the BOM list items. Only the attribute values that you specify in the template are displayed in the BOM list. If you do not specify a value for an attribute, the attribute is not shown in the BOM list.

Key Field Lets you specify whether the corresponding attribute is to be used as a key field attribute or not. The components which have the same key field are grouped together and shown in one row in the BOM list.

Locked Specifies whether you can modify the value of an attribute in the BOM list.

Filter Specifies if you want to use the value of an attribute as a filter string.

For example, if the Display Name attribute has a string value set to SUPPLIER, then the attribute is shown in the BOM list and you can also use the attribute value to filter the BOM data.

If you do not define the Display Name for the filter attributes, for example the MW_COMPONENT_NAME in the template, the attribute is not shown in the BOM UI.

Note The cells shown in gray cannot be used as they will be deleted when you export the BOM. You can enter user information at the top of the template.

Changing values

You can change the value from the predefined alteration list in the specified column by defining a common data template. There are four types of lists in the common data template, and you can define them by setting LIST_TYPE to values as shown in the following table.

LIST_TYPE	Option List	Mark if not in list	Mark if in list (ban list)	User Change
0	Yes	No	No	No

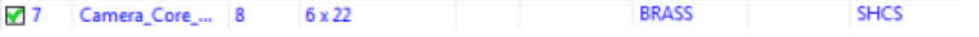
1	Yes	No	No	Yes
2	Yes	Yes	No	No
3	No	No	Yes	Yes

You can also do the following:

Generate different BOM lists by switching templates in the **Bill of Material** dialog box.

Quickly recognize modified information in the BOM list.

If you modify a row in the BOM list, the particular row is highlighted in blue.



Filter BOM data based on specific attributes which are defined in the BOM template.

A check box will flag the rows that you have exported.

These features will generate a BOM with a formatted template in Excel. You can specify different part attributes in different columns of the template, specify one or more part attributes to be a key field, a lock status, or to filter the column.

Editing

You can edit the BOM table similar to how you would edit a spreadsheet. You can:

Select individual cells.

Edit multiple cells simultaneously.

Use better filtering options to filter data.

Use visual cues such as cell color to identify certain types of data. For example, a locked cell appears in gray, and an invalid entry appears in pink.

You can either accept your manually entered BOM data by clicking **Apply** or exit by clicking **Cancel**.

Footer information

You can add footer information to the **Bill of Material** template including general text, or assembly attributes such as date and designer.

Information added to the footer section of the template appears at the bottom of your generated bill of material spreadsheet.

Where do I find it?

Application	Mold Wizard, Progressive Die Wizard, Engineering Die Design, Electrode Design
Command Finder	**Bill of Material**

q2. Bill of Material dialog box

Setting	
Select Components	Lets you select a component in the graphics window. NX then highlights the selection in the BOM list.
List	
Rows represent standard parts in the assembly. Columns represent calculated value for item number and quantity, and part attribute value. You can edit cells in a selected row by clicking them. Note You cannot edit the **NO.** and **QTY** columns. You cannot edit system-assigned attributes such as part names if they appear in your parts list note. You can choose one of the following commands at the bottom of the BOM dialog or right-click rows in the list to get the corresponding commands:	
BOM List	Shows the components that appear in the BOM list.
Hide List	Shows the components that are currently hidden from the BOM list, so that you can use a shortcut command to show them.
Create Parts List	Lets you create a new parts list.
Add Displayed Part	Adds the current displayed part to the BOM list.
Edit Stock Size	Lets you open the **Stock Size** dialog box. For multiple row selections, calculates the stock sizes according to the parameters set in the **Settings** group.
	Opens the **Component Information** dialog box, which has a list showing the name of the component part file.

Information	
Export to Spreadsheet	Creates a spreadsheet file containing the contents of the list. Transfers the BOM list data from NX to a temporary spreadsheet. You can use your default spreadsheet viewer commands to save the temporary file.
Hide Components	Moves the selected component to the **Hide List** display and removes it from the BOM List. Appears only when **BOM List** option is selected.
BOM Filter	Opens the **Filter** dialog box and lets you filter the BOM data by the given filter strings.
Setting	
Name	Generates different BOM lists by switching templates

r. View Manager

r1. Function and command call

Use the **View Manager** command to do the following to progressive die or engineering die components:

Control visibility and selection through check boxes.

Right-click a component node to open or close it, freeze or unfreeze it, isolate it, or set its color.

Note You can use the **View Manager Browser** with other commands in the Progressive Die Wizard and Engineering Die Wizard applications. Start the **View Manager Browser** first, then open the other commands.

The **View Manager Browser** is similar in style to the **Assembly Navigator**, displaying holders like assembly files. Holders look like assembly files, and represent a group of component types. Holders may contain other holders and can be used to organize other holders and nodes in the **View Manager Browser**. Nodes look like component files and represent a group of components of a single type.

View Manager Browser

Customized Name		Isolate	Freeze Status	Open Status	Attribute Count
	Top				0
	Other				0
	Diebase				0
	hide				0
	hole				0
	die / punch				1
	bend				1
	strip				0

The View Manager Browser tree structure

As shown in the previous figure, holders and nodes may have custom names in the **View Manager Browser** that are independent of the component name.

If a node is displayed "Dowel Pins" and all components in the die that contain the part attribute PDW_COMPONENT_NAME or EDW_COMPONENT_NAME with the value "DOWEL" will be managed by the "Dowel Pin" node.

XML code:

<PDW_COMPONENT_NAME Description="Dowel Pin">DOWEL</PDW_COMPONENT_NAME> or <EDW_COMPONENT_NAME Description="Dowel Pin">DOWEL</EDW_COMPONENT_NAME>

Where do I find it?

Application	Progressive Die Wizard
Command FInder	**View Manager**

s. Static Interference Check

s1. Function and command call

Use the **Static Interference Check** command to analyze selected solid bodies for overlap and to report the type of interferences found.

A static interference check helps you improve your design by showing you where interferences or missed pockets occur.

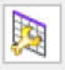
Where do I find it?

Application	Progressive Die Wizard
Command Finder	**Static Interference Check**

s2. Static Interference Check dialog box

User Defined Sets	
Select Object	You are prompted to select a target component or body.
Select Object	You are prompted to select a tool component or body.
Note The following options become unavailable after you select the first object. For any single analysis you can select either bodies (exclusively) or components (exclusively).	
Component	Selects components.
Solid Body	Selects individual bodies in component parts.
Standard Sets	
List	Displays the components from the **static_interference_check_standard_sets.xls** spreadsheet.
Add to User Defined Sets	Adds the selected set to the user defined sets that appear in the list in the **Sets Name List** group.
Sets Name List	
List	Shows the assemblies clearance sets defined in the displayed part.

Clearance Set Name	Defines the current selection as a user defined clearance analysis set, and displays the name in the list in the **Sets Name List** group. Note You can select a saved set for future rechecking without having to select the targets and tools again.
Clearance Zone	Sets the clearance tolerance for soft interference.
Remove	Deletes the selected clearance zone.
Settings	
Analysis Mode	Select one of the following: **Exact** Checks solid bodies for interference. Note This is the more accurate mode, but it can be slow if several components are analyzed. **Lightweight** Compares faceted bodies for interference. Note This mode is less accurate, but it is comparatively very fast.
Reference Set	Select one of the following: **True Body** Selects the actual assembled component model of standard parts for the analysis. **False Body** Selects the pocket cutting body in standard parts for the analysis. **Entire Part** Selects all solid bodies in standard parts for the analysis.
Include Subassemblies	Includes child components of every assembly part you select for analysis.
Include Blanked Bodies	Adds hidden bodies in selected components to the analysis.
	Includes parts that contain a symbolic thread feature in the analysis.

Include Fasteners	
 **Select Standard Sets Spreadsheet**	Lets you select the standard sets spreadsheet file.
 **Edit Standard Sets Spreadsheet**	Lets you edit the standard sets spreadsheet file.

t. Motion Simulation

t1. Preprocess Motion

Use the **Preprocess Motion** command to set and load the kinematic model, and mount the sheet metal or plastic parts of the die or the mold on the kinematic model to prepare your motion simulation data.

You can:

Clone a predefined kinematic model for tooling into your current assembly or to a directory you specify.

Change the kinematic model.

NX displays a message window asking you to confirm whether you want to remove the current kinematic model and add another one. You need to click**Yes**, and then mount the components again and redefine the cams.

Generate control data and import it to a kinematic model according to the die settings. You can also change to different control data and re-read the control data.

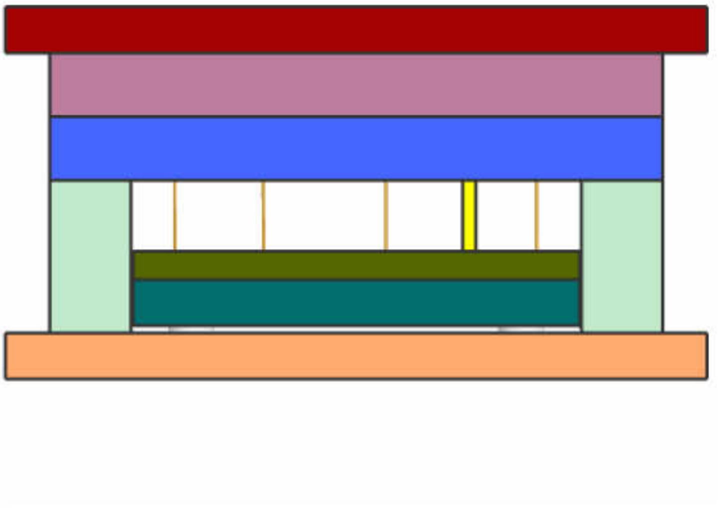
Mount components immediately to the kinematic model by using the **Mount**  command.

You can also specify different motion types for the mounted components.

Update the current simulation by using the **Re-read Configuration Data File**  command.

The check box for this option is removed.

Example of an injection molding cycle.



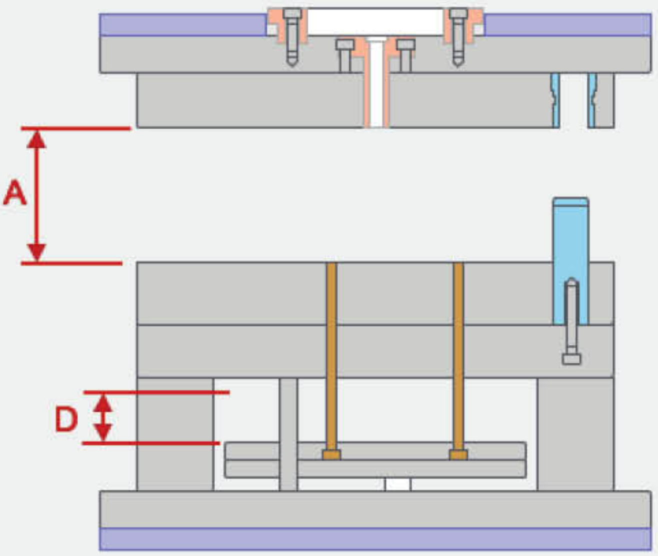
A complete mold assembly or progressive die can have thousands of components. When you simulate the motion of your tooling assembly, you can locate and resolve potential interference problems before the assembly is built.

Where do I find it?

Application	Mold Wizard and Progressive Die Wizard
Command Finder	**Preprocess Motion** 

t2. Preprocess Motion dialog box

Type	
Type list	 **Add Kinematic Model** Clones a new set of kinematic parts and adds them to the current tooling assembly as a subassembly.  **Mount Component** Lets you identify components as belonging to a specific kinematic category, such as fixed or moving components.
The following options and groups appear based on what you select from the Type list.	
Kinematic Model	
Appears when **Type** is set to **Add Kinematic Model**.	
List	Lets you select a kinematic model template. Two templates are provided as examples.
Target Directory	Sets the folder to which the kinematic components are cloned. Typically, you should place the kinematic model in a location that is included in your search paths, or with the tooling assembly.
Kinematic Parameters	
Appears when **Type** is set to **Add Kinematic Model**.	
The following options appear only in Mold Wizard.	
Two-plate Style **Three-plate Style**	Specifies whether the mold assembly has two or three plates. Lets you define the following kinematic parameters for a mold assembly: **Machine Stroke (A)** Sets the injection mold open distance. **Ejection Distance (D)** Sets the distance over which the plastic model is ejected when the mold opens. *Example of a two-plate style mold*



The following parameters appear when **Three-plate Style** is selected.

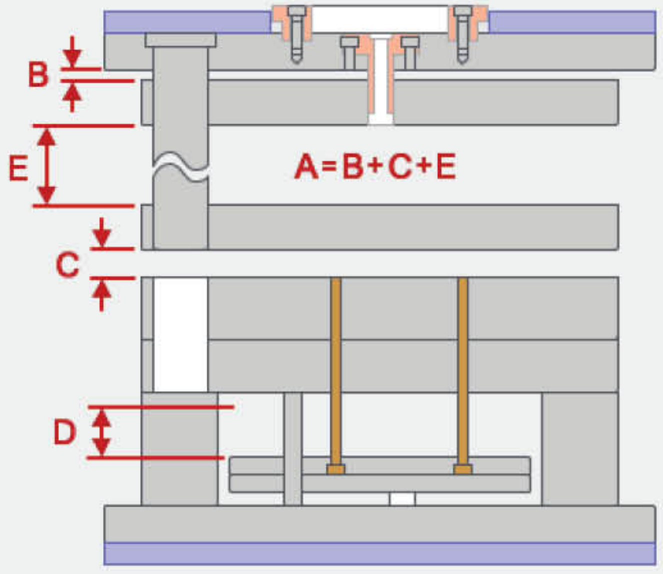
Pull Runner Distance (B)

Sets the distance over which runners are pulled.

Injection Relief Distance (E)

Sets the distance between the runner plate and the cavity plate.

Example of a three-plate style mold



Eject Product	Lets you specify when the ejector plate advances to eject the part from the mold.

<table>
<tr><td></td><td>
<table>
<tr><td>After Mold Fully Opened</td><td>Advances the ejector plate to release the part, only after the mold is fully open.</td></tr>
<tr><td>While Mold Opening</td><td>Advances the ejector plate to release the part as the mold opens.</td></tr>
</table>
<table>
<tr><td>Mold Open Angle</td><td>Available only when Eject Product is set to After Mold Fully Opened.

The mold open angle defines when to open the mold. A mold simulation cycle has a range from 0 to 360, 0 being the start of the simulation and 360 being the end of the simulation.

Note The Mold Open Angle value must be between 0 and 90 only.</td></tr>
<tr><td>Ejection Start Angle</td><td>Lets you specify the point in the 0-360 simulation range in which to activate additional hydraulic motions.</td></tr>
</table>
</td></tr>
<tr><td colspan="2">The following options appear only in Progressive Die Wizard:</td></tr>
<tr><td></td><td>Lets you define the following kinematic parameters for a progressive die assembly or an engineering die assembly:

Machine Stroke (D)

Stripper Travel (A)

Strip Lift Height (C)

Strip Lift Height (Feeder2)

Pitch (Feeder)

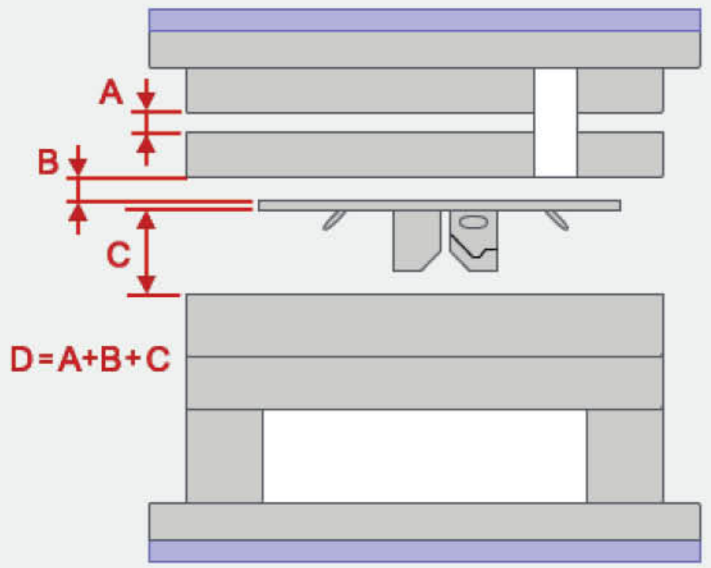
Pitch (Feeder2)

Transfer Start Angle

Transfer End Angle

Strokes per Minute

Example of a progressive die</td></tr>
</table>

	
Settings	
Rename Kinematic Model options	Appears when **Type** is set to **Add Kinematic Model**. ☑ **Rename Components** Sets a suffix that is added to cloned kinematic part names. ☑ **Hide CSYS in Kinematic Model** Hides the CSYS in the kinematic model. When this check box is cleared, the CSYS is displayed at the center of the mold base in the plane where the A-plates and B-plates meet.
☑ **Include Hidden Bodies**	Creates the kinematic model in all bodies. When this check box is cleared, NX does not include the kinematic model in bodies that are hidden.
Import Control Data	Sets a control file to import. The default file is **[pdiewizard_dir]\proj_temp\tooling_validation.xls**.
Export Control Data	Creates a standard **.csv** (Comma Separated Values) file with the current control data.
 **Re-read Configure Data File**	Appears when **Mount Component** is selected in the **Type** group. Reruns the mounting calculation. Click on the icon after you modify the configuration file **tooling_validation.xls**. The file path is**$PDIEWIZARD_DIR\configuration\tooling_validation\.**
	Appears when **Mount Component** is selected in the **Type** group. Opens the **tooling_validation.xls** file.

Edit Configure Data File	
Component Mounting	
Appears when **Type** is set to **Mount Component**.	
Motion	Lets you select one of the motion nodes in the kinematic model. For Mold Wizard, you can choose the following: **Move** This node of the kinematic model is the motion driver. Add plates such as **BP**, **UP**, **CP**, and so on, and other plates and hardware attached to these plates. **Fix** This is the fixed node of the kinematic model. If you reposition it, everything else moves with it. Add fixed parts such as the A-plates. **Ejection** This node of the kinematic model is used to push the product out of the mold. Add parts such as ejectors, lifters, and the ejector plates. **Injection** This node of the kinematic model moves along the Z-axis. **Stripper** **Product** A lifter type used to define motion. This node of the kinematic model first moves along Z-axis and then along the X or Y-axis. Add product components to this node. For Progressive Die Wizard, if you are using the Tooling Kinematic Model template, you can choose from the following: **TOP** **BOTTOM** **LIFTER** **STRIPPER** **FEEDER** For Progressive Die Wizard, if you are using the Tooling Kinematic Model 2 template, you can also choose:

	FEEDER2 LIFTER2 STRIPPER2
Mounted	Specifies the number of components mounted.
Select Object	Lets you select components when you highlight a node in the list. Selected components are added to the node, and included in the simulation. Note Components are added to the node only when you click **Apply** when **Type** is set to **Mount Component**.
Mount	Mounts the selected object into a **Motion** item.
Specify Vector	Appears when you select FEEDER/FEEDER2 motion type in Progressive Die Wizard. Lets you specify a vector for the FEEDER/FEEDER2 move direction

t3. User Defined Motion

Use this command to add user defined motion data to your kinematic model.

You can define:

Linear movement along a specific vector.

Angular movement about a specific axis.

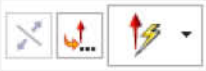
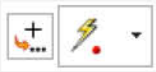
The motion curve is a set of 360 values, one for each angular increment of 0-359 degrees, that determines how far to move along or about the axis at each point of the simulation. You can use the simple motion curves created by NX, or supply your own values using a comma separated values (CSV) file.

Where do I find it?

Application	Mold Wizard and Progressive Die Wizard
Command Finder	**User Defined Motion**

t4. User Defined Motion dialog box

Bodies in Motion	
Select Bodies	Lets you select the bodies for the motion you are defining.

Motion	
Specifies the type of motion that you want to use.	
Linear	A motion that is in a straight line.
Rotary	A motion that revolves around an axis.
 **Specify Vector**	Specifies the direction of motion for the selected bodies.
 **Specify Point**	Appears when the motion is set to **Rotary**. Lets you specify a stationary point around which the selected bodies will rotate..
Motion Curve	
Linear	Sets a linear motion. You define the motion using the timing options. The motion curve is a set of 360 values, one for each angular increment of 0-359 degrees, that determines how far to move along or about the axis at each point of the simulation.
Rotary	Sets a rotary motion. You define the motion using the timing options. The motion curve is a set of 360 values, one for each angular increment of 0-359 degrees, that determines how much angle to rotate about the axis at each point of the simulation.
	Press Timing **Start** Lets you specify the start angle. **Stop** Lets you specify the stop angle. **Return Timing** **Start** Lets you specify the start angle. **Stop** Lets you specify the stop angle. **Move Distance** Sets the linear move distance. This only applies to linear motion. **Rotation Angle** Sets the rotation angle. This only applies to rotary motion.
From File	Lets you define the motion from an external CSV file.

Import Curve Data	Appears when the **Motion Curve** is set to **From File**. **Import Curve File** Lets you import an external curve file to be used for motion.
Name	
Sets the name for the motion you are defining.	
User Defined Motions	
Add New Set	Lets you define a new set of motions to be added to the list.
List	**Name** Sets the name for the motion that you are defining. **Active** Indicates whether or not the defined slide is active.
Preview Motion	Previews the motion you have defined.
Export Motion Curve	Lets you export the motion curve to an external file for later use.
Settings	
Use Control Data	Uses the kinematic control data for the user defined motion.

Add a kinematic model to a die assembly

Choose **Menu→Tools→Process Specific→Progressive Die Wizard→Validation Tools→Preprocess Motion**.

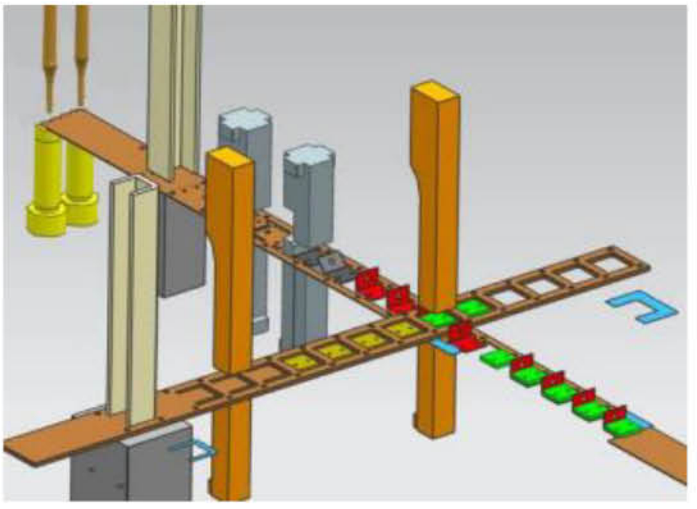
In the **Preprocess Motion** dialog box, in the **Type** group, select **Add Kinematic Model** .

In the **Kinematic Model** group, from the list, select a kinematic model template.

Two default templates are provided. You can also create other templates. (A **Machine Tool Builder** license is required.)

Tooling Kinematic Single Strip Model: This template is for a regular progressive die with one strip layout.

Tooling Kinematic Two Strips Model: This template can simulate a progressive die with two strips moving perpendicularly.



(Optional) In the **Target Directory** box, specify a file location for the kinematic subassembly.

Click **OK** or **Apply**.

t5. Run Simulation

Use this command to view a simulation that shows the kinematics of all the automatic and user-identified components in the model.

You can:

View the components and parts within the model and check for collisions and interference.

Detect and analyze the collision at each angle of the simulation.

Choose one of three accuracy methods for collision checking:

If you want a quick rough check, use the **Facet Body Distance** option. The check time is relatively fast but the precision is rough.

If you want average precision, use the **Mesh Triangle Intersection** option. The check time is average.

If you want the greatest precision, use the **Solid Body Intersection** option. The check time is longer than the other two options.

Where do I find it?

Application	**Mold Wizard** and **Progressive Die Wizard**
Command Finder	**Run Simulation** 

Run Simulation dialog box

Position in Cycle	
Angle	Lets you move directly to any angle of the simulation, or enter a specific angle. Drag the slider or enter an angle value (between 0-360) to go to a specific position in the simulation.

	The cycle of the simulation is a set of 360 values, one for each angular increment of 0-359 degrees, that determines how far to move along or about the axis at each point of the simulation.
Refresh	
Interval	Lets you control the speed of the simulation. Drag the slider or enter a value between 0-10 degrees to set the refresh rate. This controls the speed of the simulation. The refresh rate is the number of degrees between each step of the animation.
Controls	
Control buttons	Let you control the simulation. When you click **Play** , NX runs the simulation within 0-360 degrees and repeats this until you click **Stop** .
Collision	
Collision Check Method	Alows you to specify the type of collision check to be used when analyzing data for collisions. **Facet Body Distance**: Provides a quick check for collisions. The check time is relatively fast but the precision is rough. This method makes it difficult to distinguish between touching and a hard collision. **Mesh Triangle Intersection**: This method is more accurate than the **Facet Body Distance** method, and it has greater accuracy for determining a touch from a hard collision. Identifies the colliding facet bodies rather than just a mounted component. **Solid Body Intersection**: This is the most accurate method in identifying collisions. It uses a boolean analysis between the solid bodies to determine the collision. Larger assemblies will take a considerable amount of additional time to complete using this method.
Check Collision	Checks the collision and displays the collision in a list.
Stop Simulation at Collision	Available when **Check Collision** is on. Stops the simulation when the collision list changes.
Highlight Collision	Highlights the colliding bodies during the collision.

Ignore Touching	Disregards any touching information that occurs in the simulation.
Shortcut commands	Available in the shortcut menu when you right-click a collision in the list. **Analyze** Opens the **Analyze Collision** dialog box where you can see more detailed information about the collisions. **Ignore This collision** Ignores the selected collision including interference. Note Applies only to one collision between the two parts. If the parts collide again at a different point in the cycle, NX detects the collision. **Ignore Between Parts** Ignores the collision for the selected pair of parts.
Collision list	Displays all currently colliding or interfering pair of parts in a list. The list changes as the simulation progresses. New collisions are added at the top and removed when they no longer collide. You can right-click an entry in the list and choose **Ignore** to turn off the collision. Tip You can choose to ignore just the collision in the current range or all collisions between the two parts. You can turn off collisions that you are not interested in, for example, a collision between the sheet metal and a die at the closed position of the press, which is an expected collision.
Collision Configuration	Opens the **Collision Configuration** dialog box where you can specify more detailed parameters regarding collisions..
Collision Information	Opens an **Information** window which includes the detailed collision data.
Reset Ignored Collision	Restores any ignored collisions to their original state. The restored collisions will appear in the simulation.

t6. Run and analyze a simulation

Choose **Menu→Tools→Process Specific→Progressive Die Wizard→Tooling Validation→Tooling Motion Simulation**.

In the **Tooling Motion Simulation** dialog box, in the **Type** group, select **Run Simulation** .

In the **Run Simulation** group, click **Run Simulation** .

In the **Run Simulation** dialog box, click **Play**  to observe the current motion.

Only components that are assigned to a node in the kinematic assembly are shown.

To perform a dynamic collision check, select the **Check** check box.

The analysis takes a while to perform.

With a really large assembly, it is useful to add only a few components at a time.

You can also click **Modify Checks** to reduce the number of checks that are done for each step of motion.

When **Check** is selected, you can also select the following:

Stop — Stops motion when a collision occurs.

Highlight — Highlights pairs that touch or interfere within the tolerance specified in the **Clearance** box.

You can play the motion cycle one step at a time by clicking **Step Forward** , or with the **Stop** option, click **Play**  and allow the motion to proceed until a change is detected.

You can control the number of degrees of the 360 degree cycle that occurs when you click

When you find a motion step, click **List Changes** for a detailed description of the changes during the most recent motion.

For example, the **Information** window might contain the following:

Collision changes from 30.0 to 32.0 degrees

TOOLING_ATTACH_FIX_MNT and **TOOLING_ATTACH_PRODUCT_MNT** are no longer colliding

TOOLING_ATTACH_EJECTION_MNT and **TOOLING_ATTACH_PRODUCT_MNT** are colliding

Click **Close** when you are done.

u. Component Drawing

u1. Function and command call

Use the **Component Drawing** command to automate the creation and management of drawings for components in a mold or die assembly. You can:

Select components in the Modeling application to create a drawing directly.

Assign the same or different templates to multiple parts.

Edit drawings; for example, add a new sheet, delete a sheet, delete a self-contained drawing, or change the drawing template.

Use a user-defined drawing template, which dynamically changes the drawing frame size according to the part size.

Where do I find it?

Application	Mold Wizard, Progressive Die Wizard
Command Finder	**Component Drawing**

u2. Component Drawing dialog box

Drawing	
Create Drawing	Lets you create drawings for all the components that you select in the Components table. You can filter the components that are listed, using the **Component Type** list. Note Only components that do not have associated drawings are listed in the Components table.
Update Drawing	Lists in the Components table the drawings for the selected components in the current assembly.
Component Type	Specifies the components to display. **All** displays all the components in the assembly for which drawings do not exist. A selected component type, such as PLATE or SCREW filters the list to display only the components of that type. The component type attribute (PDW_COMPONENT_NAME) values are defined in **$PDIEWIZARD_DIR\templates\pdw_attribute.xls::Attribute_list**. You can customize these values.
Select Component	Lets you select one or more components for which you want to create drawings. You can select components: From the Components table in the **Component Drawing** dialog box. In the graphics window. The view must be either a drawing view or a model view. From the **Assembly Navigator**.

<table>
<tr><td></td><td>Note In the Assembly Navigator, make sure that you select the objects that you want. For example, if you open the Assembly Navigator, right-click the top-level node, and choose Select Assembly, only one object, your top-level assembly, is selected.</td></tr>
<tr><td>Components table</td><td>Lists the components eligible for drawing creation when you click Create Drawing. Lists components that have drawings, when you click Update Drawing.
<table>
<tr><td>Name column</td><td>Displays the name of the component.</td></tr>
<tr><td>Drawing File column</td><td>Displays the default name for the potential Master Model drawing of the component. You can specify a name for a Self Contained drawing.

When Update Drawing is selected, you can select a drawing in the table and press the right mouse button to display the following menus:

Add Sheet: You can specify a new sheet name.

Edit Sheet: Appears If you have changed the template for the drawing. Select it to apply the new template.

Delete Sheet: Lets you delete the sheet currently listed in the Sheet column.

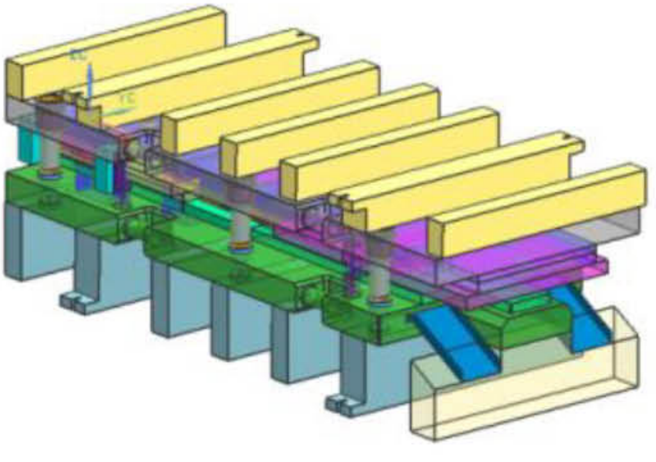
Open Drawing: Opens the selected drawing.

Delete Drawing: Deletes the selected drawing, unless it is the displayed part or the work part.</td></tr>
<tr><td>Sheet column</td><td>Displays the name of the sheet on which the potential drawing will be created.</td></tr>
<tr><td>Template</td><td>Displays the name of the template currently assigned to the component.</td></tr>
<tr><td>Type column</td><td>Lists the component type that is selected from the Component Type list.</td></tr>
</table></td></tr>
<tr><td>Attributes Copied To Drawing Part</td><td>
<table>
<tr><td>Toggle column</td><td>Whether to copy this title and value as an attribute to the drawing part.</td></tr>
<tr><td>Title column</td><td>Displays the attribute titles.</td></tr>
<tr><td>Value column</td><td>Displays the attribute values.</td></tr>
</table>
Note You can double-click the title or value column to edit it.</td></tr>
<tr><td></td><td>Appears when a component type is selected.</td></tr>
</table>

Add Component to List	Lets you add to the Components table a component that you select in the graphics window and assign to it the current **Component Type** part attribute. You select the component first and then click this button.
 **Create Drawing**	Appears when **Create Drawing** is selected at the top of the dialog box. Creates drawings of the currently selected components.
Templates	
Template	Lists the templates available for the current assembly, which vary depending on the whether the assembly units are English or metric. Metric templates are available for drawing sizes: A0, A1, A2, and A3. English (inch) templates are available for drawing sizes: B, C, D, and E.
Settings	
Projection	Lets you specify the angle of a projected view in your drawing. See the *Drafting* Help for more information about projected views, drawings, and drawing types.
Keep Drawing Open	Keeps all the drawings that you created open after you close the **Component Drawing** dialog box. You can access the drawings from the **Window** menu

u3. Create drawings for selected components in a die assembly

This example shows how to create drawings for one or more components of a specified type in a die assembly.



Die assemblies can have hundreds or even thousands of components. Sometimes, you want to create drawings only for specified types of components, such as the die plates.

Open the **Component Drawing** dialog box in one of these ways:

Choose **Progressive Die Wizard** tab→**Drawing Automation** gallery→**Component Drawing** .

Choose **Menu**→**Tools**→**Process Specific**→**Progressive Die Wizard**→**Drawing Automation**→**Component Drawing**.

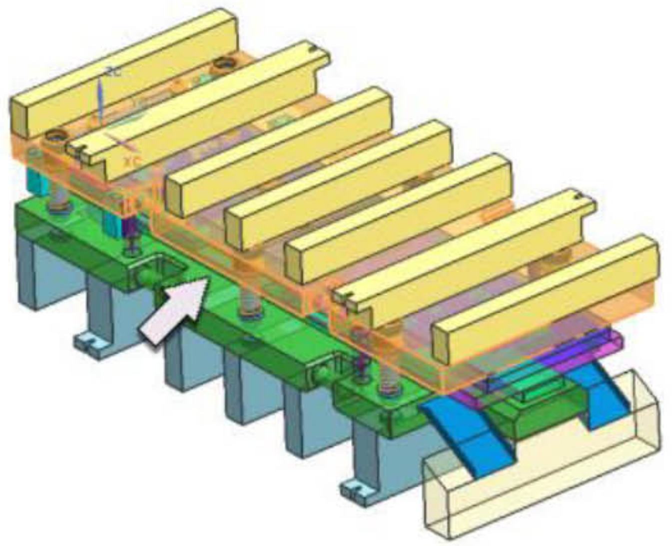
In the **Drawing** group, click **Create Drawing**.

From the **Component Type** list, select **PLATE**.

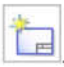
All the plate components are listed in the **Components** table.

Select the top plate.

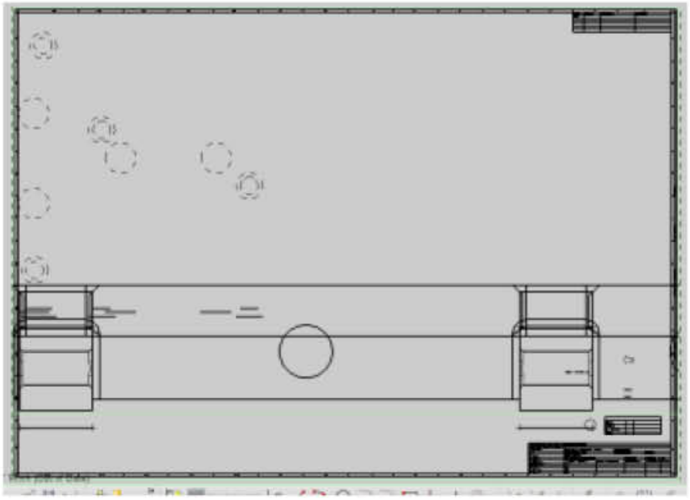
The component is highlighted in the graphics window.



In the row of the selected component, click in the **Drawing File** cell and from the drop-down list, select **Self Contained**.

Click **Create Drawing** .

NX creates a drawing of the top plate.



In the **Drawing** group, click **Update Drawing**.

Click **Return to the Root Part** .

The drawing closes and the assembly is displayed in the graphics window.

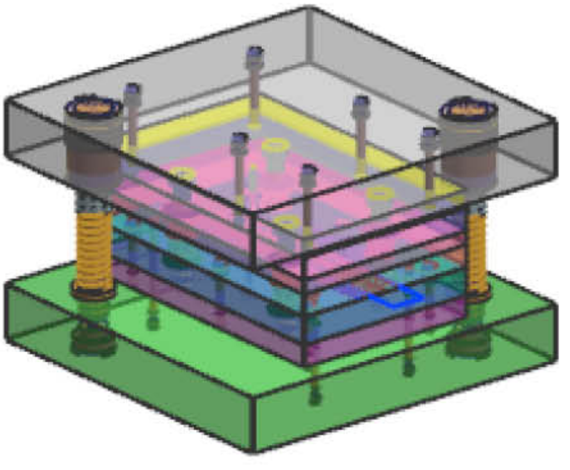
Click **Close** to close the **Component Drawing** dialog box.

Start the Drafting application.

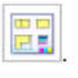
In Drafting, you can modify views, change view style, add annotations, notes, and balloons, and change text style, according to the requirements of the drawing and the standards of your company.

u4. Create drawings for all components in a die assembly

This example shows how to create component drawings for all the components in a progressive die assembly.



Open the **Component Drawing** dialog box in one of these ways:

Choose **Progressive Die Wizard** tab→**Drawing Automation** gallery→**Component Drawing** .

Choose **Menu**→**Tools**→**Process Specific**→**Progressive Die Wizard**→**Drawing Automation**→**Component Drawing**.

In the **Drawing** group, click **Create Drawing**.

All the components in the current assembly are listed, with their default drawing file names, sheet number, template, and type, if applicable.

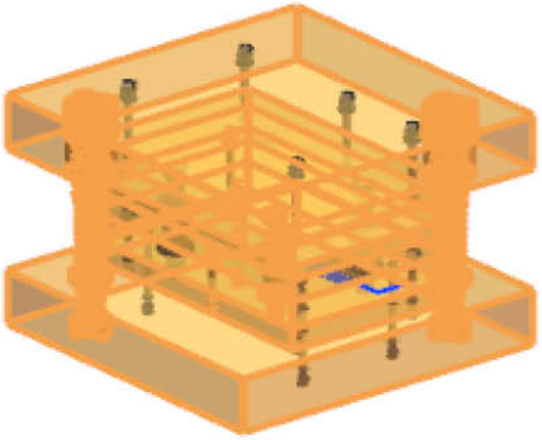
From the **Component Type** list, select **All**.

Select all components in your assembly.

In the Components table, select the top row.

Scroll to the bottom of the table, press and hold Shift, and select the bottom row.

Although there are other ways to select all the components in an assembly, you must be careful that your method selects the objects that you want. For example, if you open the **Assembly Navigator**, right-click the top-level node, and choose **Select Assembly**, only one object, your top-level assembly, is selected.



In the **Templates** group, from the **Template** list, select the template file you want to use for all the drawings.

For this example, **template_a0_comp_mm.prt** was selected.

Click **Create Drawing** .

NX creates a drawing for each component in your assembly.

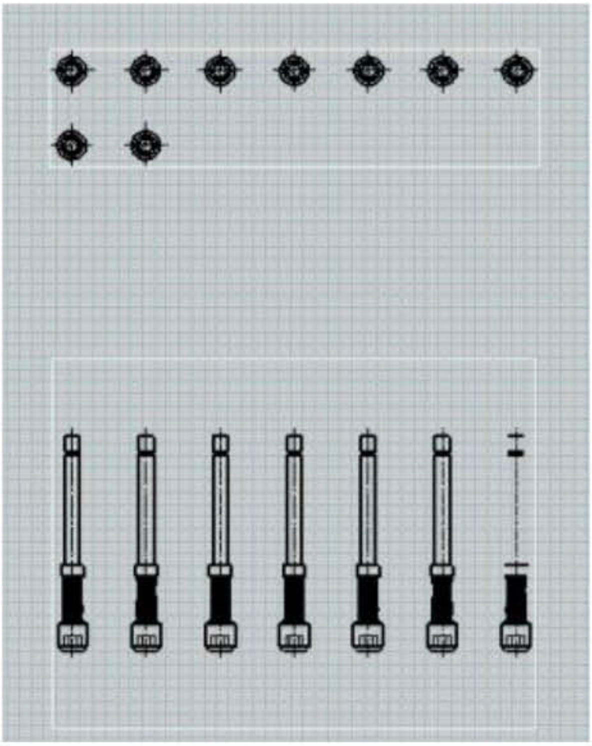
In the **Drawing** group, select **Update Drawing** to confirm that the drawings were created.

All the drawings for the die assembly are listed.

Select a drawing from the list, right click, and choose **Open Drawing**.

The component drawing opens with the default template settings.

For this example, a set of lifter components was selected.



To close the drawing, click **Return to the Root Part** .

The drawing closes and the assembly is displayed in the graphics window.

Click **Close** to close the **Component Drawing** dialog box.

Start the Drafting application.

In Drafting, you can modify views, change view style, add annotations, notes, and balloons, and change text style, according to the requirements of the drawing and the standards of your company

4.3. Designing the Progressive Die using NX

4.3.1. Create intermediate stages

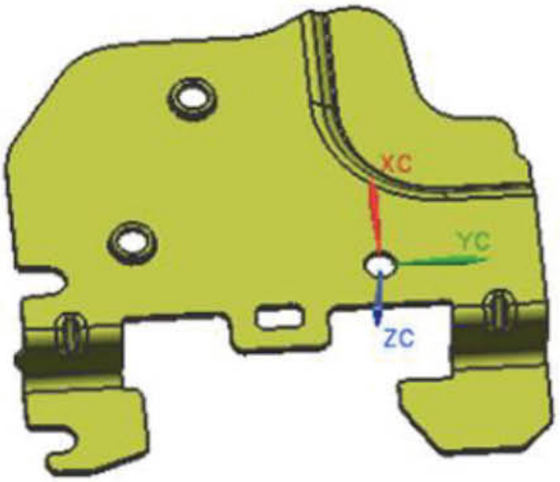
a. Detailed workflow 1: Create intermediate stages

This workflow shows you how to:

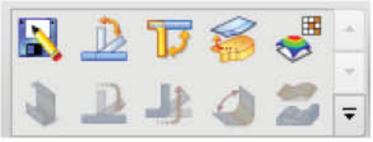
Prepare the bracket part shown.

Create intermediate stages.

Select a start stage.



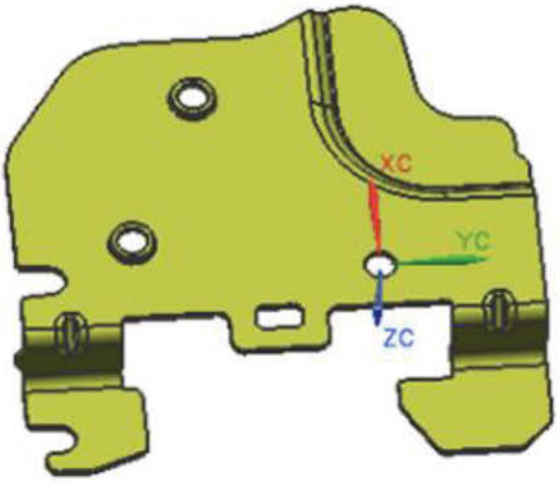
1.



Start the Progressive Die application.

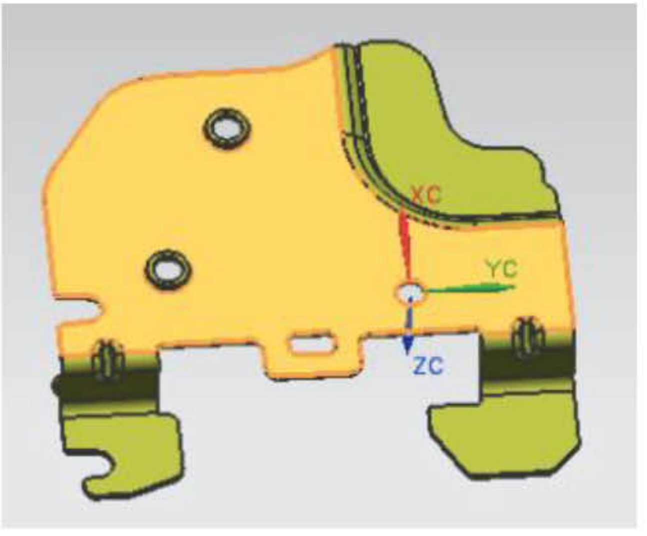
Most of the commands that you will use are in the **Progressive Die Wizard** tab→**Intermediate Stage Tools** group.

2.



Open the part.

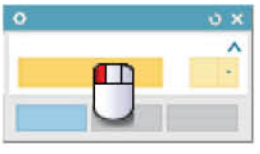
3.



Choose **Direct Unfolding** .

Set **Type** to **Convert to Sheet Metal** and select the top face as the base face.

4.



Choose **Define Intermediate Stage** .

Set the following:

From Part to Blank

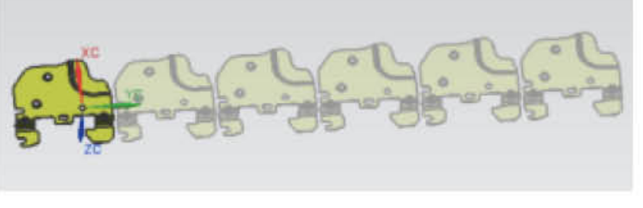
Number of Intermediate Stages

Start Station

Pitch

Orientation of Pitch

5.

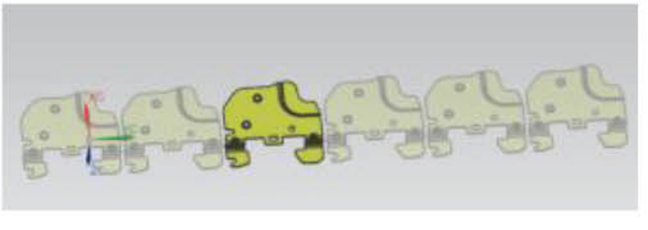


Create the intermediate stages.

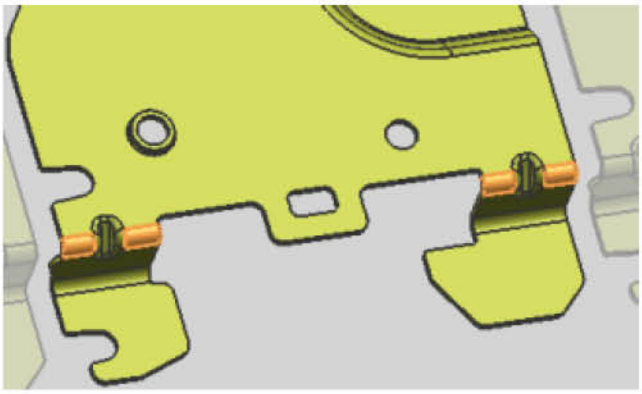
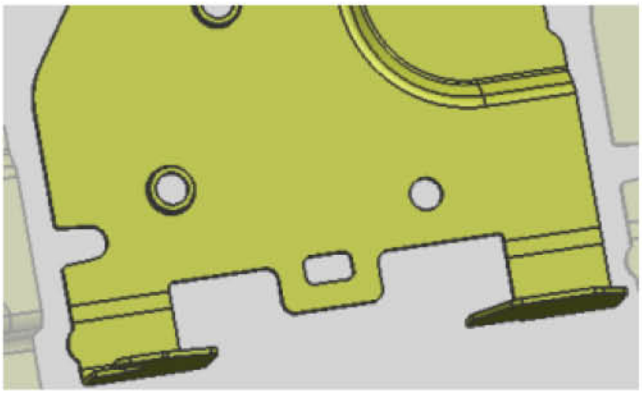
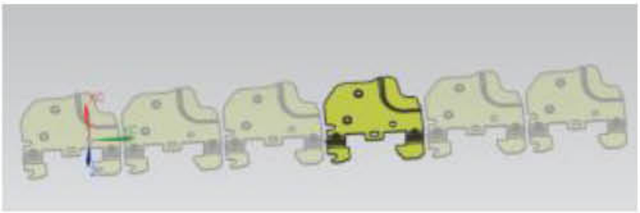
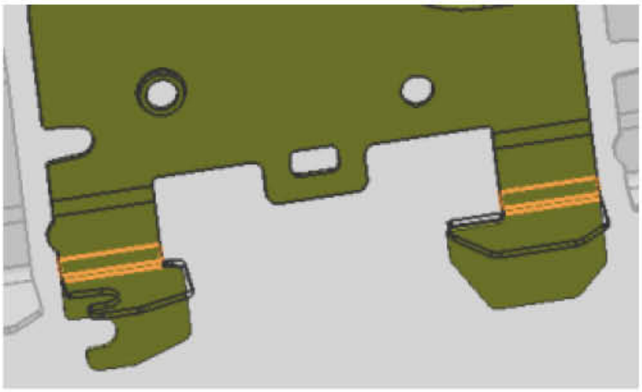
This workflow uses 5 intermediate stages.

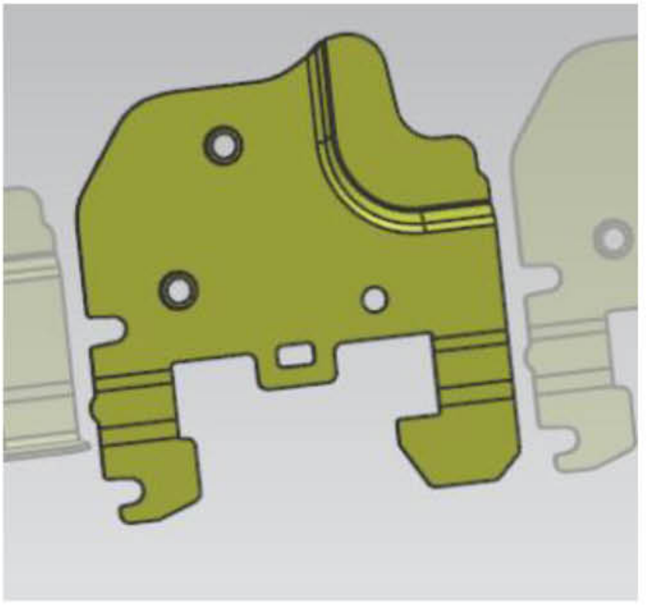
In the **Assembly Navigator**, the stages appear as **Final**, **Final-1**, **Final-2**, **Final-3**, and **Final-4**.

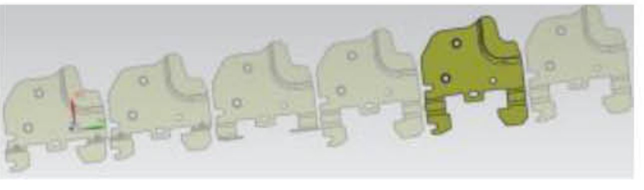
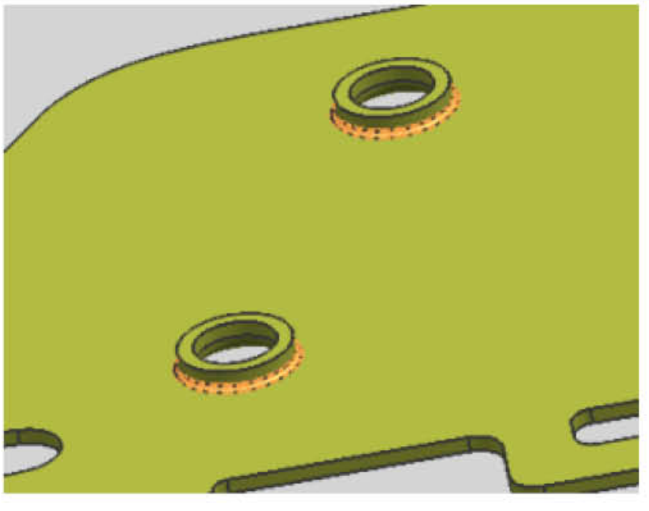
6.

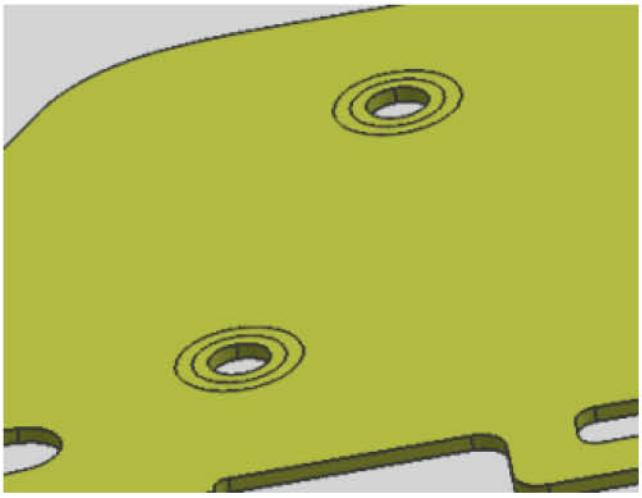


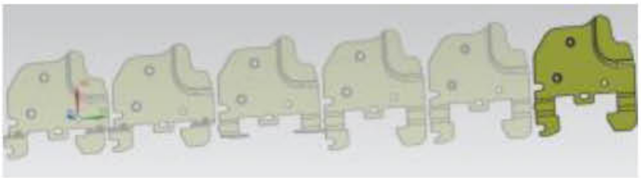
Make **Final-1** the work part.

7.	 	Choose **Universal Unform** . Set **Bend Type** to **Straight Bend**  and unbend the deformed bend regions.
8.		Make **Final-2** the work part.
9.		Choose **Bend Operation** . Straighten the two remaining bends.

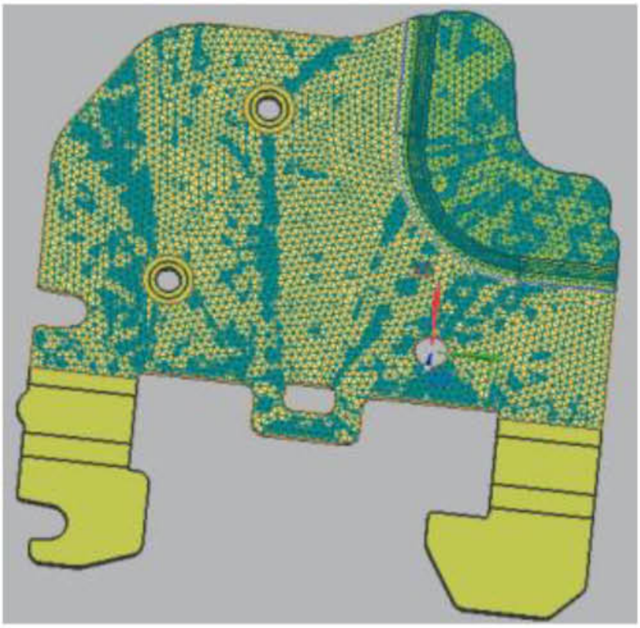


10.		Make **Final-3** the work part.
11.		Choose **Universal Unform** . Set **Bend Type** to **Burring**  and flatten the burring.



12. 

Make **Final-4** the work part.

13. 

Choose **Analyze** **Formability-One Step** .

Select the faces to be unformed.

Set the following:

Type = Intermediate Unform

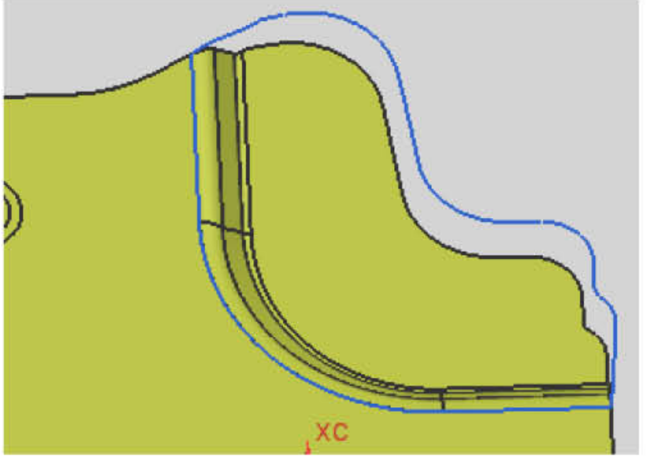
Object Type = Face

Select the target region face.

Set the overall element size.

Click **Mesh** .

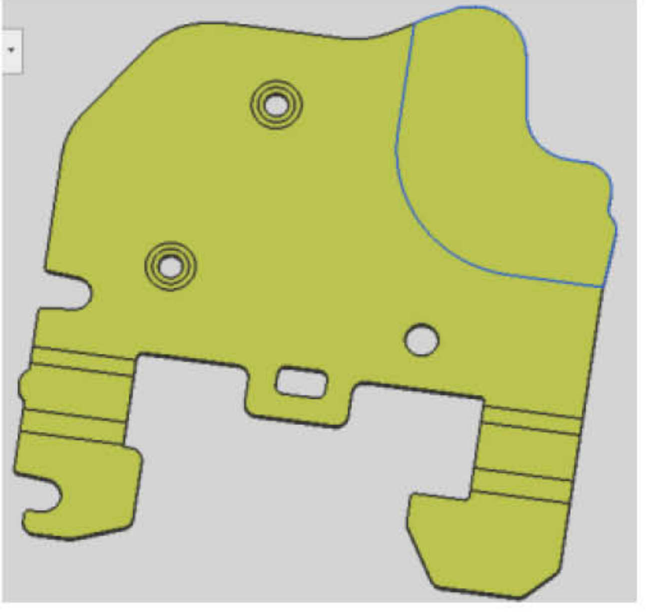
14.



In the **Calculation** group, click **Calculation** .

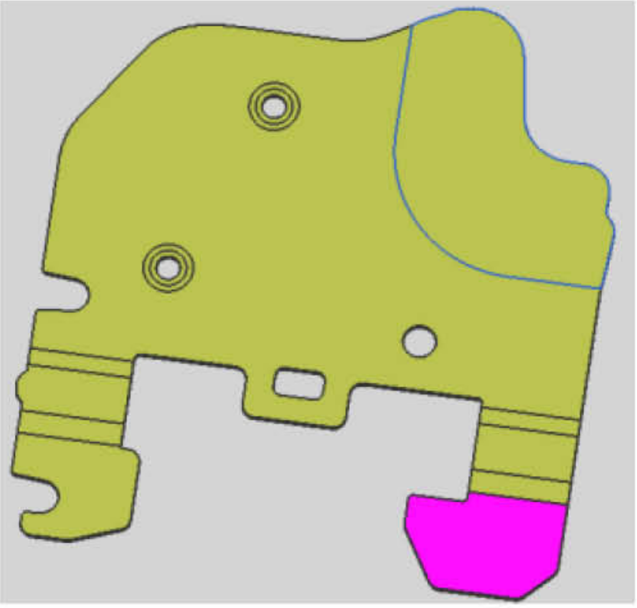
The results display a section that represents a flattened version of the deformed region.

15.



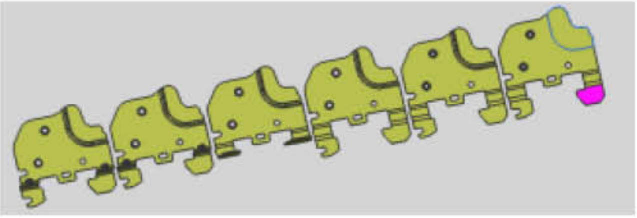
Use Modeling commands such as **Trim Body** and **Extrude** to remove the deformed region and replace it with a flat face that conforms to the shape defined by the **Analyze Formability - One-Step** command.

16.



To clearly distinguish this final stage from the other intermediate stages, select a face and change its color.

17.

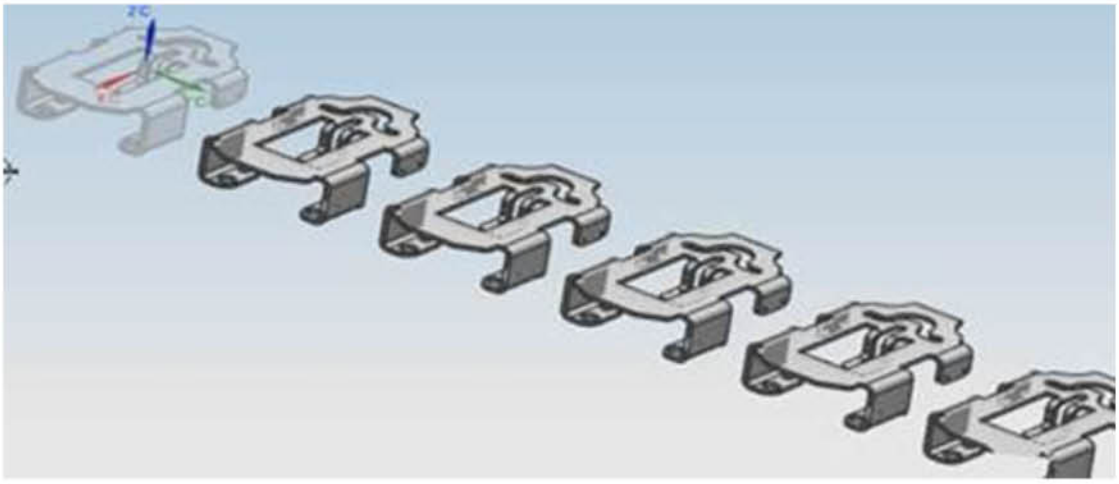


Display the parent assembly.

b. Define Intermediate Stage

b1. Function and command call

Use the **Define Intermediate Stage** command to create a specified number of associative intermediate unfolding stages in your assembly. You can then use the **Bend Operation** command to unbend, rebend, or flatten the straight-brake bends in each of these intermediate stages. You can also use the **Universal Unform** or **Analyze Formability — One-step** commands to unform freeform geometry, or use other modeling commands to generate the intermediate geometry.



The command supports a default item type in Teamcenter for newly-created intermediate parts. You can establish the naming rule using the **Custom Part Naming Rules** customer default.

Tip To find a customer default, choose **File→Utilities→Customer Defaults** and click **Find Default** .

Where do I find it?

Application	Progressive Die Wizard, Engineering Die Wizard
Command Finder	**Define Intermediate Stage**

b2. Define Intermediate Stage dialog box

Define Intermediate Stage	
Stage Sequence	Specifies the order in which the intermediate stages are created. **From Part to Blank** Assigns station numbers of intermediate stages in decreasing order. **From Blank to Part** Assigns station numbers of intermediate stages in increasing order.
Number of Stages	Sets the number of intermediate stages for your assembly.
Start Station	Sets the stage at which the unfolding begins.
Pitch	Sets the distance between the intermediate stages in the graphics window.
Orientation of Pitch	Sets the axis on which the intermediate stages will be aligned: X, Y, or Z.
Preview	Displays the stages using the selected pitch orientation.
Edit Intermediate Stage	
Select Intermediate Stage	Lets you select an intermediate stage for inserting a new stage, or for deleting one.
Settings	
Naming Rule for Intermediate Stages	Defines how to name the intermediate stages.

Top Part Name	Lets you specify the name of the top part when creating an intermediate stage.
☑ **Create Copy of Sheet Metal Part**	Creates a copy of the sheet metal part within the intermediate stages. Note The copied part has the suffix **copy** following the part name, and is not assigned any attributes associated with the original part.
☑ **Rename Components**	Opens the **Part Name Management** dialog box, allowing you to rename the component.
☑ **Link Sheet Body for Intermediate Stages**	Lets you link intermediate stages to a sheet body instead of, or in addition to, a solid body

c. Direct Unfolding

c1. Function and command call

Use the **Direct Unfolding** command to:

Automatically recognize straight-brake sheet metal bends, including bends with a 0 bend radius, or bends without a planar base.

Define prebend parameters for each bend. The **Prebend** command combines **Divide Face** and **Extrude** features into a single **Prebend** feature in the **Part Navigator**.

Merge bends of equal radius that share a common bend axis.

Change the K Factor and developed length parameters for individual or multiple bends in a sheet metal part.

Specify additional geometry to be used to flatten a sheet metal part by using **Select Additional Bend Faces** within your operation.

Where do I find it?

Application	Progressive Die Wizard, Engineering Die Wizard
Command Finder	**Direct Unfolding**

c2. Direct Unfolding dialog box

Type	
Type list	Lets you specify the direct unfolding operation to perform: **Convert to Sheetmetal** converts an imported part or a part created in the Modeling application to an NX Sheet Metal part. **Merge Bends** merges bend regions that have been split into multiple bend segments, or that have the same axis, radius, and reference face or edge. **Define Prebends** divides bend regions into multiple bend segments. **Delete Bends** deletes bend regions.
Recognize Bends	
Appears when **Type** is set to **Convert to Sheetmetal**.	
Select Base Face	Lets you select a planar face to serve as the base face for the part you are converting to an NX Sheet Metal part.
	Lets you specify additional bend faces when converting to sheet metal.

Select Additional Bend Faces	
Bend List	
Bodies in Part	Available when more than one body is available in the current part. Lets you designate a body to use when selecting bends for unfolding operations.
 **Select Bend**	Lets you select a bend from the list or in the graphics window for direct unfolding operations.
 **Select Start Edge**	Available when **Type** is set to **Define Prebends**. Lets you select the start edge that is used to calculate the angle for multiple prebends.
Bend list	Lists the recognized bends with their radii, angles, K Factors, and developed lengths.
Define Neutral Factor	
Lets you specify the material, neutral factor, and developed length.	
Developed Length Calculation	Selects the source for material and bend parameters in the **process_data.xls** spreadsheet. **Neutral Factor Table** Designate the **Neutral Factor** page as the source for bend parameters. **BAF Table** Designate the **BAF** (Bend Allowance Formula) page as the source for bend parameters. NX uses the user defined **BAF** to calculate the developed length of bend faces.
Material	Lets you select a material that is defined on the **Material** page of the **process_data.xls** spreadsheet.
 **Load Material Database**	Opens the **process_data.xls** spreadsheet.
Neutral Factor	Available when **Developed Length Calculation** is set to **Neutral Factor Table**. Shows the neutral factor that is interpolated from the table. You can override it by selecting a different value.
Developed Length	Shows the calculated unfolded length of material. You can override it by typing a new value.

Define Prebends	
Appears when **Type** is set to **Define Prebends**.	
Number of Bends	Specifies the number of prebends to create in the selected bend.
Angle	Specifies the angle of a prebend. If **Number of Bends** is set to more than 2, multiple **Angle** boxes appear.
Merge Bends	
Appears when **Type** is set to **Merge Bends**.	
Select Bend	Lets you select the following bend types and merge them into one bend: A parent of the prebends in the **Bends List** Bends with the same axis, the same radius, and the same reference face or edge
Delete Bends	
Appears when **Type** is set to **Delete Bends**.	
Select Bend	Lets you select a bend in the **Bends List** and delete it.

d. Bend Operation

d1. Function and command call

Use the **Bend Operation** command to:

Unbend, rebend, prebend, and overbend sheet metal parts without entering the NX Sheet Metal application. Prebends also appear in the **Direct Unfolding** dialog box.

Define prebend parameters for each bend. The **Prebend** command combines **Divide Face** and **Extrude** features into a single **Prebend** feature in the **Part Navigator**.

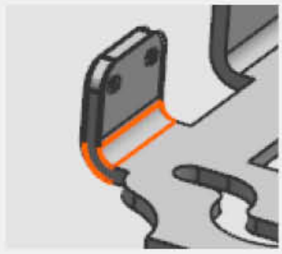
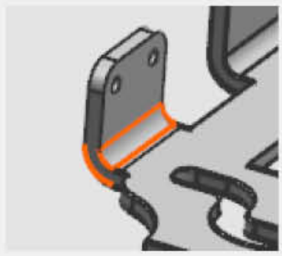
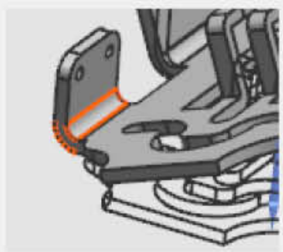
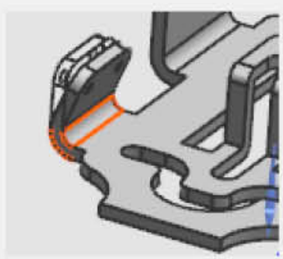
Note You can use the **Bend Operation** command only on parts that you converted to sheet metal in the **Direct Unfolding** operation, or on sheet metal parts you created in the NX Sheet Metal application.

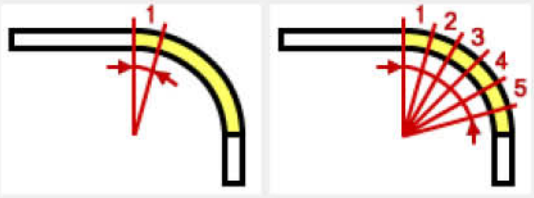
Where do I find it?

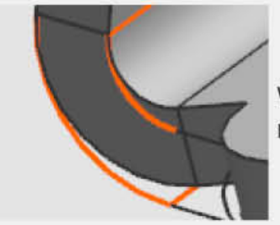
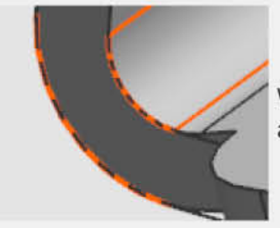
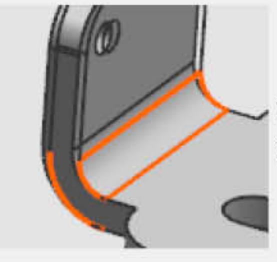
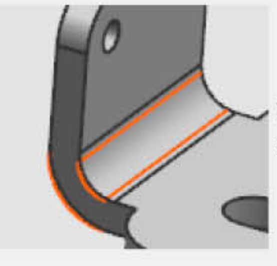
Application	Progressive Die Wizard, Engineering Die Wizard
Command Finder	**Bend Operation**

d2. Bend Operation dialog box

Type	
Type menu or buttons	Specifies the type of bend operation. **Unbend** **Rebend** **Prebend** **Overbend**
Inputs	
Select Intermediate Stage	Lets you select an intermediate stage in the graphics window with bends that you want to modify. You can create intermediate stages with the **Define Intermediate Stage** command.
Select Bend	Lets you select one or more bends for the bend operation.
Show Alternate Result	Appears when **Type** is set to **Unbend**, **Rebend**, or **Overbend**. Displays alternate results for your inputs.

	 Alternate overbend radii   Alternate overbend angle positions. 
Prebend Parameters	
Appears when **Type** is set to **Prebend**.	
Developed Length Calculation	Designates a tabbed page in the **process_data.xls** spreadsheet from which the material and bend parameters are taken to develop the length of bend material. **Neutral Factor Table** Use the **Neutral Factor** page **BAF Table** Use the **BAF** page
Material list	Lists you specify a material that is defined on the **Material** page of the **process_data.xls** spreadsheet.

Load Material Database	Opens the **process_data.xls** spreadsheet.
Neutral Factor	Available when **Developed Length Calculation** is set to **Neutral Factor Table**. Shows the neutral factor that is interpolated from the table. You can override it by selecting a different value.
Developed Length	Shows the calculated unfolded length of material. You can override it by typing a new value.
picture	Varies with the total number of bends that you specify, from 2 to 6.  The number of bends shown is always one less than the total number of bends. The first bend is the one defined by the geometry; the rest are prebends.
Number of Bends	Sets the number of prebends, from 2 to 6.
Angle(*n*)	Lets you specify intermediate angles. The number of input boxes (*n*) ranges from 1 to 5.
Prebend Parameters	
Appears when **Type** is set to **Overbend**.	
Overbend Type	Specifies the type of overbend operation. **Resize Bend Angle** **Resize Bend Radius**
Target Angle	Appears when **Overbend Type** is set to **Resize Bend Angle**. Sets the angle to which the feature is bent in the overbend stage.
Keep Radius Fixed	Appears when **Overbend Type** is set to **Resize Bend Angle**.

	 When not selected, the overbend can change the radius.  When selected, the bend radius remains constant, and only the angle can change.
Target Radius	Appears when **Overbend Type** is set to **Resize Bend Radius**. Sets the radius to which the feature is bent in the overbend stage.
Fixed Tab/Flange Position	Appears when **Overbend Type** is set to **Resize Bend Radius**.  When not selected, the overbend can change the position of the tab or flange.  When selected, the feature position is constant, and only the radius can change.
Restore Part Position	Available when a result moves the fixed face of the part. Restores the fixed face and adjacent features to their position before the overbend.

d3. Perform a bend operation on a sheet metal part

Choose **Progressive Die Wizard** tab→**Intermediate Stage Tools** gallery→**Bend Operation**, or choose **Menu**→**Tools**→**Process Specific**→**Progressive Die Wizard**→**Intermediate Stage Tools**→**Bend Operation**.

In the **Bend Operation** dialog box, set **Type** to the operation you want: **Unbend**, **Rebend**, **Prebend**, or **Overbend**.

In **Settings**, select the **Preview** check box if you want to see a dynamic preview in the graphics window as you define the operation's parameters.

In the graphics window, select a bend.

You can select more than one bend, or the software may select multiple bends for you if the bends have the same reference plane or edge.

Select the **Show Alternate Result** check box if you want to see alternate results for your inputs.

If **Type** is **Overbend**, in the **Inputs** group, do one of the following:

Specify the **Target Angle**.

If you want the bend radius to be fixed, select the **Keep Radius Fixed** check box.

Select the **Resize Bend Radius** check box and, in the **Target Radius** box, enter the radius you want.

Click **Show Result**  to preview the appearance of your part.

Click **Undo Result**  to return your part to its pre-operation appearance.

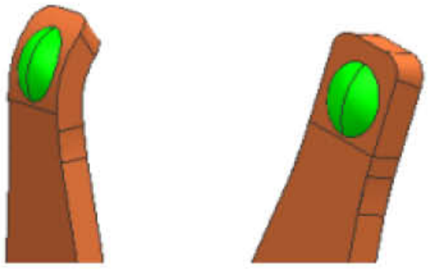
To apply the bend operation to your part, click **OK** or **Apply**

e. Universal Unform

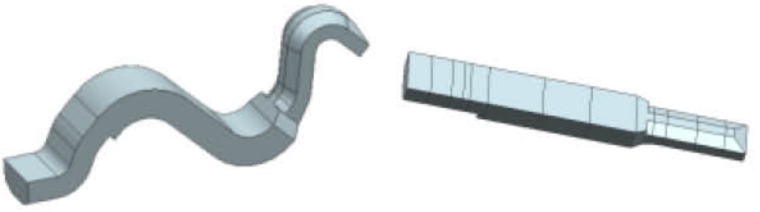
e1. Function and command call

Use the **Universal Unform** command to flatten complicated regions such as:

Bend regions that include deformation features, such as embosses.

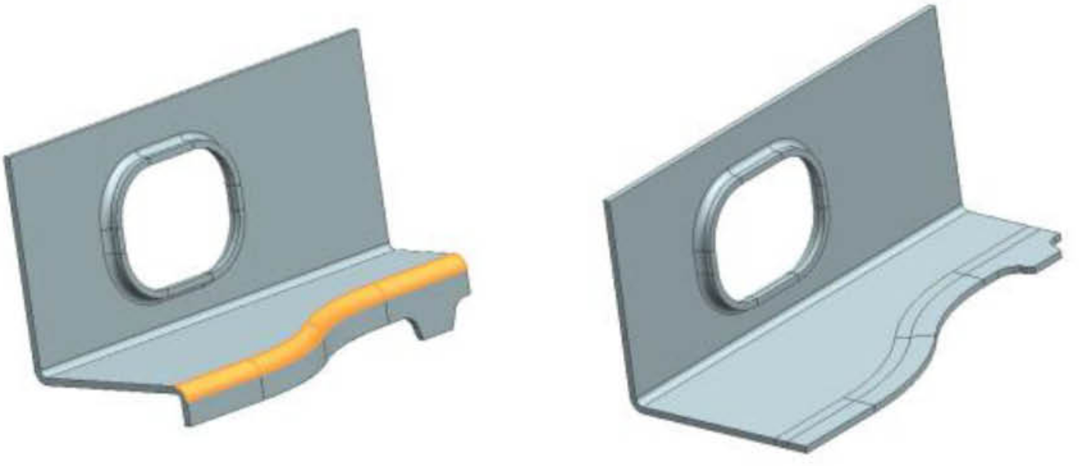


Bend regions on parts with non-uniform thickness.



Bend regions that include ribs, gussets, beads, or burring.

Contour flange bend regions.



Where do I find it?

Application	Progressive Die Wizard, Engineering Die Wizard
Command Finder	**Universal Unform**

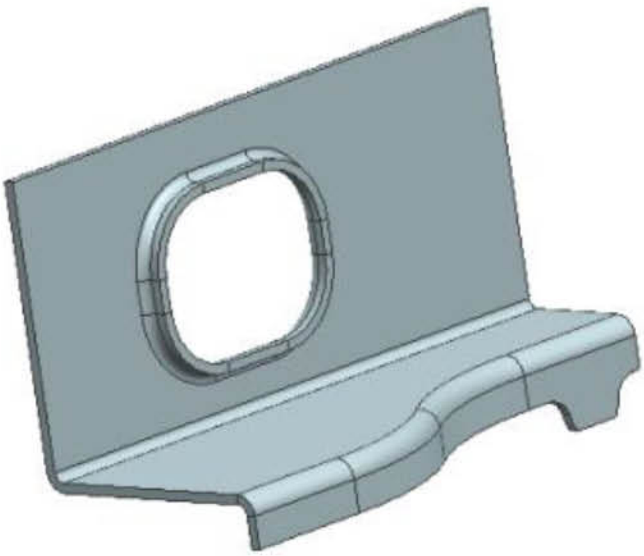
e2. Universal Unform dialog box

Define Bend	
Bend Type	**Straight Bend** Flattens straight bends with deformations such as embosses or ribs. **Contour Flange** Flattens a contour flange-type bend. The contour flange must lie on the same plane and maintain a consistent angle in relation to the reference face. **Burring** Flattens deformations that have blends at the base of the feature.
Select Reference Face or Edge	Lets you select a face or plane to indicate the position of the bend region when it is unbent.
Select Blend Faces **Select Bend Faces**	Lets you select a blend area or a bend face as the area to unform.
Developed Length Calculation	Selects the source for material and bend parameters in the **process_data.xls** spreadsheet. **Neutral Factor Table** Designate the **Neutral Factor** page as the source for bend parameters. **BAF Table** Designate the **BAF** (Bend Allowance Formula) page as the source for bend parameters. NX uses the user defined **BAF** to calculate the developed length of bend faces.
Material	Lets you select a material that is defined on the **Material** page of the **process_data.xls** spreadsheet.
Load Material Database	Opens the **process_data.xls** spreadsheet.

Neutral Factor	Available when **Developed Length Calculation** is set to **Neutral Factor Table**. Shows the neutral factor that is interpolated from the table. You can override it by selecting a different value.
Developed Length	Shows the calculated unfolded length of material. You can override it by typing a new value.

e3. Flatten complex areas with Universal Unform

This example shows how to unform a contour bend and a burred feature on a sheet metal part.



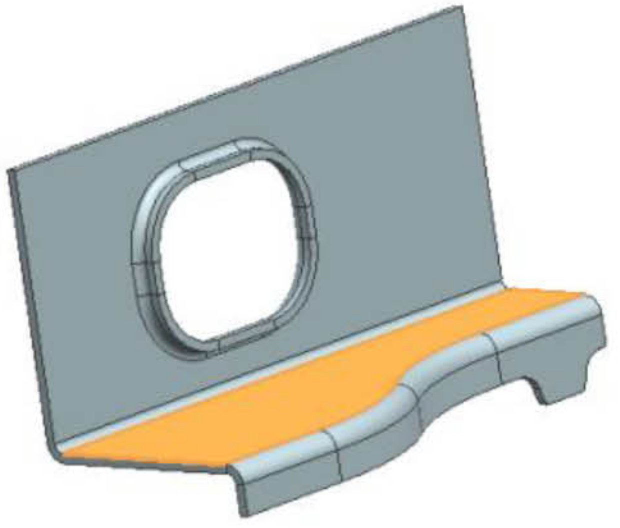
Open the **Universal Unform** dialog box in one of the following ways:

Choose**Progressive Die Wizard** tab→**Intermediate Stage Tools** gallery→**Universal Unform** .

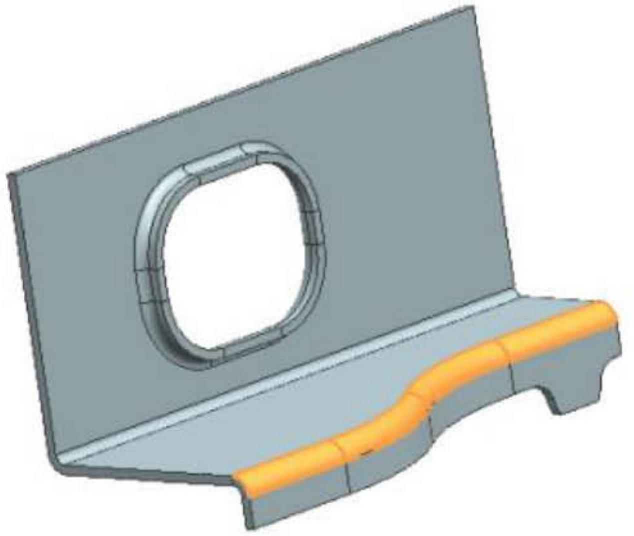
Choose **Menu**→**Tools**→**Process Specific**→**Progressive Die Wizard**→**Intermediate Stage Tools**→**Universal Unform**.

In the **Define Bend** group, under **Bend Type**, click **Contour Flange** .

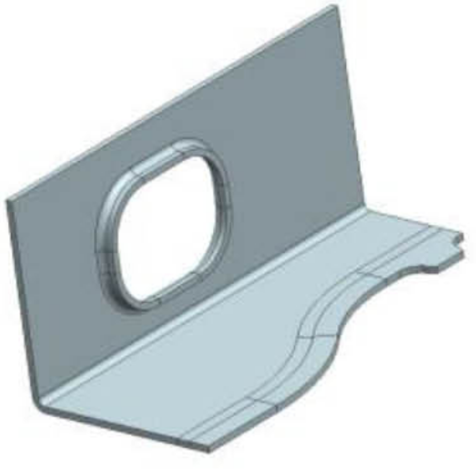
For the reference face, select a planar face, as shown.



With the **Face Rule** set to **Connected Blend Face**, select one of the bend segments and then click **Apply**.

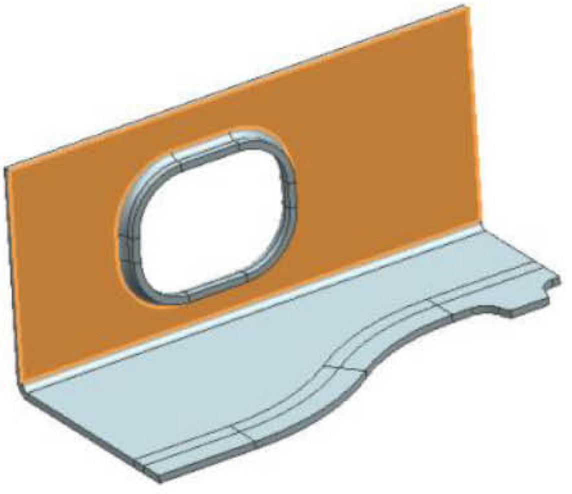


The contour flange is flattened.

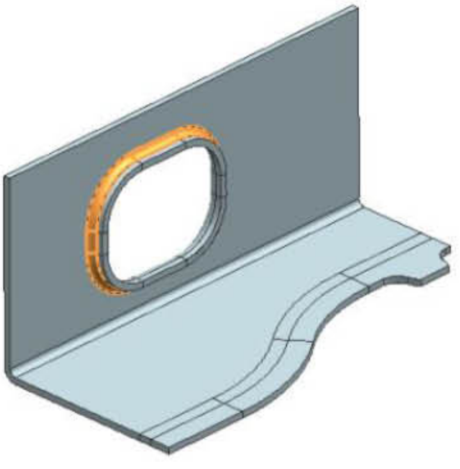


Under **Bend Type**, click **Burring** .

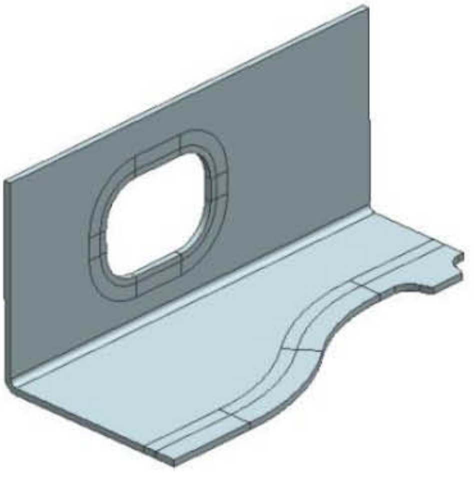
For the reference face, select a planar face as shown.



With the **Face Rule** still set to **Connected Blend Face**, select one of the blend segments and then click **OK**.



The burring feature is flattened.



f. Analyze Formability – One-step

f1. Function and command call

Use the **Analyze Formability – One-step** command to flatten all or some faces of a sheet metal part using FEM analysis and calculate the thinning, stress, strain, and springback to predict the risk of forming.

You can:

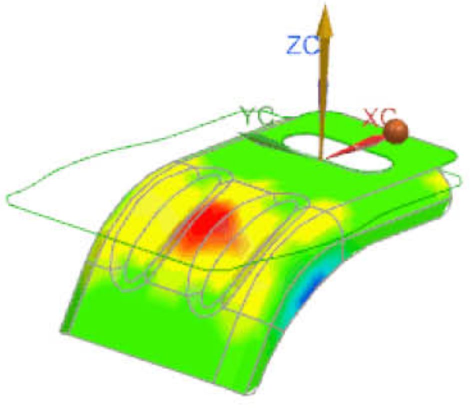
Perform complete or intermediate unforming, or flatten a Sheet Metal part.

Output the flattened profile or springback faceted bodies for complete part unforming. You can also specify the target region and the unform region from a different sheet body.

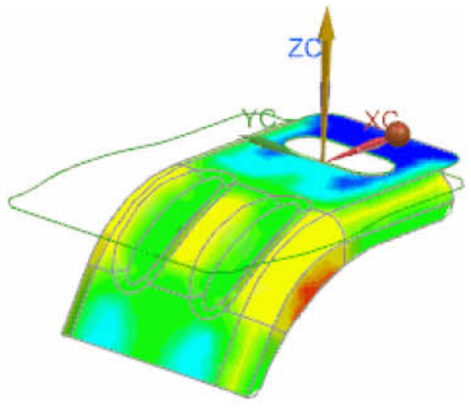
Define different constraint types, both geometry and process, to control unforming.

Create trim line feature curves on the addendum surfaces when your part has a flange and addendum sheet bodies with a common boundary.

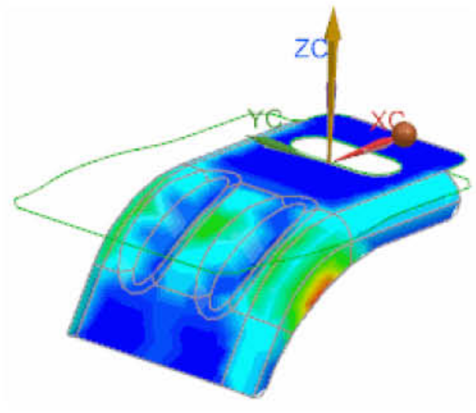
The flattened profiles are produced as spline curves and the analysis results are post-processed and displayed in a color-coded plot mode. The analysis report is generated based on the subsequent results.



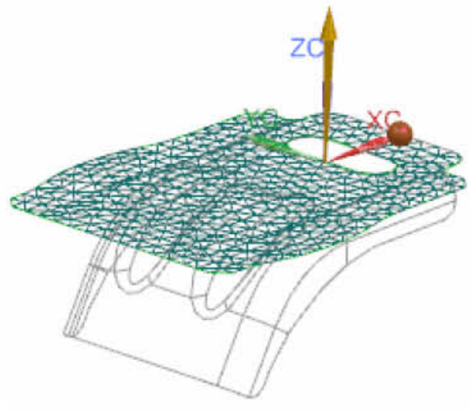
Thickness results



Stress results



Strain results



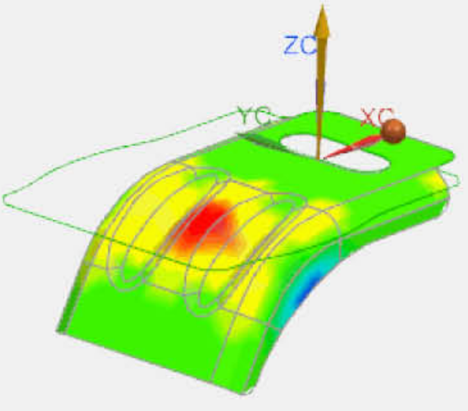
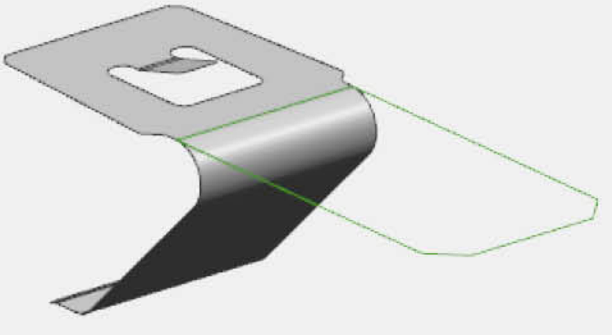
Flattened shape

The Onestep Unform feature which saves your inputs and settings, is listed in the **Part Navigator**, and can be edited if required. The data is retrieved when you reopen the part file and run the **Analyze Formability – One-step** command.

Where do I find it?

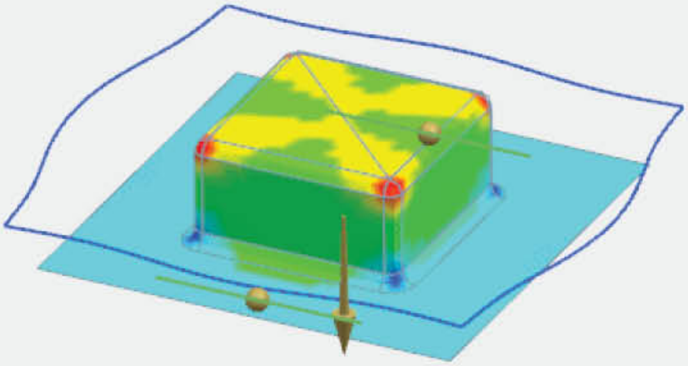
Application	Gateway, Modeling, Progressive Die Wizard, Engineering Die Wizard, Sheet Metal, Die Engineering, Die Design, Press Die Checker
Prerequisite	Simcenter Pre/Post installation (License not required) To work with Die Engineering commands, you must be in the Modeling application.
Command Finder	**Analyze Formability – One-step** 

f2. Analyze Formability – One-step dialog box

Type	
Type list	Specifies the type of unform feature you want to create.  **Entire Unform** Performs complete unforming of the part.  **Entire Unform — Thickness analysis results**  **Intermediate Unform** Performs intermediate unforming of the part.  **Intermediate Unform results**

Advanced Unform

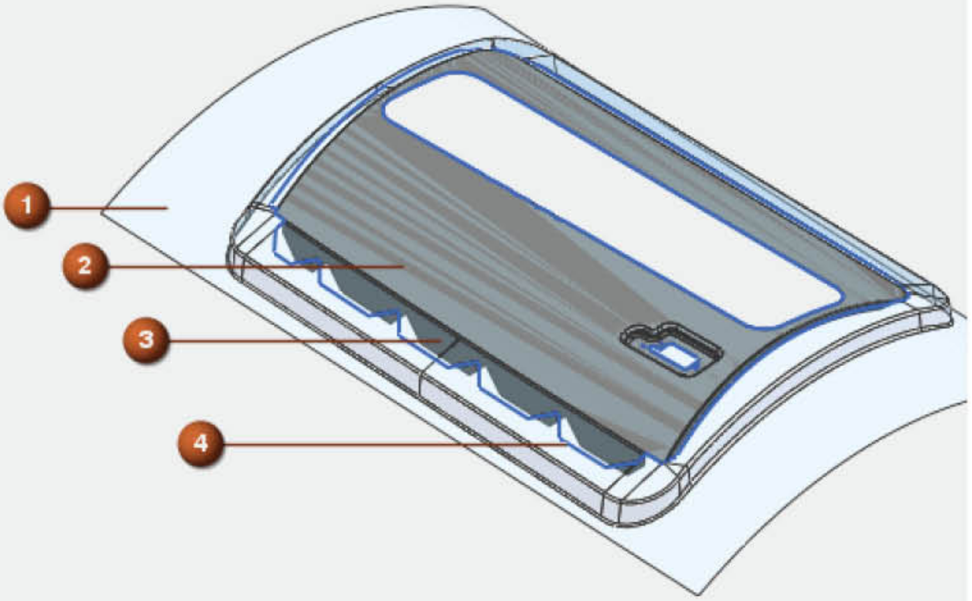
Performs advanced unforming. Additional advanced constraints and options are available to specify springback match points, or draw bead constraints with or without addendum, and create the springback faceted body.



Advanced Unform — Thickness analysis results with Draw Bead constraints

Trim Line

Creates trim line feature curves on the addendum surfaces when your part contains a flange and addendum sheet bodies with a common boundary.

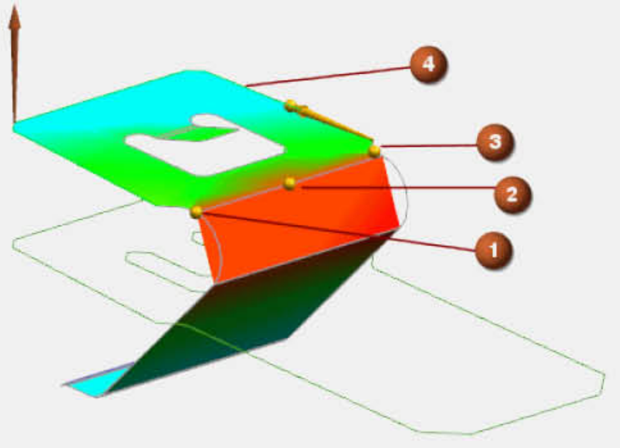


1Addendum face

2Product face

	3Flange face 4Trim line profile
Object Type	
Object Type list	Not available when **Type** is set to **Trim Line**. Lets you specify if you want to perform the formability analysis on a solid body or on a face. **Solid** Lets you select solid bodies to perform the formability analysis. **Face** Lets you select faces to perform the formability analysis.
Unform Region	
Select Faces	Not available when **Type** is set to **Trim Line**. Lets you define the surfaces to be unformed and calculated. If you select the entire surface of a body, a complete unform is performed. If you select a partial surface, an intermediate unform is performed.
Unform Curve or Point	
Select Curve or Point	Not available when **Type** is set to **Trim Line**. Lets you select curves or points on the selected faces to perform the formability analysis.
Target Region	
Select Faces	Available when **Type** is set to **Intermediate Unform**. Lets you define the target surfaces to which the unform region is mapped.

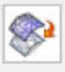
	If the target region and unform region faces are from the same body, they must have common edge which is defined as the **Curve to Curve** boundary condition constraint type for intermediate unform.
Boundary Conditions	
Not available when **Type** is set to **Trim Line**.	
Constraint Type list	Specifies the boundary constraints depending on the region selected to unform, and the unform type. **Curve to Curve** Lets you specify the curves that define the boundary for the unform region. For an intermediate unform, select common edges between the unform region and the target region. For an entire unform, select curves from the boundary of the unform region. The **Select Curve from Unform Region** option is available. **Point to Point** Lets you specify the points on the surface of the unform region to lock the position. The **Specify Point from Unform Region** options are available. **Curve along Curve** Lets you specify the curves that define the region for an entire unform. The **Select Curve from Unform Region** option is available.
Constraint List	
Lists the constraints you applied. Click **Add New Set** to specify additional constraints.	
Springback Match Points	
Available when **Type** is set to **Advanced Unform**.	
Match Point One **Match Point Two**	Lets you specify three points to define the springback constraints. For example, the following graphic shows how the three springback match points are fully constrained within six degrees of freedom.

Match Point Three	 1 First springback match point with zero displacement in the X, Y, and Z direction 2 Second springback match point with zero displacement in the X and Y direction 3 Third springback match point with zero displacement in the X direction 4 Constraint curves
Advanced Constraints	
Available when **Type** is set to **Advanced Unform**.	
Part Type list	Specifies the part type: **With Addendum** For a part with addendum, you can specify the binder region and the draw beads. **Without Addendum** For a part without addendum, you can specify the equivalent binder region.
Blank Holder	
These options are available when **Part Type** is set to **With Addendum**.	
Binder Region	Lets you specify the binder region.
Force (kN)	Sets the blank holding force. During stamping of sheet metal parts, appropriate control of the blank holding force minimizes the wrinkling tendency, material discontinuity, and non homogeneous stress or strain distribution. Generally, a lower blank holding force must be applied on the thicker blank side to allow more metal to flow in for obtaining a more uniform strain distribution.

These options are available when **Part Type** is set to **Without Addendum**.	
 **Equivalent Binder**	Lets you specify the equivalent binder region.
Tension (kN/mm)	Sets the value for the tension applied to the binder region.
Force Strength	Sets the value for the force applied to the binder region.
Draw Beads	
Available when **Part Type** is set to **With Addendum**.	
 **Specify Draw Bead**	Lets you specify the location of the draw bead.
Tangential Tension (kN/mm)	Sets the value for the tension applied in a direction that is tangent to the draw bead.
Normal Tension (kN/mm)	Sets the value for the tension applied in a direction normal to the draw bead.
Force Strength	Sets the value for the force applied to the draw bead.
List	Lists the specified draw beads.
Product and Flange Faces	
Appears when **Type** is set to **Trim Line**. Lets you select product and flange faces that define the unform region.	
Addendum Faces	
Appears when **Type** is set to **Trim Line**. Lets you select addendum faces that are tangential to the product and flange faces.	
Material	
Material list	Lists the materials available in the standard Default Material Library.

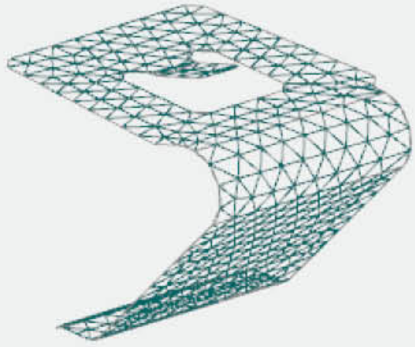
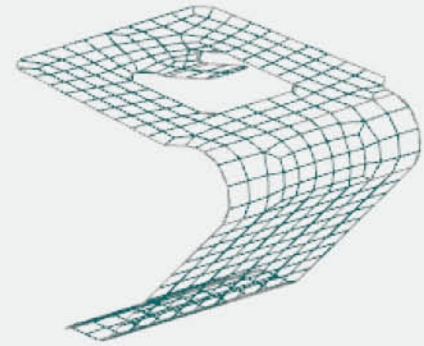
	If you customize the properties in the standard Default Material Library, the changes are reflected in the **Material** list and the analysis results.
Location	
Local Material	Specifies that a local material is used.
Default Material Library	Specifies that materials from the standard Default Material Library are used.
Site MatML Library	Lets you specify the location of an alternate materials from the MATML Library.
User Mat ML Library	Lets you specify a user defined material library.
Materials	
List	Displays the list of materials available from the specified material library.
Inspect material	Available when you select a material from the list. Opens the **Isotropic Material** dialog box in which you can inspect the material properties.
Information	Available when you select a material from the list. Displays the material properties in the **Information** window.
Draw Direction	
Not available when **Type** is set to **Intermediate Unform**. Appears when **Type** is set to **Entire Unform**, **Advanced Unform**, or **Trim Line**.	
Specify Vector	Lets you define the normal of the unform plane. The unform region is flattened normal to the plane in the specified draw direction.
Thickness	
Surface Type	Specifies the surface from which the unforming region is extracted. The solver automatically offsets the surface in the right direction. **Inner Surface**

	Creates an unform region with an offset, where the result is an unform profile larger than the original. **Middle Surface** Creates an unform region with no offset. **Outer Surface** Creates an unform region with the specified offset, where the result is an unform profile smaller than the original.
Infer Thickness	Available only for solid bodies. Infers the thickness automatically if the unform region surfaces are from a solid body.
Calculation	
Infer Element Size	Appears when **Type** is set to **Trim Line**. Infers the size of the mesh element.
Overall Element Size	Appears when **Type** is set to **Trim Line** and available when you clear the **Infer Element Size** check box. Sets the size of the mesh element.
Tolerance to Find Contact Size	Specifies the default tolerance that NX uses when, after creating mesh elements, NX searches for contact points between the product or flange faces and the addendum faces. NX uses the mesh elements to find the constraints, and defines the boundary of the trim line profile. You can specify the default tolerance using the **Tolerance to Find Contact Size** customer default. Tip To find a customer default, choose **File** tab→**Utilities**→**Customer Defaults**, and click **Find Default** .
Mesh	Creates the mesh in an interim FEM part and gets mesh elements data back to modeling for the Onestep solver to calculate.
Mesh Quality Check	Examines the mesh quality in term of mesh shape, mesh element node overlapping.
Calculation	Launches the solver to start calculating the unforming results.

Results Display	
 **Display Thickness**	Displays the thinning simulation results in a color-coded plot.
 **Display Stress**	Displays the stress simulation results in a color-coded plot.
 **Display Strain**	Displays the strain simulation results in a color-coded plot.
 **Display Springback**	Displays the springback simulation results with before and after images of the model.
 **Output Springback Facet Body**	Outputs the springback facet body and creates a .spb file which includes node coordinate and element information.
 **Display Flatten Shape**	Displays the results for the flattened model with mesh elements.
 **Create Profile**	Creates an unform profile of the flattened sheet body.
 **Report**	Captures static images of the analysis results and represents them in an analysis report in the XML format. The XML file is generated using an XSL template.
Show Model Boundary	Hides or displays boundary edges of the model while displaying the results.
Settings	

Lets you modify the material, mesh, solver, and report parameters.
Material
Lets you modify the following material properties: **E (Elastic Modulus)** **Density** **Poisson's Ratio** **Yield Stress** **n (Hardening Coefficient)** **Initial Strain** **K (Strength Coefficient)** **r0 (Anisotropy Coefficient)** **r45 (Anisotropy Coefficient)** **r90(Anisotropy Coefficient)** Note **n (Hardening Coefficient)** is a material constant that is used while performing stress-strain analysis. It is calculated using the following formula: $\sigma = K \varepsilon^{n}$ where, σ = The applied stress on the material K = Strength coefficient ε= Strain n= Hardening coefficient The value of n lies between 0 and 1. A value of 0 denotes that the material is a perfectly plastic, and a value of 1 represents material that is perfectly elastic. The usual range of n for most metal is between 0.10 and 0.50.
Mesh

Element Type	Specifies the element type to be used for performing the surface meshing. **Triangle** Specifies the triangle element type for surface meshing. **Quad4** Specifies the quadrilateral element type for surface meshing.

	 Triangle mesh  Quad4 mesh
Infer Element Size	Appears when **Type** is set to **Entire Unform**, **Intermediate Unform**, or **Advanced Unform**. Infers the overall element size automatically.
Split Quad	Available when **Element Type** is set to **Quad4**. Sets the surface meshing method to split quad.
Attempt Free Mapped Meshing	Performs the mapping.
Mesh Size Variation (%)	Sets the value (as a percentage) for the tolerable variation between the largest and smallest elements of the mesh.
Small Feature (% of Element Size)	Sets the percentage of the element size for surface meshing of small features.
Solver	
Lets you modify the solver settings.	
Convergency Level	Specifies the level of convergence (low, medium, or high) of the iterative process. An iterative solution is considered to be sufficiently converged (or attained a particular level of accuracy) when any additional iterations produce negligible changes in the variable values.
Maximum Iteration Steps	Specifies the maximum number of steps to be performed by the solver to reach the specified convergency level.
Friction Coefficient	Specifies the coefficient of friction.

	During sheet metal forming processes, the coefficient of friction, if controlled properly, can generate the required stresses to deform the metal to the required shape and predict the fretting fatigue damage.
Save Analysis Results into Feature	Saves the specified formability analysis results into the One-Step feature.
Join Output Curves	Joins the resulting output curves.
Calculate Springback	Calculates springback, which predicts the springback results. Select this check box to create spring back faceted bodies.
Display Springback mode	
Available when **Type** is set to **Advanced Unform**. Lets you specify the springback display options. **Displacement** Displays the springback in 3D displacement. **Along X** Displays the springback displacement in the X direction. **Along Y** Displays the springback displacement in the Y direction. **Along Z** Displays the springback displacement in the Z direction.	
Report	
Display Thickness	Lets you specify the analysis result images you want to include in the XML report.
Display Stress	
Display Strain	
Display Springback	
Display Flatten Shape	

Allow View Change	Lets you change the view orientation before you export the analysis results.

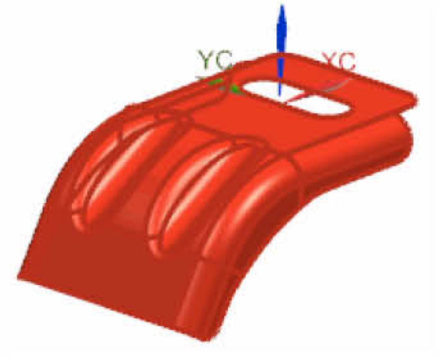
f3. Analyze sheet metal forming

Choose **Analysis** tab→**More** gallery→**Part** gallery→**Analyze Formability – One-step** .

In the **Analyze Formability – One-step** dialog box, from the **Type** list, select **Entire Unform**.

In the **Object Type** group, from the **Object Type** list, select **Face**.

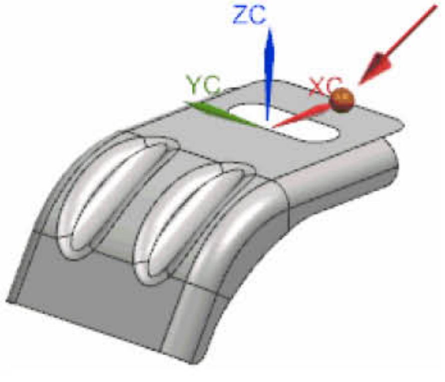
In the graphics window, select the faces to unform.



In the **Boundary Conditions** group, from the **Constraint Type** list, select the boundary condition.

For this example, select **Point to Point**.

In the **Constraint List** sub group, click **Specify Unform Point** and select the constraint point as shown in the following figure.



In the **Material** group, do the following:

Set the **Material List** to specify the location of the material library from which you want to use a specific material.

For this example, select **Library Materials**.

In the **Libraries** subgroup, make sure that the **Default Material Library** check box is selected

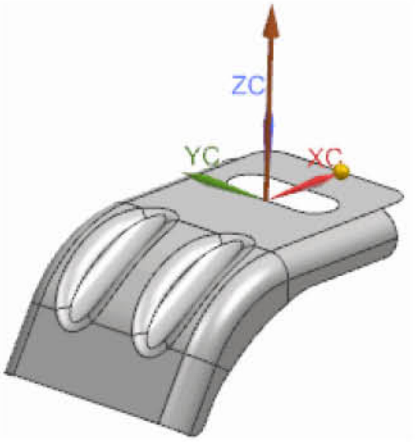
In the **Materials (Filtered)** sub group, select the required material from the list.

For this example, select **Steel**.

Note All the available materials in the standard **Default Material Library** are listed in the **Materials** list. Click **Display material properties for selected material(s)** , or **Inspect Material**  to view information about the selected material. You can also modify the material properties in the **Settings** group.

Specify the draw direction.

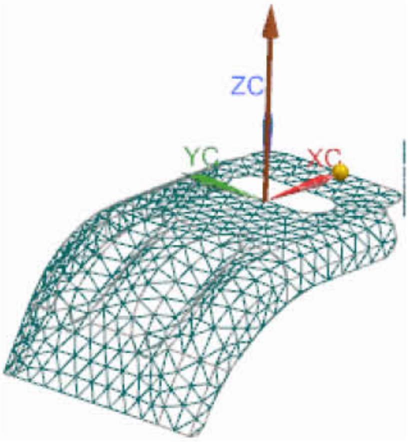
For this example, the draw direction is specified along the +ZC axis.



In the **Thickness** group, from the **Surface Type** list, select **Outer Surface**.

In the **Calculation** group, do the following:

Click **Mesh**  to perform the surface meshing.

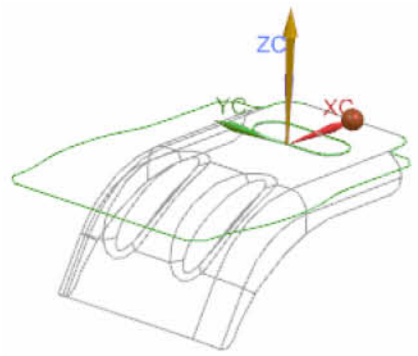


Click **Mesh Quality Check**  to examine the mesh quality.

In the message box that appears indicating the status of the mesh quality check, click **OK** if the mesh passed the quality check.

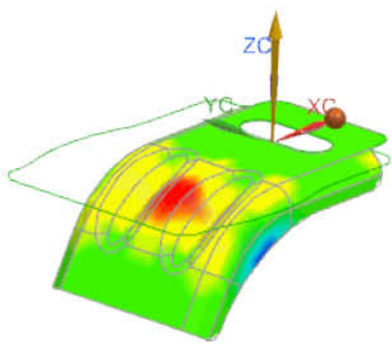
If the mesh quality is poor, you can change the surface meshing options in the **Settings** group, under **Mesh**, and then re-calculate the surface meshing results.

Click **Calculation**  to calculate the unforming results using the solver.

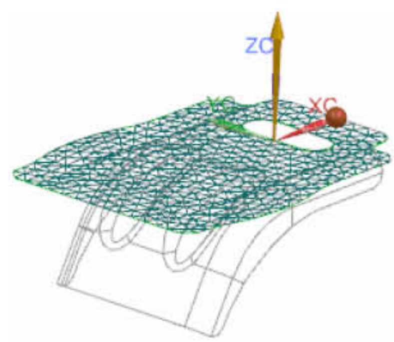


In the **Results Display** group, click **Display Thickness**  and **Display Flatten shape** .

The thickness and flattened shape unforming results are displayed in a color-coded plot.



Display Thickness results



Display Flatten Shape results

Click **Report** .

An external XML file with the XSL template is generated. The analysis results are automatically saved in the XML file.

In the **Settings** group, under **Solver**, select the **Save Analysis Results into Feature** check box.

This ensures that the formability analysis results are saved with the Onestep Unform feature.

Click **OK** or **Apply** to finish the analysis and save the analysis results with the Onestep Unform feature.

4.3.2. Create a strip layout

Detailed workflow 2: Create a strip layout

This workflow shows you how to:

Create the strip layout shown.

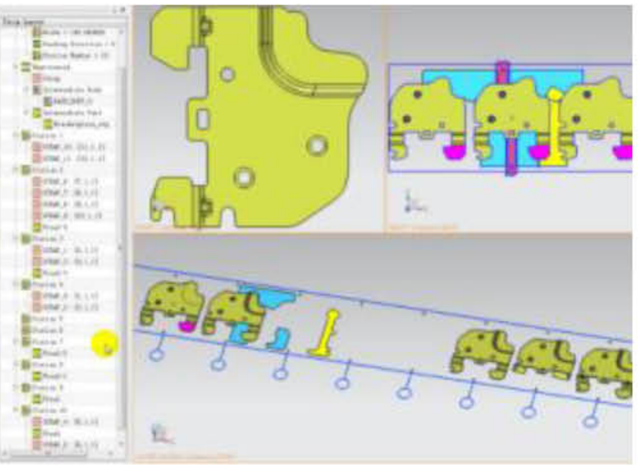
Specify the number of stations in the layout.

Position scrap at appropriate stations.

Position intermediate stages at appropriate stations.

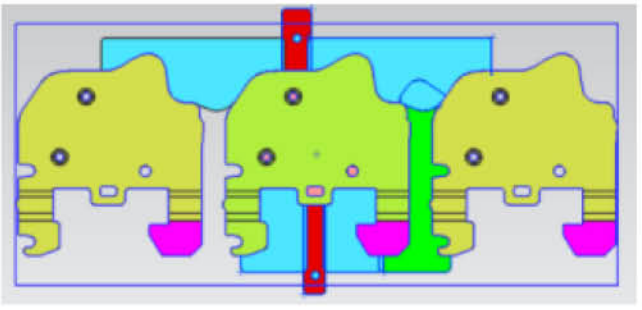
Run a simulation of the layout process.

You must have previously prepared your part, created intermediate stages, and designed scrap.

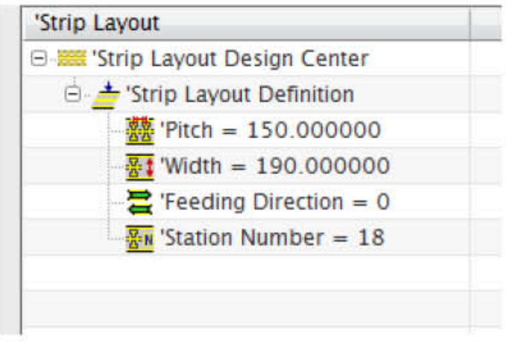


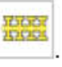
1. 

Start the Progressive Die application.

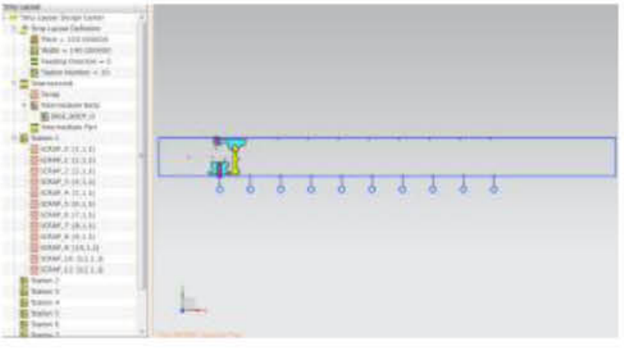
2. 

Open the top assembly.

3. 

Choose **Progressive Die Wizard** tab→**Main** group→**Strip Layout** .

NX displays the **Strip Layout Navigator**. All the steps in this workflow are done in this navigator.

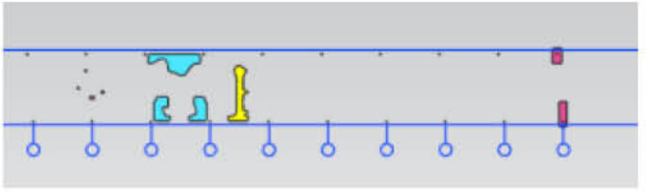
4. 

Set the number of stations.

Right-click **Strip Layout Definition** and choose **Create**.

This workflow uses 10 stations.

5.



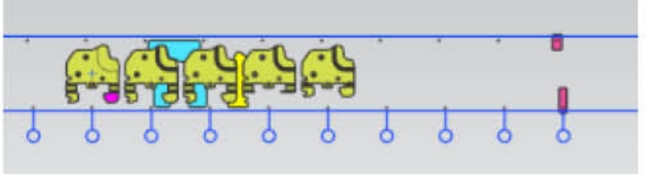
Position the scrap at appropriate stations.

Drag scrap from **Station 1** to the other stations.

For this workflow, the following are used.

Station Number	**Scrap**
Station 1	SCRAP_10 SCRAP_11
Station 2	SCRAP_6 through SCRAP_9
Station 3	SCRAP_1 SCRAP_3
Station 4	SCRAP_0 SCRAP_2
Station 10	SCRAP_5 SCRAP_4

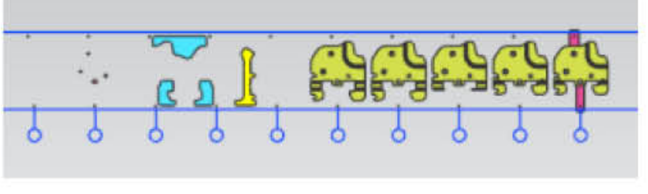
6.



Open and display the intermediate stages in the strip layout.

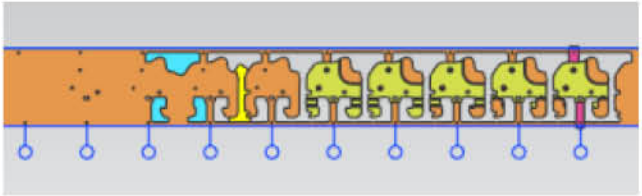
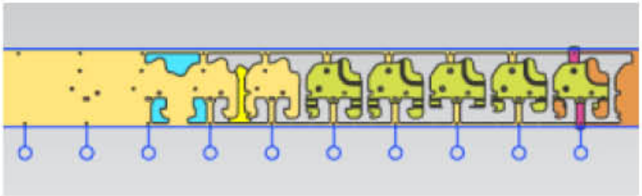
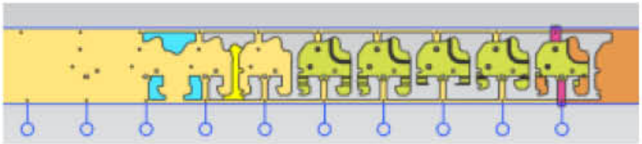
Right-click **Intermediate Part** and select the previously created intermediate part from the folder on your system.

7.



Drag the Final stages nodes under the appropriate Station nodes so that they represent the progress of the part through the die.

For example, in this workflow **Final** appears under both the **Station 9** and **Station 10** nodes. **Final–1** appears under the **Station 8** node, **Final-2** under the **Station 7** node and so on in descending order.

8.		Simulate the strip. Right-click **Strip Layout Definition** and choose **Simulate Piercing**. Enter the numbers for your From and To stations. Run the simulation.
9.		Remove the blank material to make the simulation resemble more closely the actual physical strip. Right-click **Strip Layout Definition** and choose **Remove Blank Material**. Enter the numbers for your From and To stations. Run the simulation.
10.		Close the **Strip Layout Design Center**

4.3.3. Create piercing inserts

Detailed workflow 3: Create piercing inserts

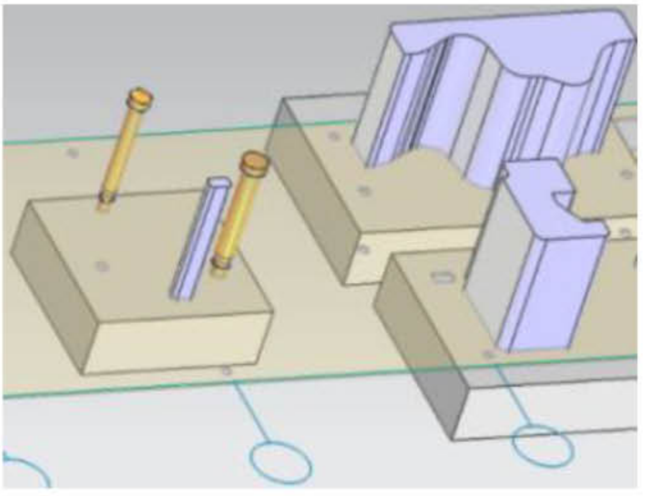
This workflow shows how to create:

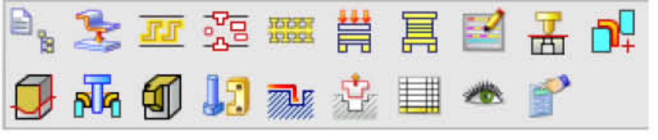
User defined punch and die blocks

Standard punch and die inserts

Die cavity and slug holes

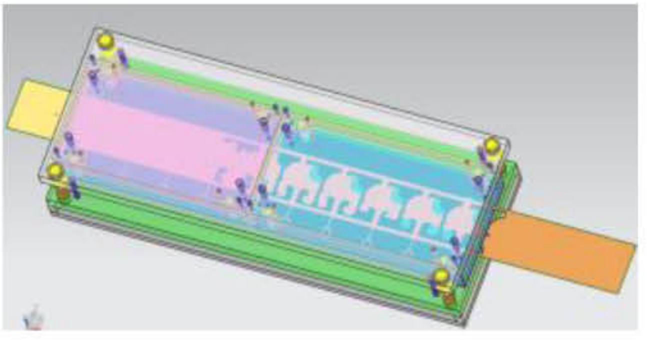
You must have previously created intermediate stages, designed scrap, created a strip layout, and added a die base.



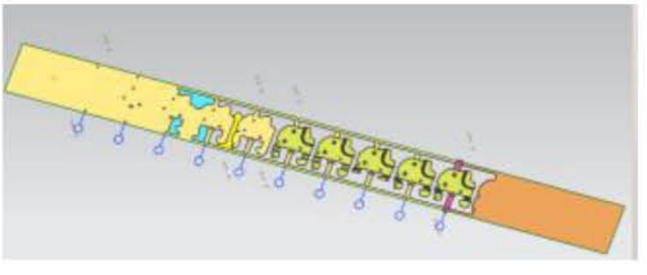
1. 

Start the Progressive Die application.

The commands that you will use are in the **Progressive Die Wizard** tab→**Main** group.

2. 

Open the top level assembly.

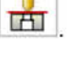
3. 

Choose **View Manager** .

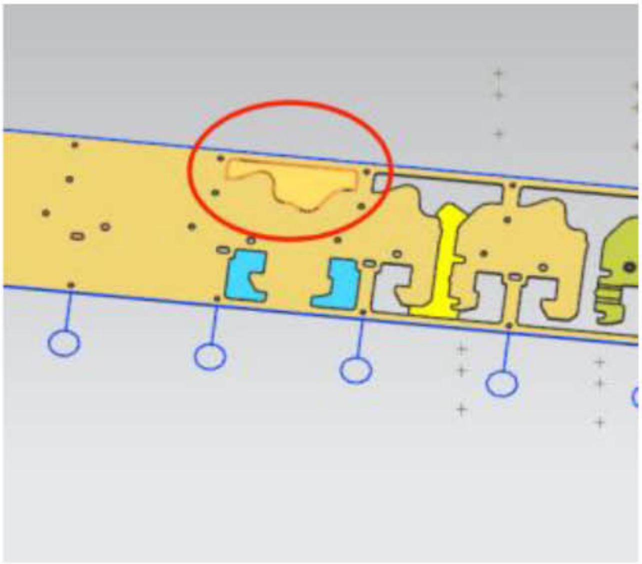
In the **View Manager Browser** hide the die base so that you see only the strip layout.

4. 

Create inserts for user defined scrap.

Choose **Piercing Insert Design** .

5.



In the **Piercing Insert Design** dialog box, do the following:

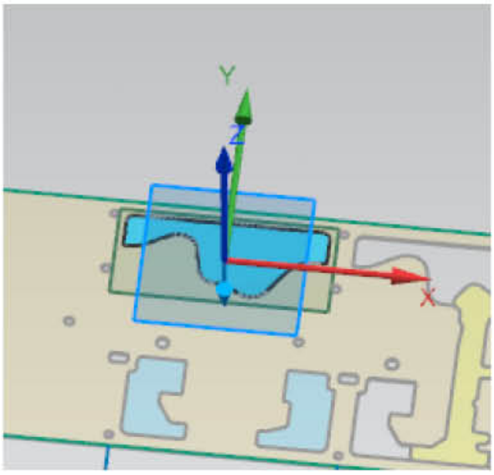
Set **Type** = **Die Insert**

Click **Select Scrap** 

Select the scrap.

For this workflow, **SCRAP_3** is selected.

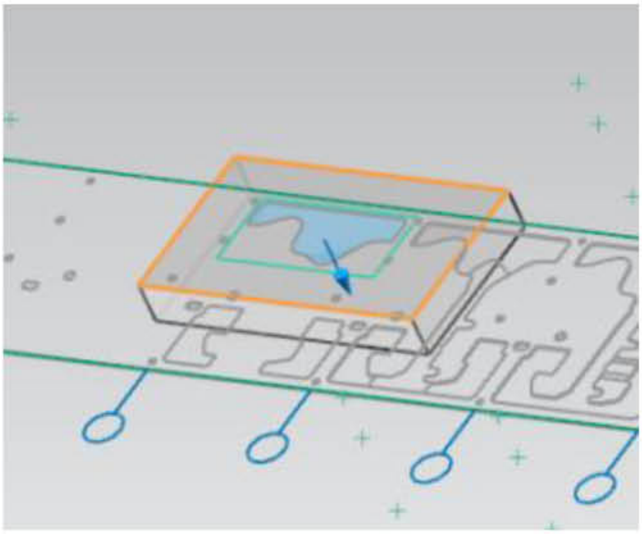
6.



In the **User Defined Insert** group, click **Create Datum Plane** .

Select the highlighted scrap to accept the default datum plane and open the Sketch task environment.

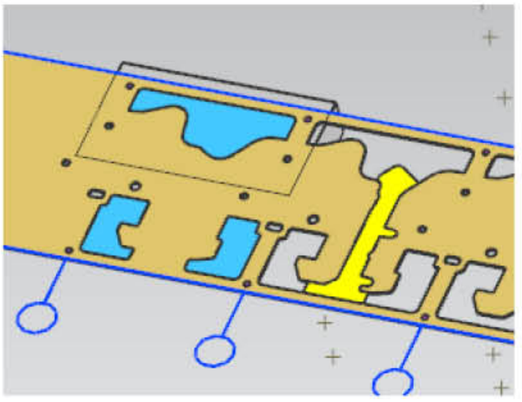
7.



Sketch a rectangle that contains the selected scrap.

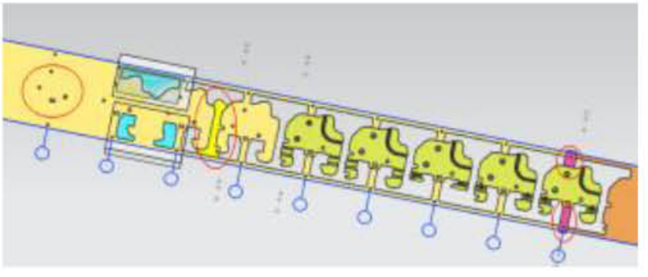
Click **Finish Sketch** 

8.



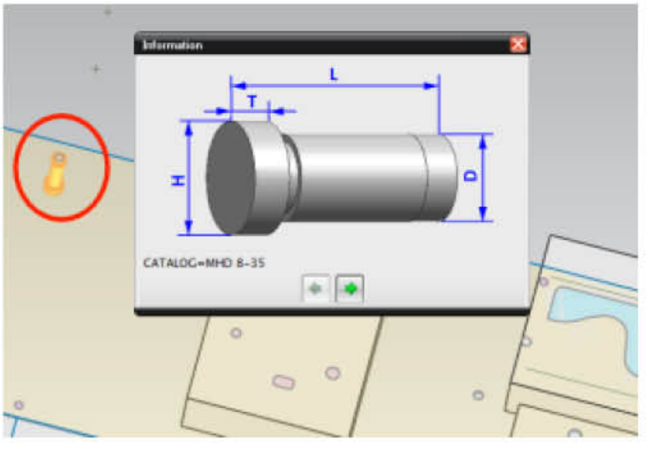
In the **User Defined Insert** sub-group, click **Create Insert** .

9.



Create additional die inserts for all the user-defined scrap, using the same design method.

10.



Create standard die inserts for individual hole scraps.

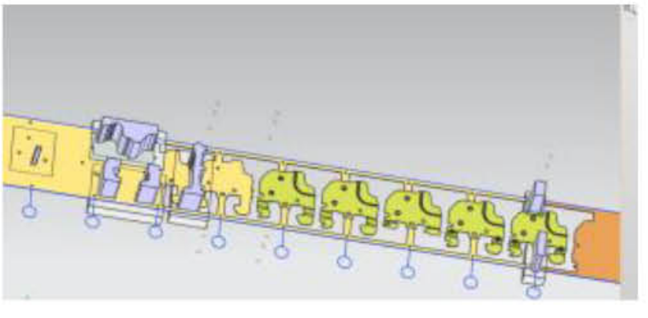
Select the scrap.

In the **Die Insert** group, click **Standard Insert** .

Use the **Standard Part Management** dialog box to add an instance of a standard die insert.

Add standard inserts for the remaining individual scraps in the layout.

11.



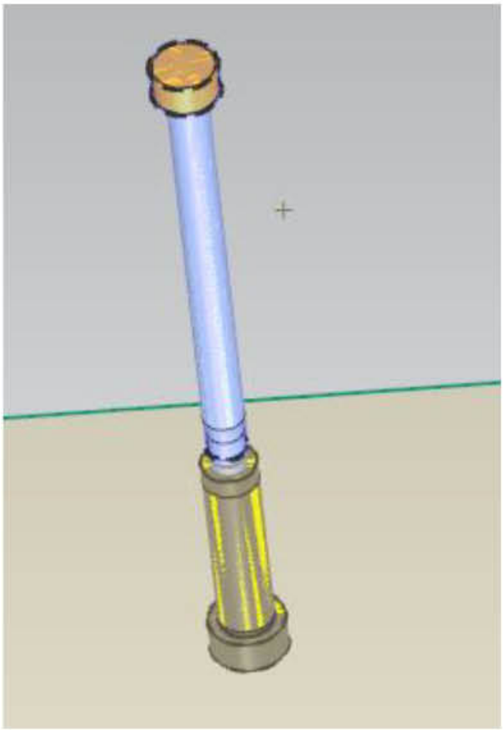
Create punch inserts to correspond to the user defined die inserts you created.

From the **Type** list, choose **Punch Insert**.

Click **Select Scrap** .

One by one select all the user defined inserts you created.

In the **Punch Insert** group, click **Create User Defined Punch**

12. 

Create a standard punch insert to correspond to the standard die insert you created for the remaining scrap.

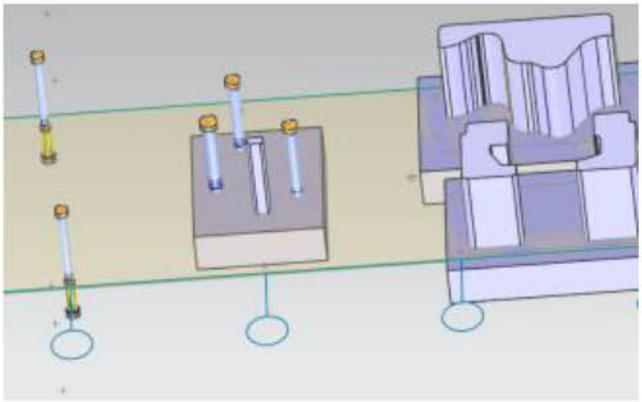
From the **Type** list, select **Punch Insert**.

Click **Select Scrap** .

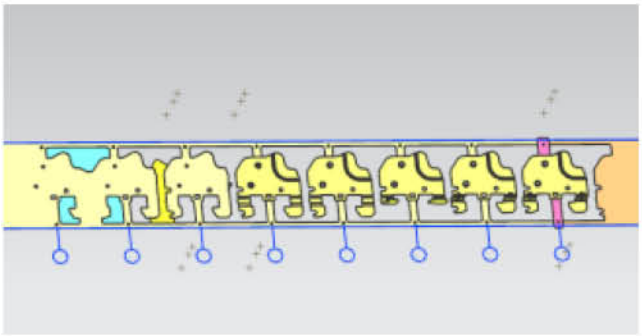
Select the scrap.

In the **Punch Insert** group, click **Standard Punch**.

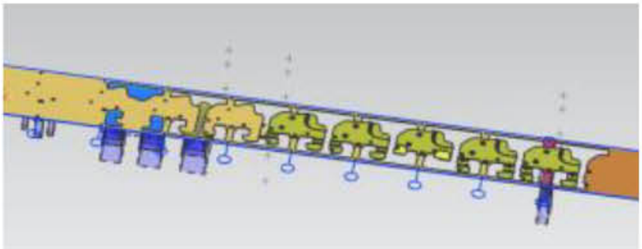
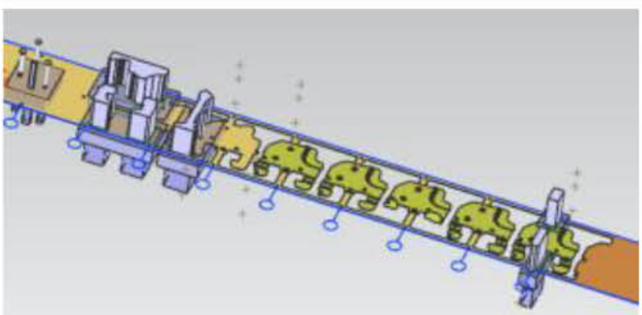
Use the **Standard Part Management** dialog box to add an instance of a standard punch insert.

13. 

Create standard punch inserts for the remaining scrap.

14. 

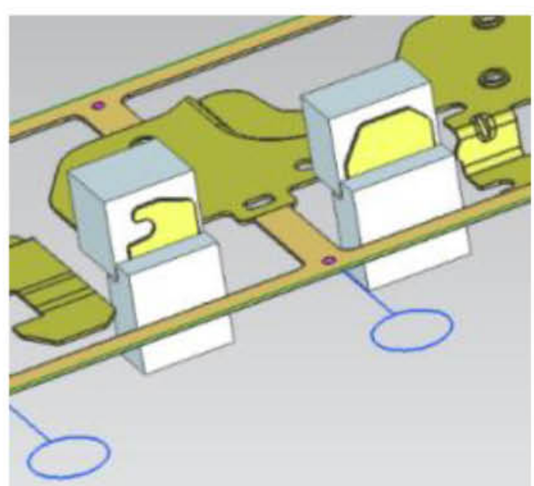
In the **View Manager**, expand the **Insert Group**, and hide the **Piercing** sub-assembly.

15.		Create die cavity and slug holes. The die cavity slug holes cut through the die block, backing plate, and die shoe to let the scraps drop from the strip. From the **Type** list, select **Die Cavity and Slug Hole** . Click **Select Scrap** . Select all the scrap on the strip layout. Click **Create Die Cavity and Slug Hole** .
16.		In the **View Manager**, show the **Piercing** sub-assembly.

4.3.4. Create bending inserts
Detailed workflow 4: Create bending inserts

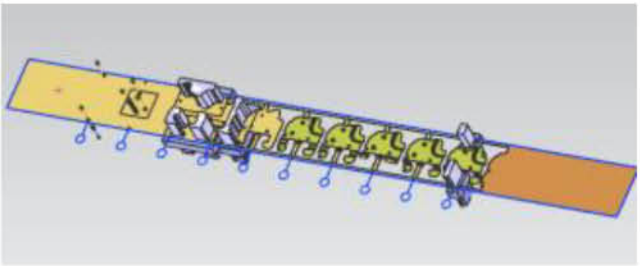
This workflow shows how to create die and punch bending inserts for regular straight brake sheet metal bends.

You must have previously created intermediate stages, designed scrap, created a strip layout, and added a die base.



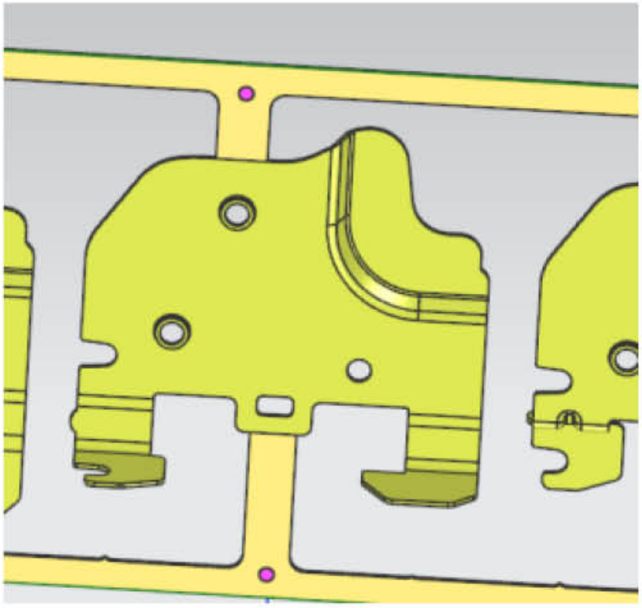
1. 

Start the Progressive Die application.

2. 

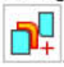
Open the top level assembly.

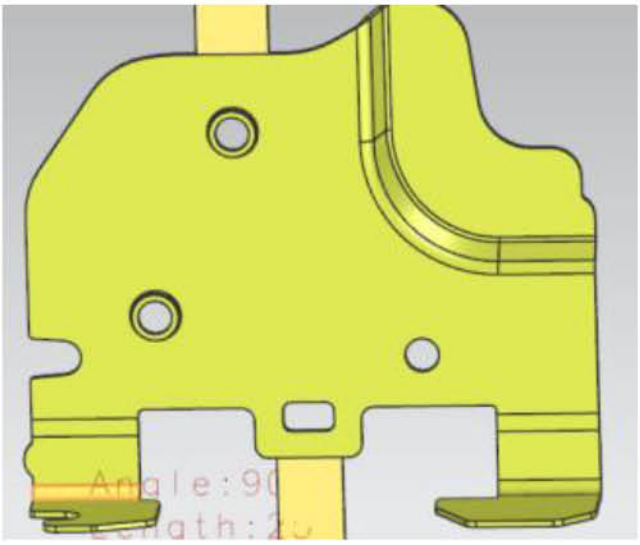
Hide the die base.

3. 

Zoom in on the stage that contains the standard straight-brake sheet metal bends.

For this workflow, the stage is **Final-2** at **Station 8**.

Choose **Progressive Die Wizard** tab→**Main** group→**Bending Insert Design** .

4. 

Set the following:

Type = User Defined Insert

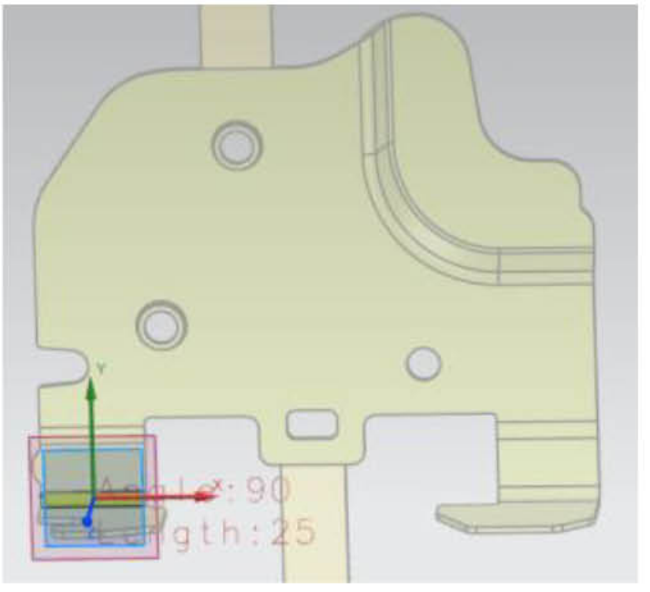
Bending Type =  **90 Degree Bending**

Insert Type =  **Die**

Position = Stripper Plate

Select the bend.

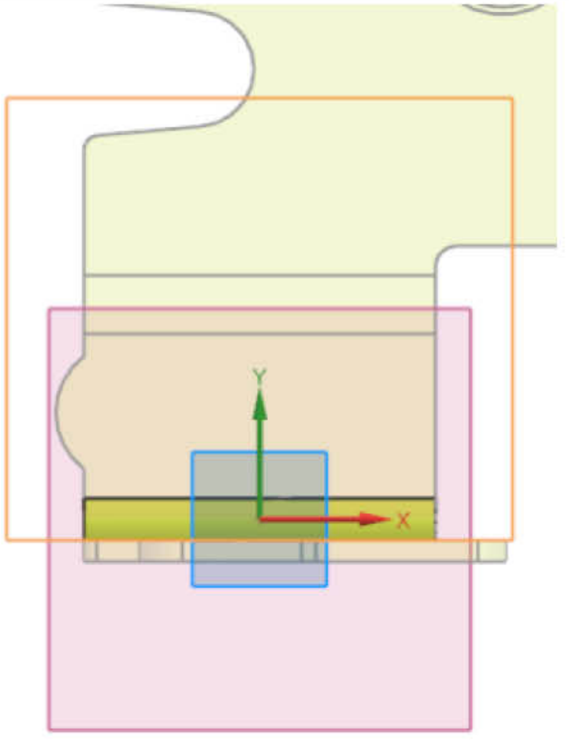
5.



Click **Create Datum Plane** .

Select the highlighted face to accept the default datum plane and open the Sketch task environment.

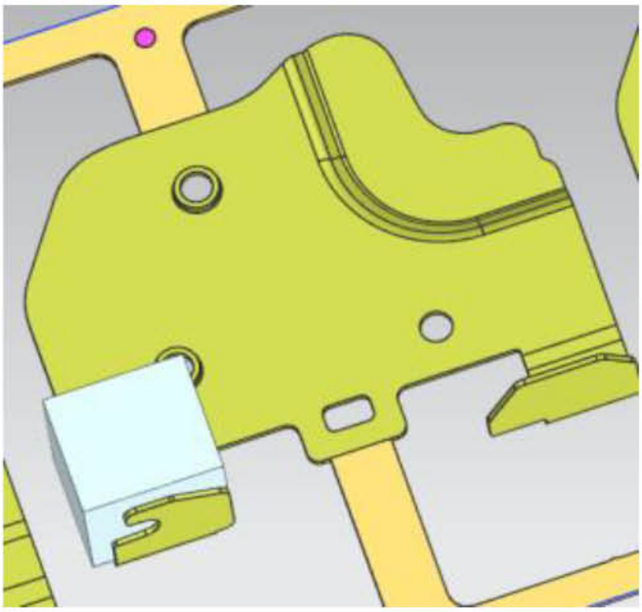
6.



Draw a rectangle as shown.

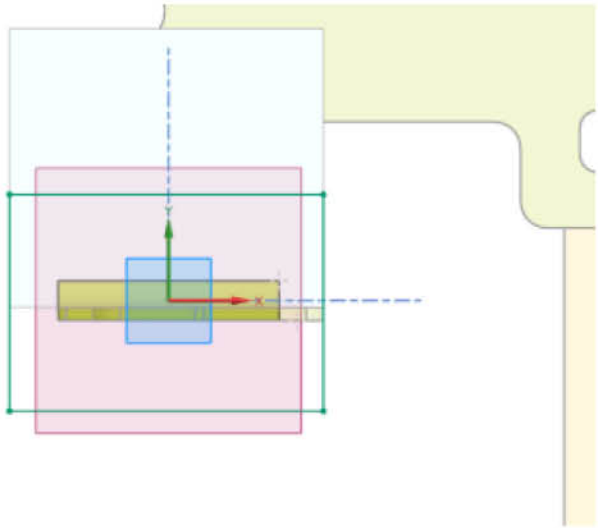
Make sure that the bottom of the rectangle is collinear with the lower bend line.

7.



Finish the sketch and click **Apply** to create the insert.

8.



Create the corresponding punch insert.

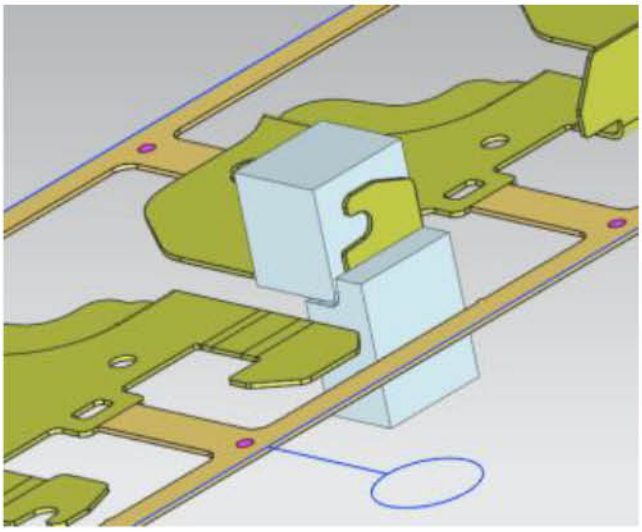
Rotate the strip so that you are looking at the back face of the part.

Change the **Insert Type** to **Punch**.

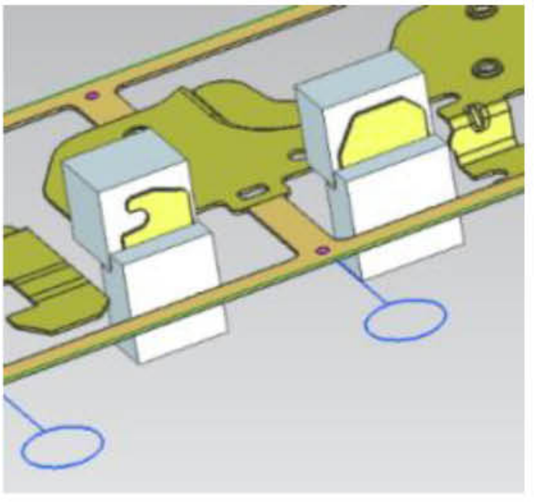
Select the back face of the same bend you selected for the die insert.

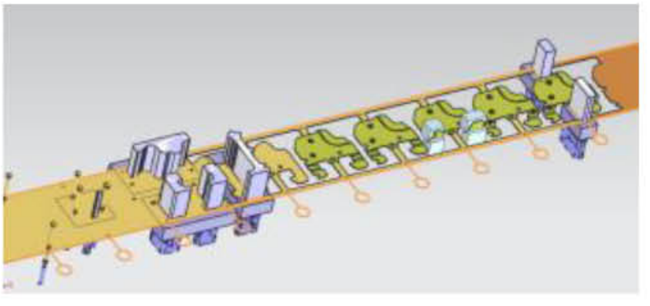
Create a datum plane and sketch a rectangle. Make the sides of the rectangle collinear with the edges of the die insert, as shown.

9.



Finish the sketch and click **Apply** to create the punch insert, as shown.

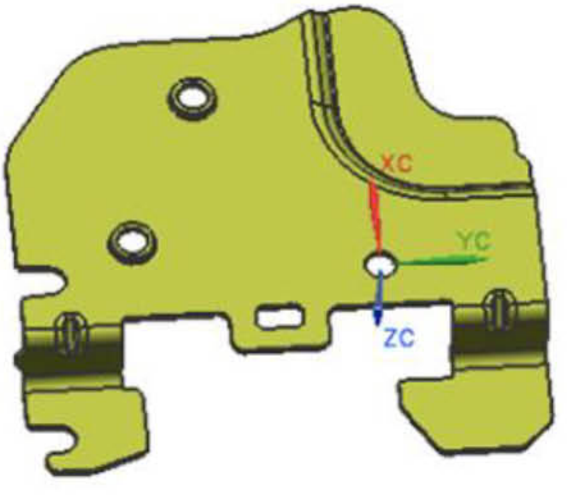
10. 



Create die and punch bending inserts for the remaining bend on **Final-2**, as shown.

4.3.5. Design Worlflow of a Progressive Die Set

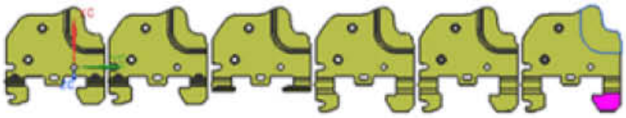
NX Progressive Die Wizard (PDW) provides a streamlined workflow for creating high volume sheet metal parts efficiently and cost-effectively.

1. 

Prepare your sheet metal part.

Design a solid-based sheet metal part in NX or import one from another 3D CAD system. PDW creates a progressive die based on a solid model. Repair the part if there are any apparent flaws or quality issues.

2.



Prepare intermediate stages.

Use the **Define Intermediate Stage**  command to specify the estimated number of stages you need to form the part and the initial stage number.

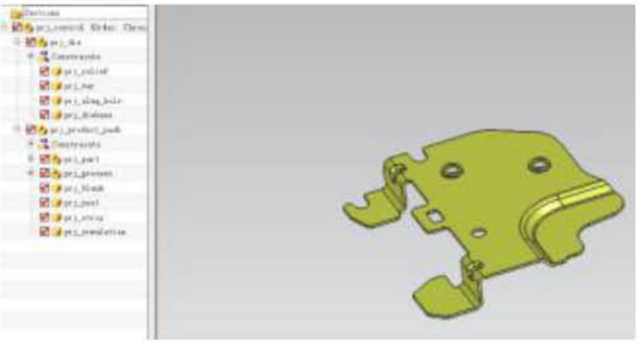
Use the **Direct Unfolding**  command to convert parts with standard straight-brake bends.

Use the **Bend Operation**  command to flatten standard sheet metal bends.

Use the **Universal Unform**  command to flatten straight-brake bends with deformation features such as beads, ribs, gussets, contour flanges, and burring.

Use the **Analyze Formability-One Step**  command for more complex freeform deformation areas.

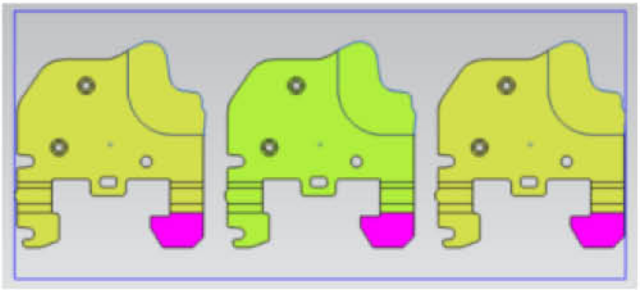
3.



Create a PDW project.

Use the **Initialize Project**  command to create a project for this part, name parts, and assign materials.

4.

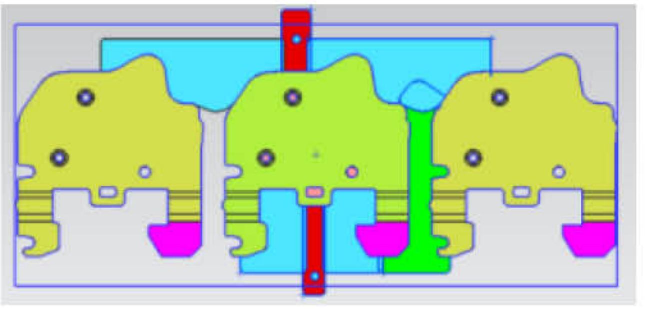


Create the blank layout.

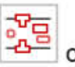
Use the **Blank Generator**  command to import a blank from the fully flattened part.

Use the **Blank Layout**  command to create a layout for blanks. The layout must allow for efficient use of material and adequate spacing between blanks to form the scrap.

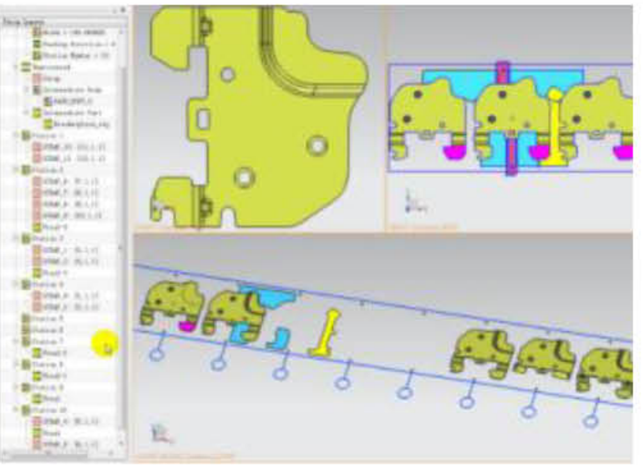
5.



Design scraps.

Use the **Scrap Design**  command to design scrap to remove.

6.

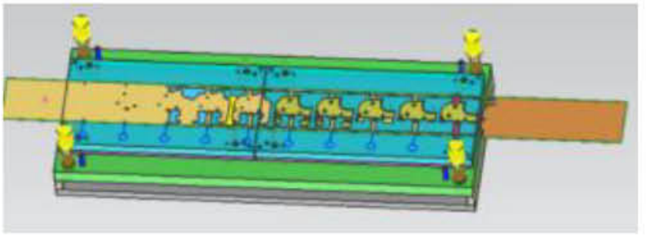


Create a strip layout.

Use the **Strip Layout**  command to define the number of stations and assign scrap to be removed at each station. You can drag scrap to assign it to a station.

You can also add intermediate stages into the strip layout and drag stages to the appropriate stations.

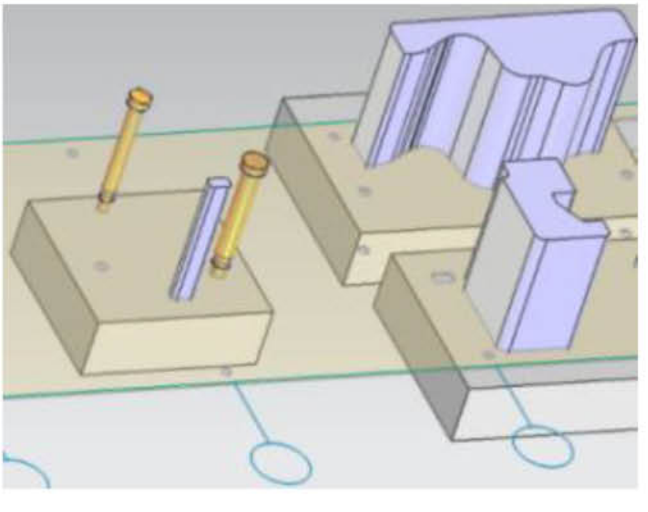
7.



Add a die base.

Use the **Die Base**  command to design and add a die base to hold the strip layout.

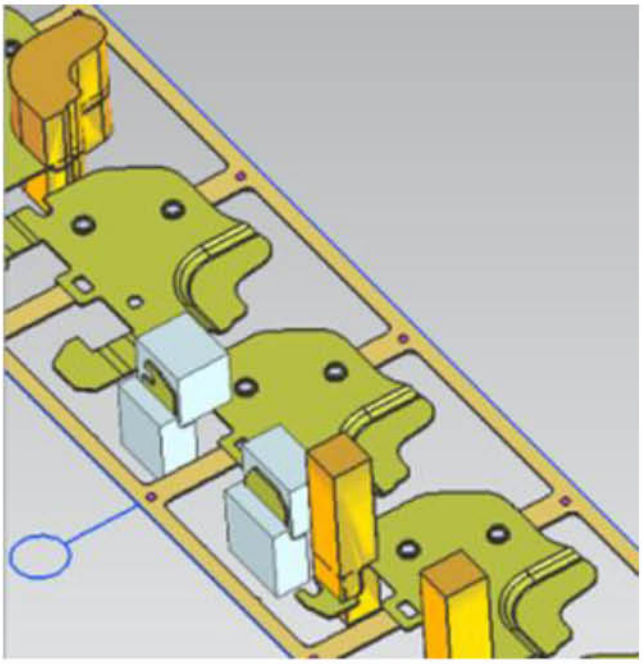
8.



Design cut punch and die inserts.

Use the **Piercing Insert Design**  command to create standard and custom shaped punches and dies to remove scrap from the strip layout.

9.



Design punch and die blocks for bending, forming, and burring.

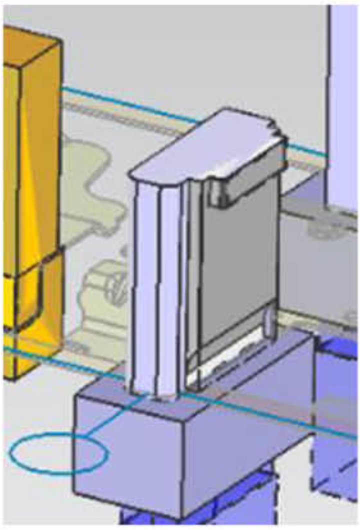
Use the following commands to create bends, burring, and other deformed shapes:

 **Bending Insert Design**

 **Forming Insert Design**

 **Burring Insert Design**

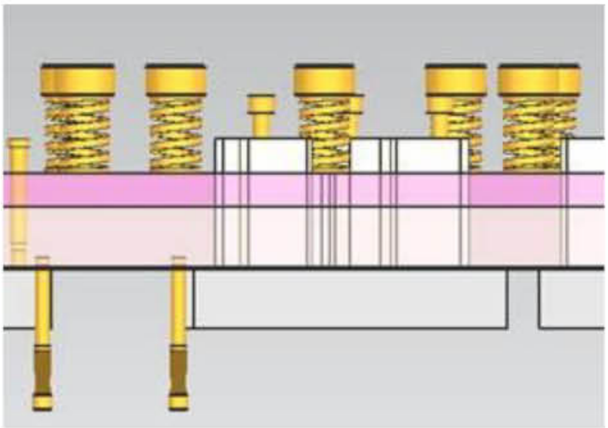
10.



Add auxiliary features to punches and dies.

Use the **Insert Auxiliary Design**  command to create strengthening features such as shanks for various inserts.

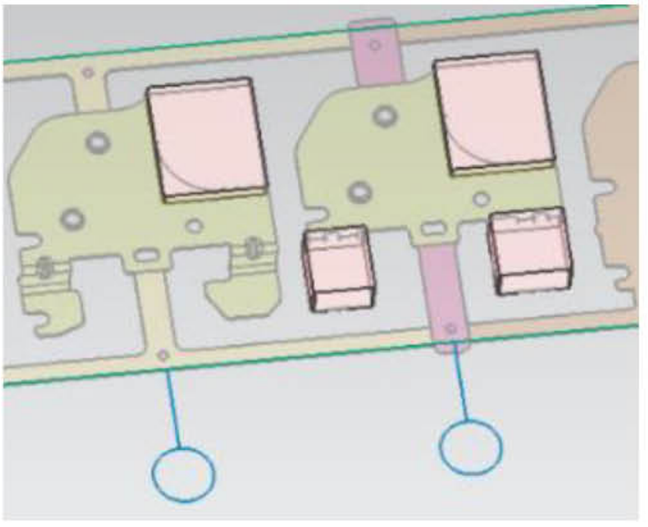
11.



Add standard parts.

From PDW libraries of standard parts in **Standard Parts**  or the **Reuse Library** , select standard parts such as lifters and springs.

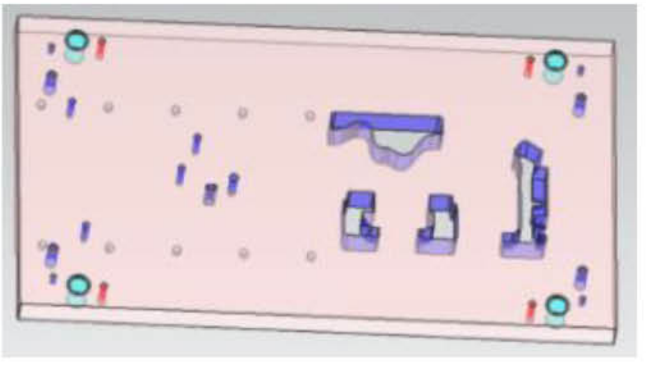
12.



Design relief pockets.

Use the **Relief Design**  command to create solid bodies to cut out pockets and holes in the die plate or strip plate.

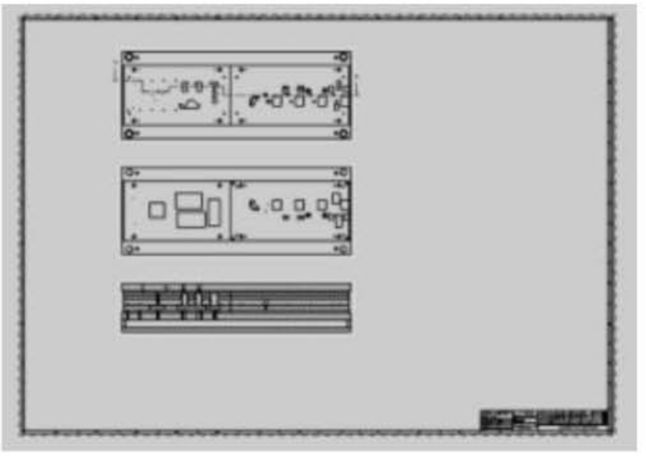
13.



Cut pockets.

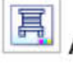
Use the **Pocket Design**  command to create pockets in the die plates to accommodate the standard parts and other components. The pockets are associative to the tool bodies.

14.

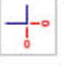


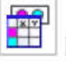
Do detailing.

Use the following commands to document your die assembly with drawings:

 **Assembly Drawing**

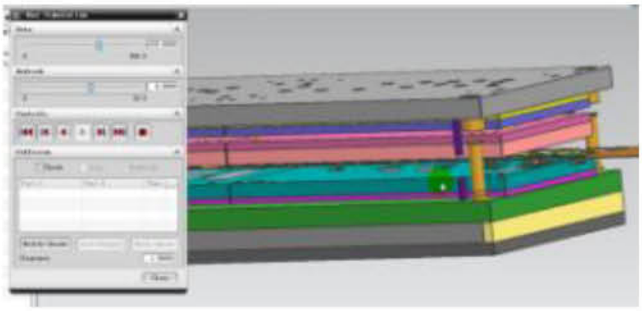
 **Component Drawing**

 **Auto Dimension**

 **Hole Table**

 **Hole Manufacturing Note**

15.

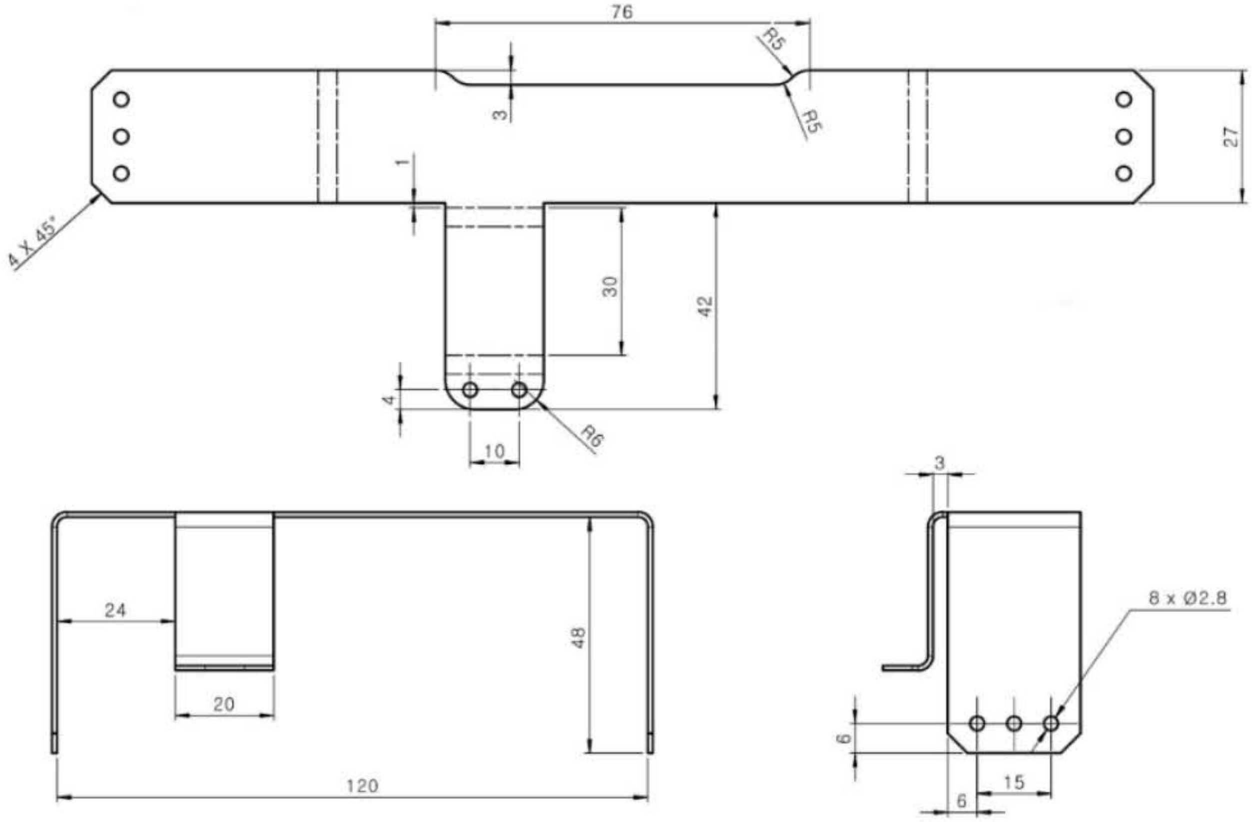


Perform static interference checks and dynamic interference checks.

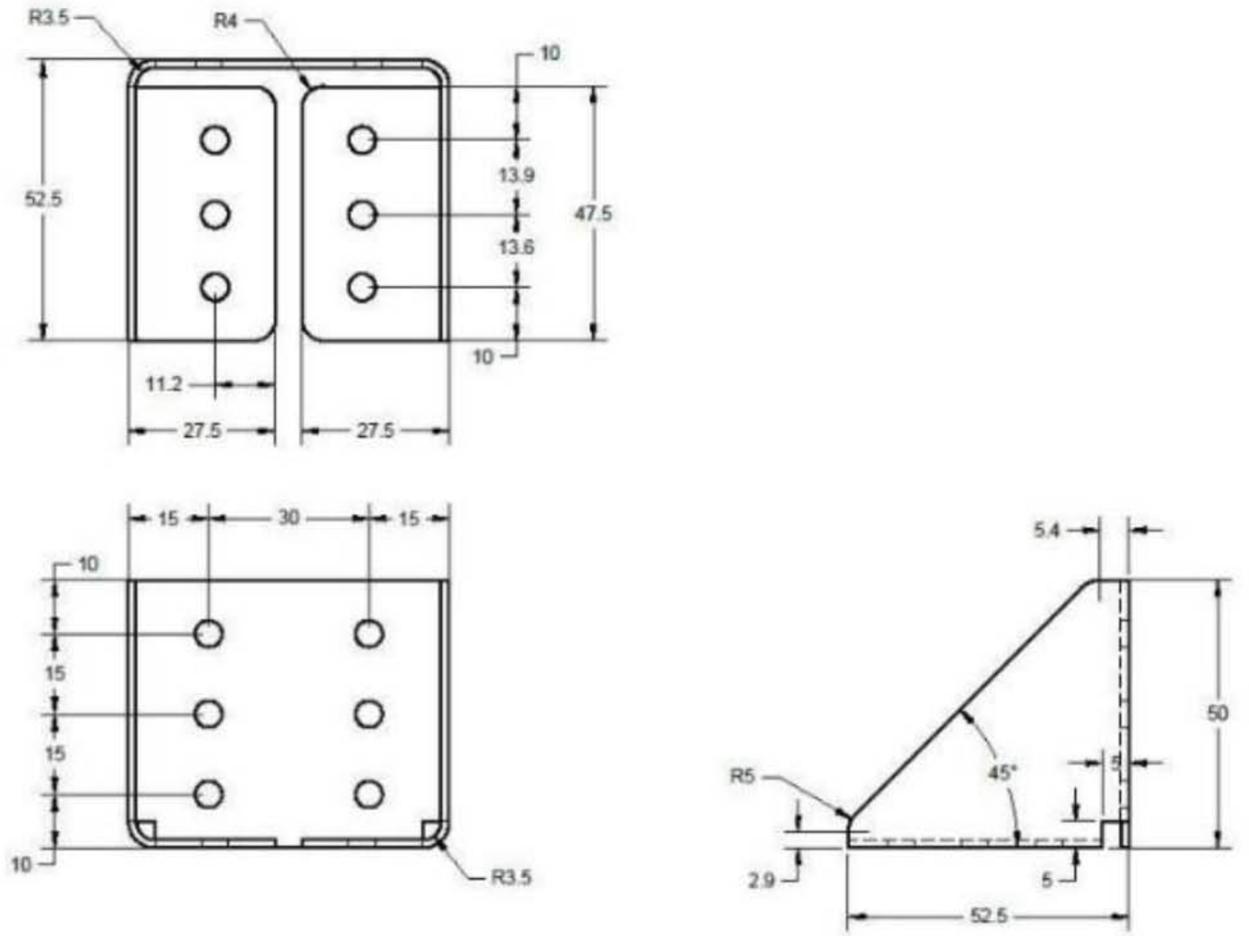
Use the **Tooling Motion Simulation**  command to simulate the progress of the strip through the die stations.

4.4. Application Assignments

4.4.1. Application Assignment 1



4.4.2. Application Assignment 2



UNIT 5 : INTEGRATED NX EASY FILL: PLASTIC INJECTION MOLD DESIGN THROUGH SIMULATION

Objective: By the end of this training unit, the trainees will be able to

Read and understand simulation and analysis results

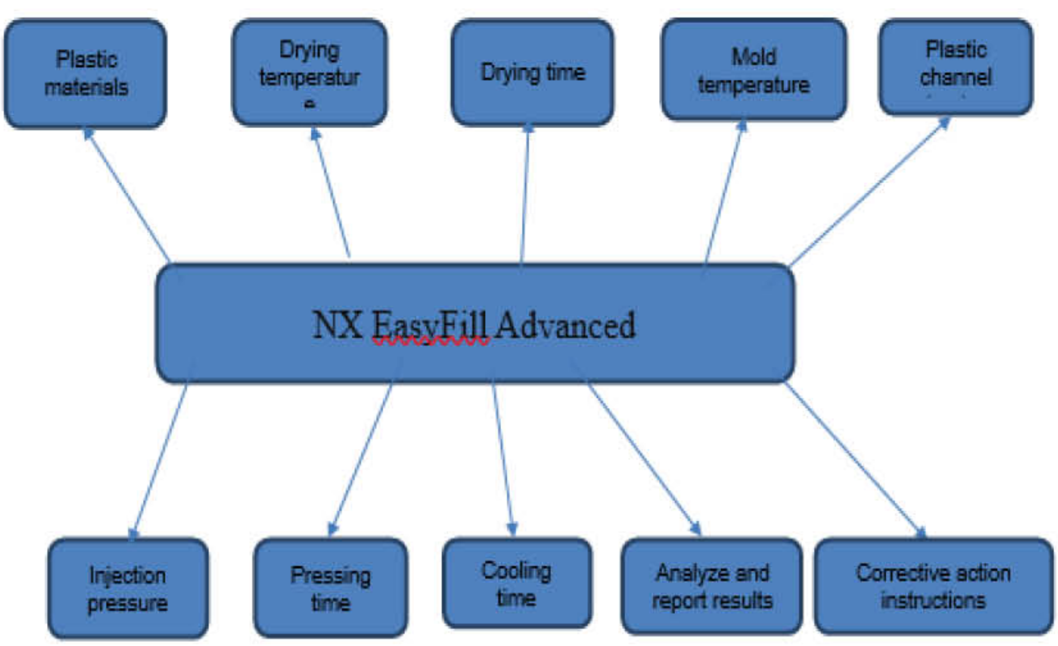
Analyze the causes and find ways to fix design errors

Understand the process of simulating plastic flow with NX software

Export the report of simulation results of plastic injection mold with NX software.

Contents:

5.1 Overview of nx easyfill advanced technology.

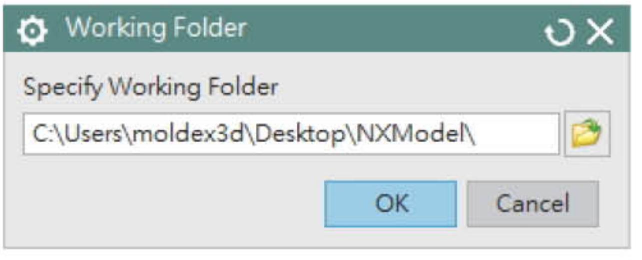


5.2 Product analysis process with NX Easy Fill.

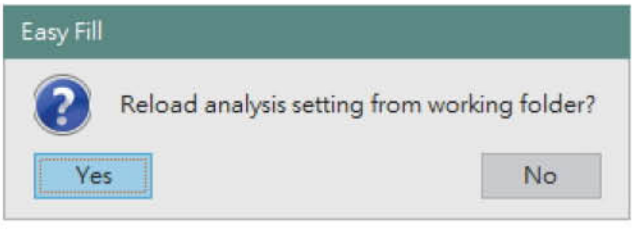
5.2.1 Set Working Folder

This section demonstrates how to set working folder.
Attribute of molding components and analysis result will be saved to the working folder.
This is optional, and the system will set current work part folder as working folder if user does not specify one.

Click **Set Working Folder** in the ribbon.



Click **Browse…** to select working folder.
Click **OK** to confirm your settings.

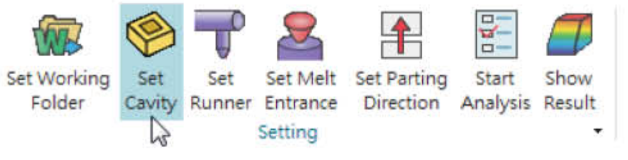


If there already exists data in the selected working folder, the massage will show up.

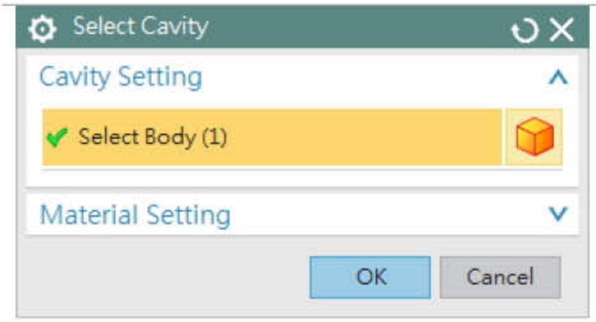
The attribute of molding components is saved as a .XEDS file, including the path of analysis result folder, with the same name as model file.

5.2.2 Set Cavity

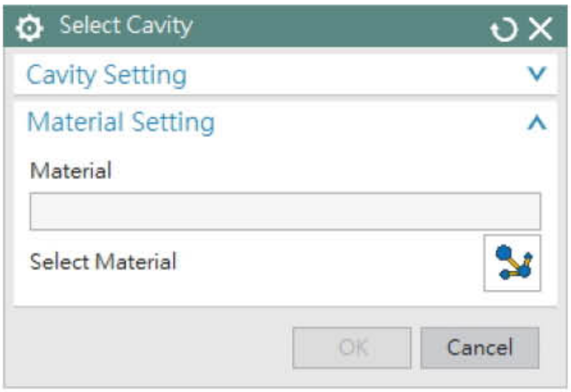
This section demonstrates how to set up a part and corresponding parameter.



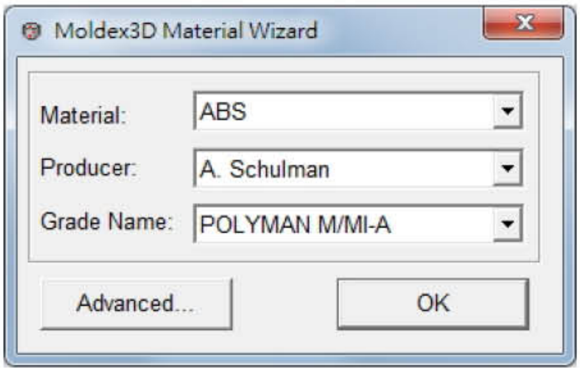
Click **Set Cavity** in the ribbon to start setting.



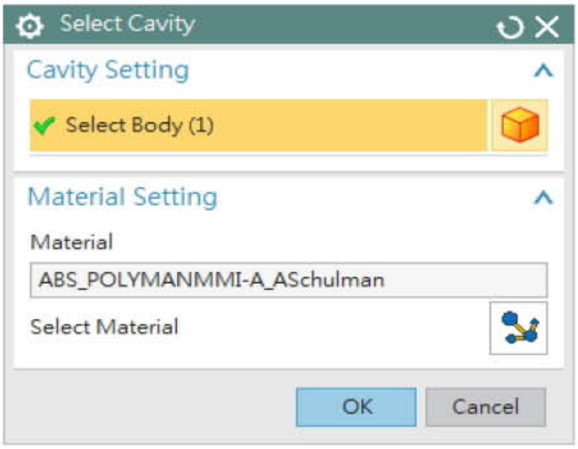
Click **Select Body** and then click on your model to make selection



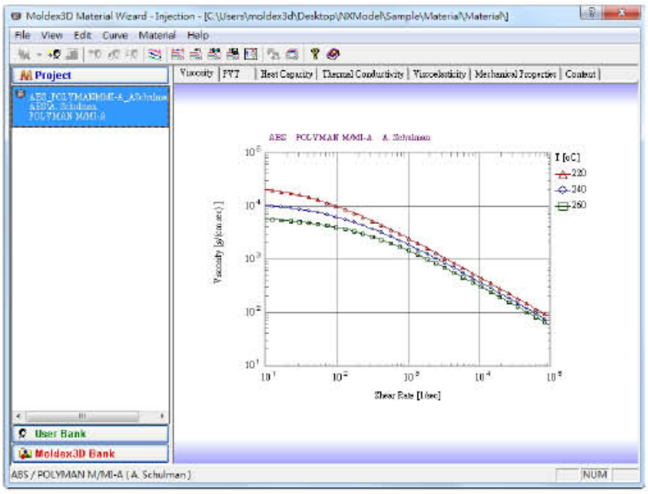
Click **Select Material** to activate Moldex3D Material Wizard.



Select a material from the pull-down list and click **OK**.
If the complete Moldex3D Material Bank is needed, click on **Advanced….**

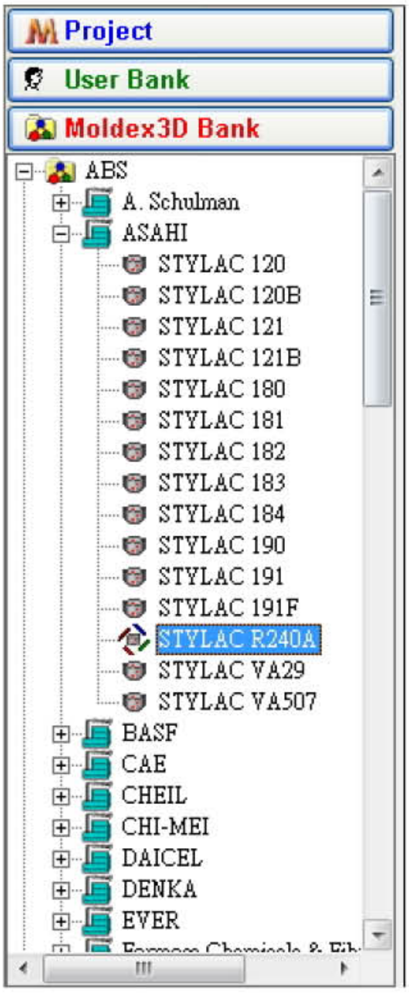


Click **OK** to apply the settings.

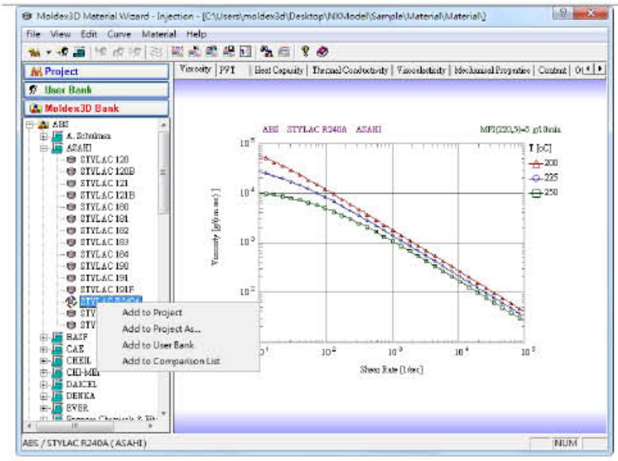


How to manipulate Moldex3D Material Bank

By clicking on **Advanced…** of material wizard, users can access Moldex3D Material Bank.



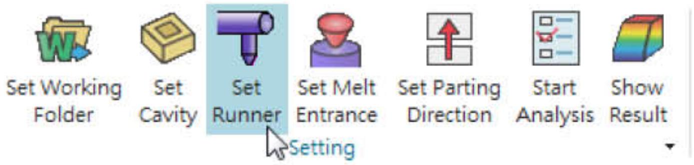
Click **Moldex3D Bank** on the left panel to load Moldex3D material bank. Find the material for the plastic melt.



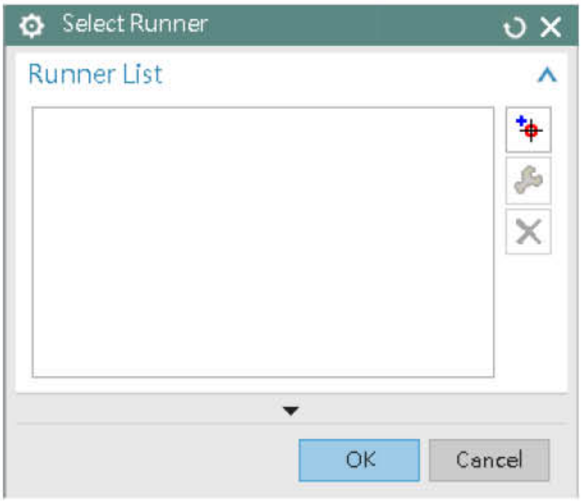
Right-click the target material and click **Add to Project** in the context menu. Click **OK** in the pop-up confirmation. In the next pop-up dialog, click **Yes** to add the material to User Bank.

5.2.3 Set Runner

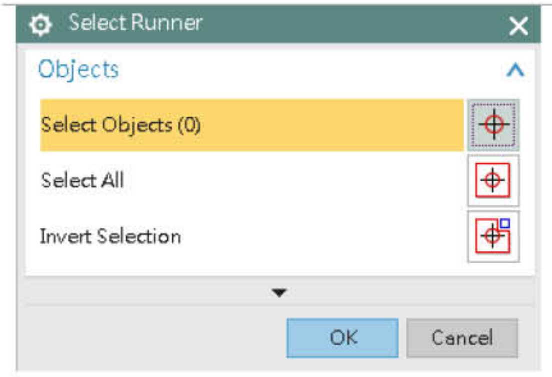
5.2.3.1 Set Runner - Create Runner System from Solid Bodies



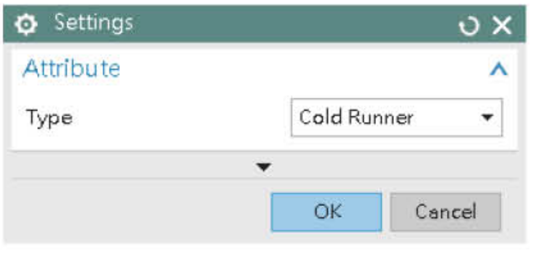
Click **Set Runner** in the ribbon to set up runner.



Click **Specify Runner** to set object with the runner attribute. It is supported to specify either solid runner or line runner.

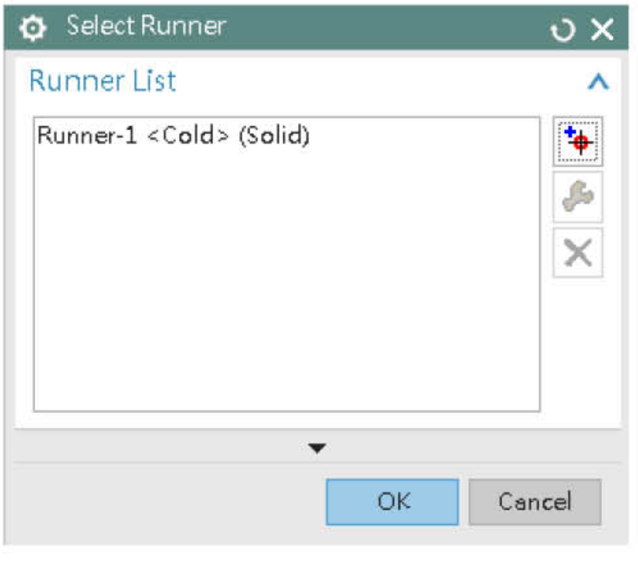


Click on the target solid object and click **OK** to confirm the selection.



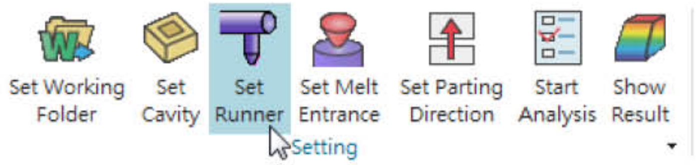
Specify runner type and click **OK** to confirm.

To modify runner design such as shape and sizes, use CAD functions.

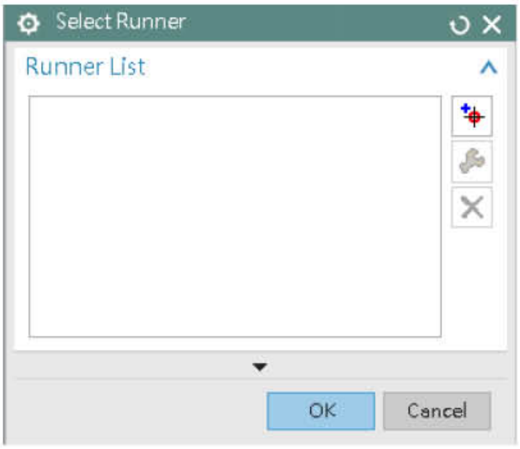


An entity representing the solid runner will appear in the tree menu, double click to edit runner type or click **delete** button on the right side of the list to delete.

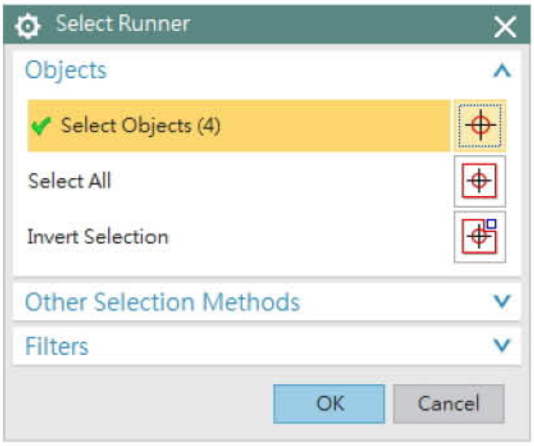
5.2.3.2 Set Runner - Create Runner System from Line Elements



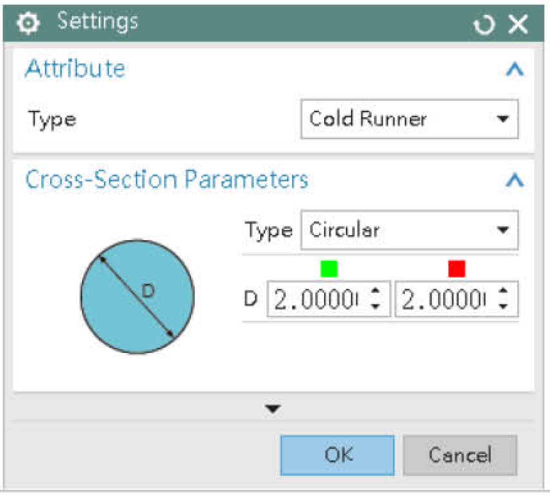
Click **Set Runner** in the ribbon to set up runner.



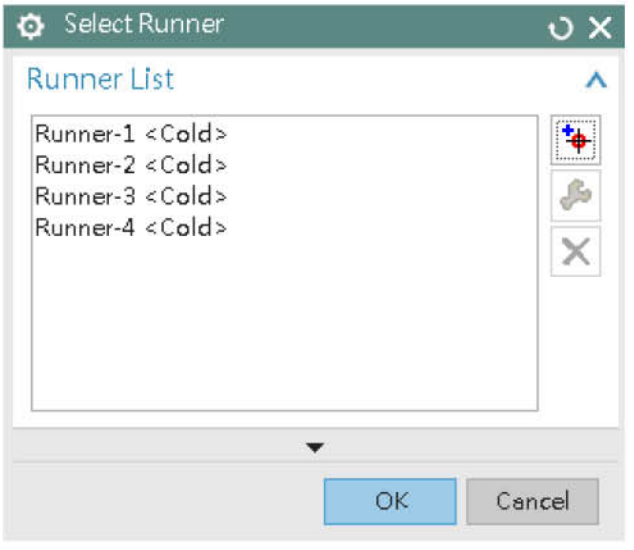
Click **Specify Runner** to set object with the runner attribute. It is supported to specify either solid runner or line runner.



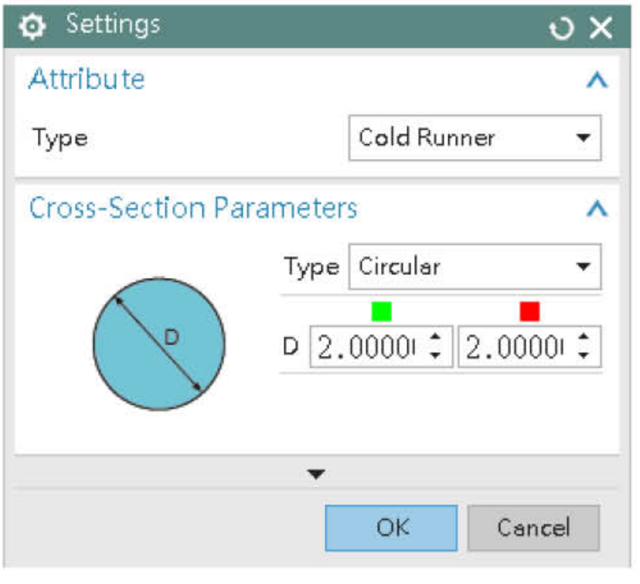
Click on the target line objects to select and click **OK** to confirm the selection.



Specify runner attribute, type and diameter and then click **OK** to apply.



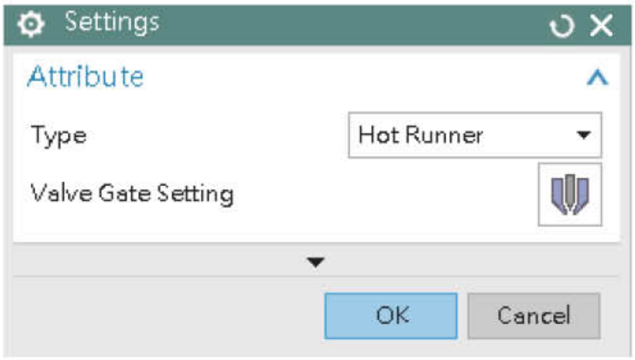
To modify an existing runner segment, double click its entity in Runner List, or otherwise click delete button on the right side of the list to delete.



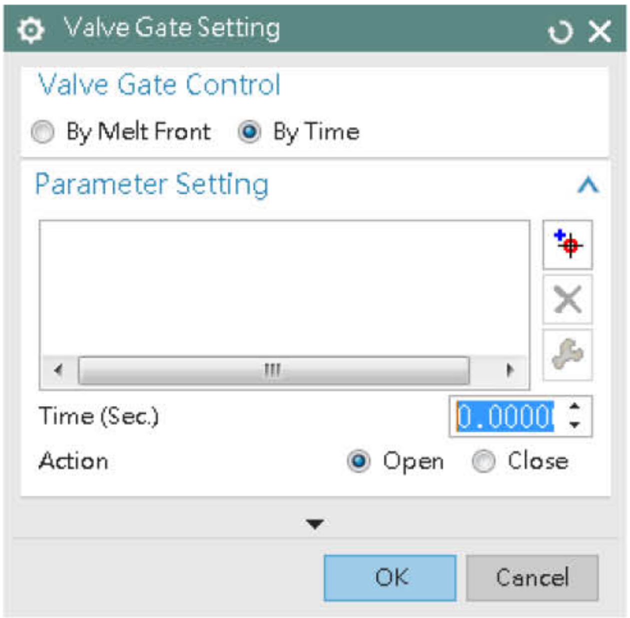
After change runner attribute, shape and size, click **OK** to confirm the settings.

5.2.3.3 Set Runner - Valve Gate Control

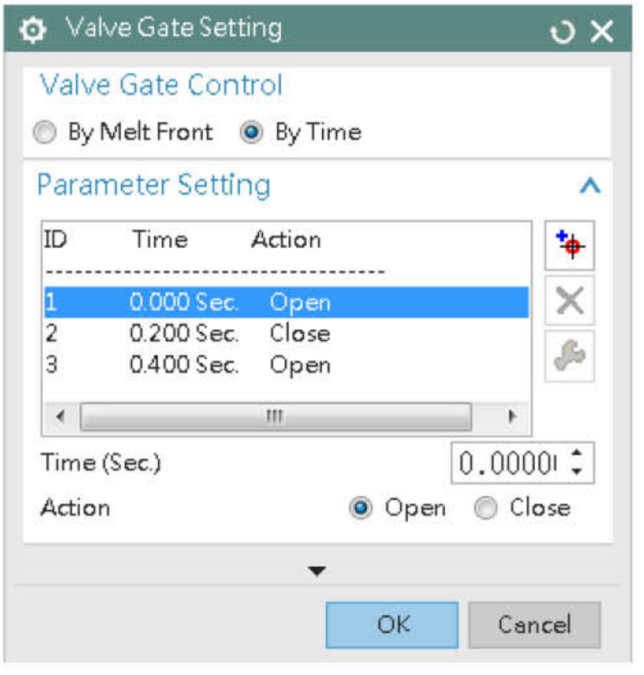
This section demonstrates how to set up valve gate control in hot runner attribute.



Select Hot Runner for runner attribute. Click **Valve Gate Setting** to access valve gate control settings.

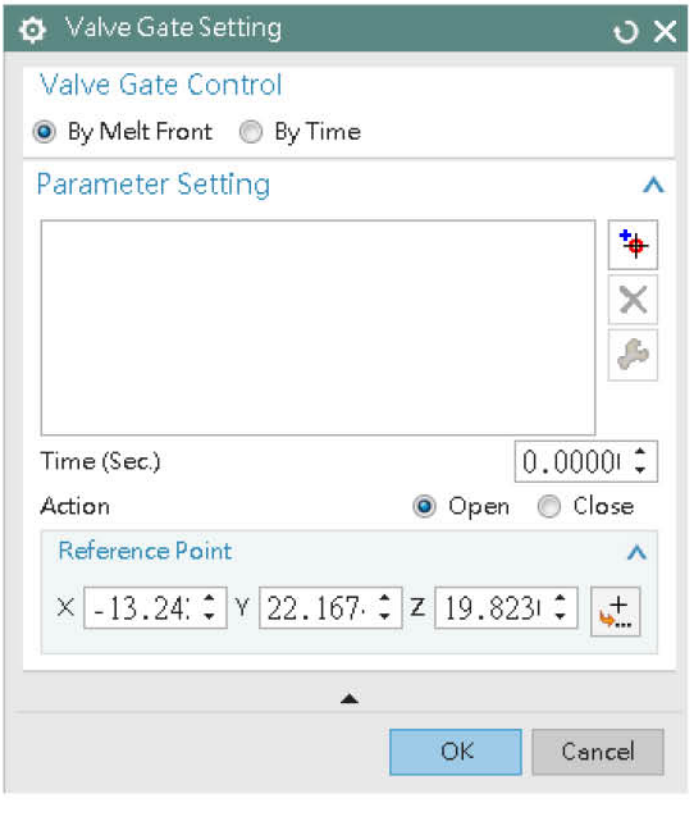


Specify whether trigger by Melt Front (melt front position) or Time (filling time).



Triggered by Time
Specify the gate action to open or close, and set the delay time.

Click **Add** to add a new control point.
Click **Modify** to modify a new control point.
Click **Delete** to delete an existing control point.



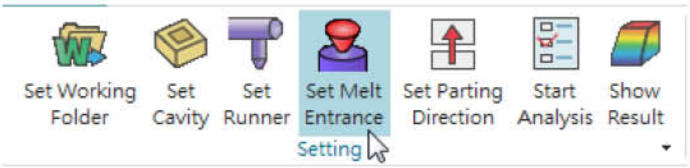
Triggered by Melt front

Enter coordinates or click [icon] to select a specify point for trigger point. Select the gate action, open or close, and set the delay time.

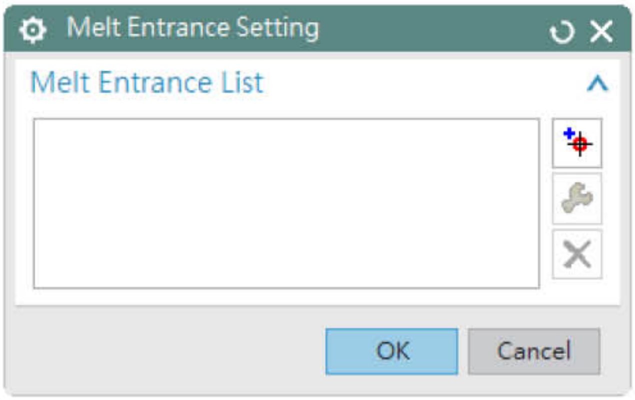
Click **Add** to add a new control point.
Click **Modify** to modify an existing control point.
Click **Delete** to delete an existing control point.

5.2.3.4 Set Melt Entrance

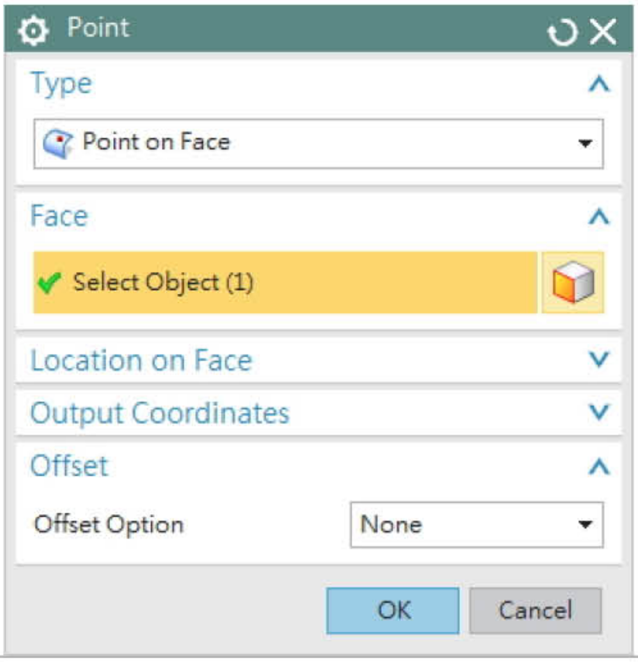
Melt entrance is essential to the simulation. This section demonstrates how to set up the melt entrance in the model.



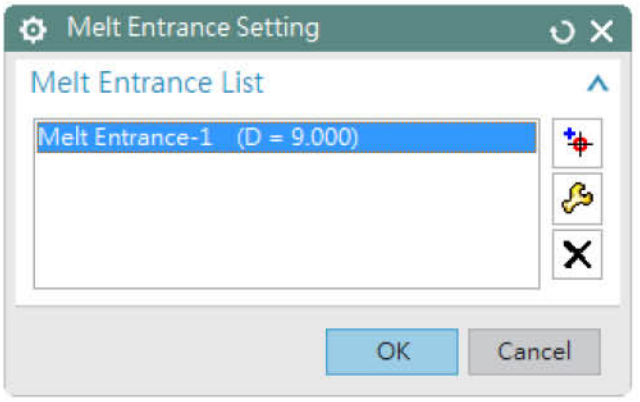
Click **Set Melt Entrance** in the ribbon menu to define a melt entrance.



Click **Specify Melt Entrance Position** to create melt entrance and click **OK** to proceed.

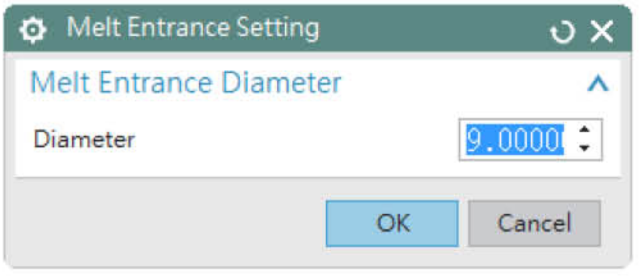


Click on the surface of runner or part to create melt entrance and click **OK** to proceed.



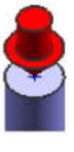
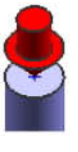
An entity will appear in the list upon completion of set-up.

Click  to modify any previous settings, or click the  on the right side of the list to delete.



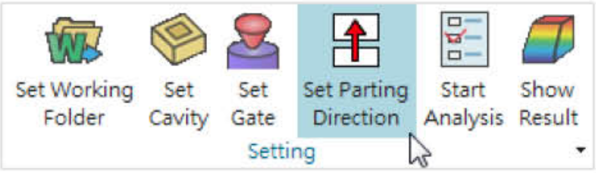
Note: if the melt entrance is directly set on the part, the diameter of the melt entrance is adjustable.

There are some differences among building melt entrance on the runner with solid bodies, on the runner with line element, by runner wizard, and directly on the part surface. Building melt entrance on the runner with line element is the same as using runner wizard to create melt entrance, which its diameter is defined by that of runner end point. The melt entrance on solid body is defined by the surface area of runner end. In addition, a complete runner system, including gates and runner, is prerequisite for creating melt entrance on solid body. Users also can add melt entrance directly on the part surface, and the diameter is defined by that of a hypothetical gate on the cavity surface. The comparison table is shown below.

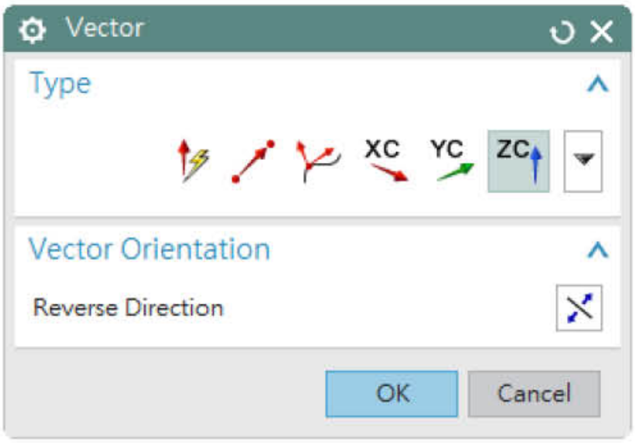
Building melt entrance on the runner with line element	**Building melt entrance by runner wizard**	**Building melt entrance on the runner with solid bodies**	**Building melt entrance on the part surface**
Its diameter is defined by that of runner end point	Its diameter is defined by that of runner end point	Its diameter is defined by the surface area of runner end	Its diameter is defined by the diameter of a hypothetical gate
			

5.2.4 Set Parting Direction

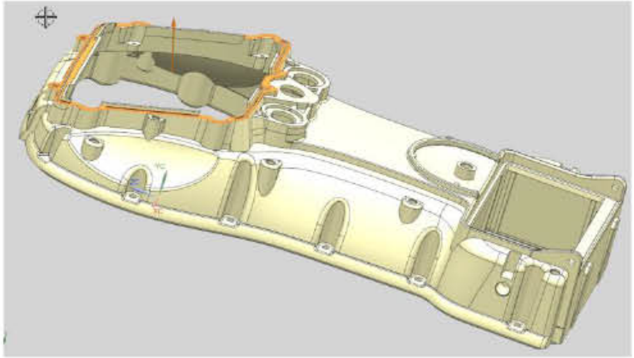
A customize mold open direction function.



Click **Set Parting Direction** on the ribbon to open the setting dialogue.



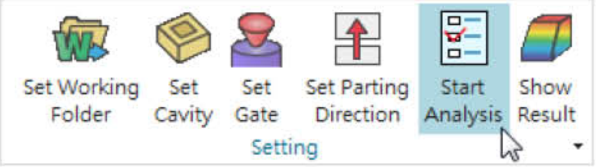
Type
Select from many types of direction vector selection.
Vector Orientation
Use **Reverse Direction** to flip the parting direction.
Click **OK** to apply the setting.



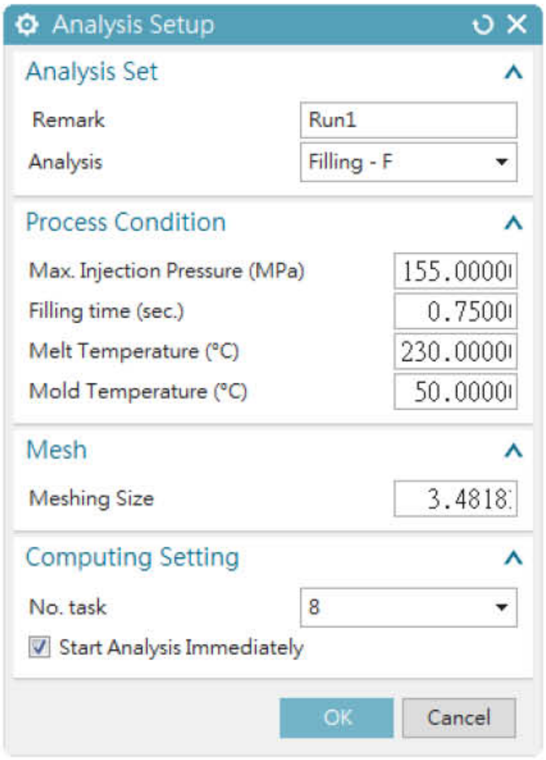
The specified parting direction will be shown in orange arrow.

5.2.5 Start Analysis

This section will demonstrate the analysis setup in default settings. Before starting an analysis, please make sure all the model components of injection molding are properly set.



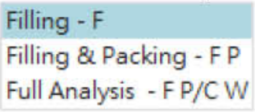
After setting up all the model components, click **Start Analysis** on the ribbon

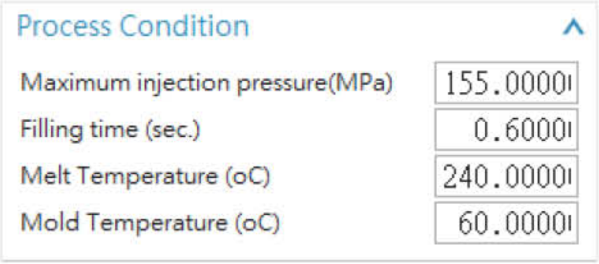


Analysis Setup
There are four sections in the analysis setup dialog: Analysis Set, Process Condition, Mesh, and Computing Setting.

Analysis Set
Add a remark for this Analysis.

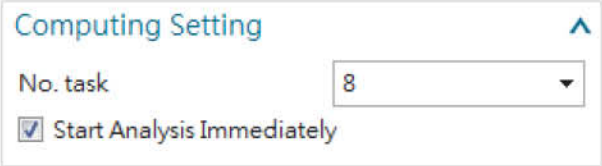
Choose the molding process you would like to simulate from the pull-down menu.





Process Condition:
The default process condition is set based on the cavity dimension and the material selection.
In this example, **Filling Analysis** is chosen; Only Maximum injection pressure, Filling time, Melt temperature and Mold temperature can be modified. Packing time is adjustable if the Analysis Sequence is selected as **Filling & Packing**

Mesh
Generate mesh with customized mesh size so that tiny feature would be remained.

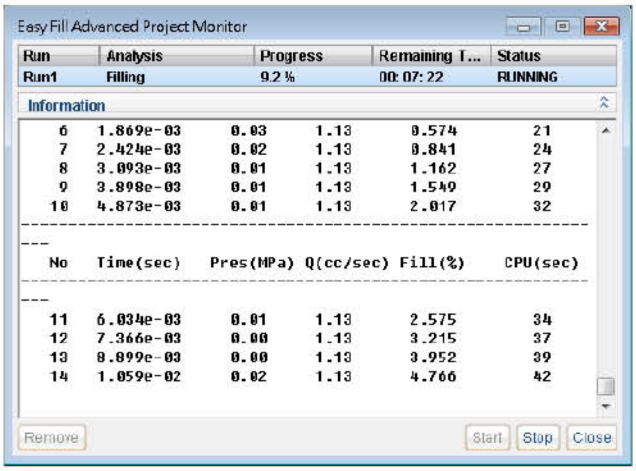


Computing Setting
User can set the number of working core to run analysis; the number of working core can't exceed max core number of CPU.

If **Start Analysis Immediately** is unchecked, current analysis would be exported to project monitor list. Multiple run data can be added for simultaneous simulation.

After setting above-mentioned process conditions, click **OK** to start or click **Cancel** to cancel it.

5.2.6 Start Analysis - Analysis Monitor

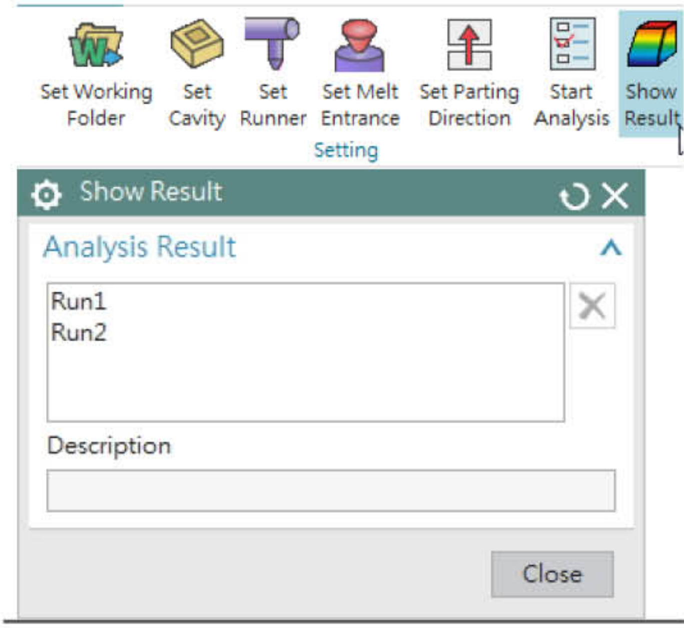


The analysis monitor shows the run ID, analysis sequence, estimate time, calculation percentage, and status.

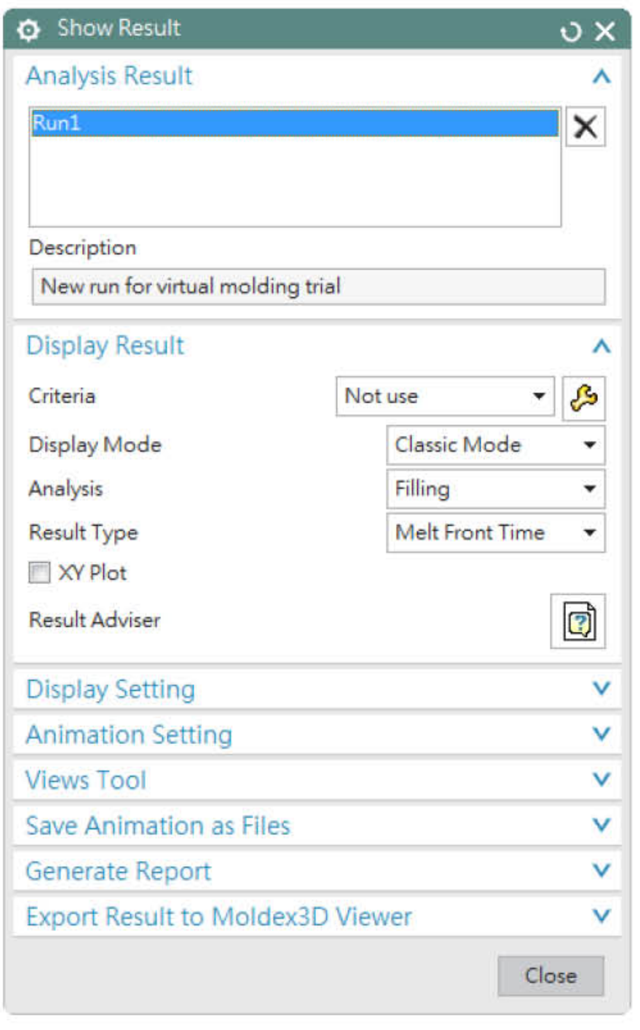
Analysis log shows the calculation data of the current run.

To control the calculation process, use the button on the bottom of the monitor:
Use the **Remove** to remove a selected analysis which is pending.
Use **Run** to start the pending run.
Use **Stop** to abort the calculation.
Use **Close** to Abort the calculation and close the monitor.

5.2.7 Show Result



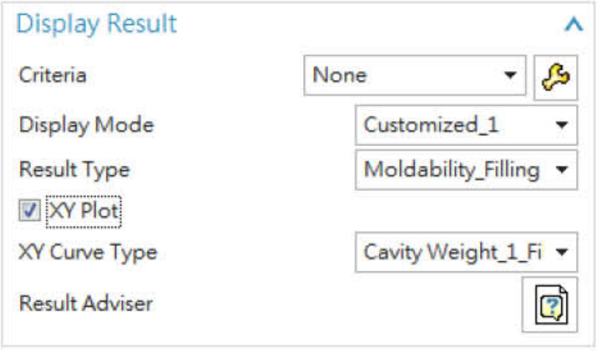
After analysis finished, click **Show Result** in the ribbon to open the dialogue.



Analysis Result

Select a run to view by clicking on it, or click **Delete** on the right side of the list to delete.

There are eight sections in the Show Result Dialog: Analysis Result, Display Result, Display Setting, Animation Setting, Views Tool, Save Animation as Files, Generate Report and Export Result to Moldex3D Viewer.

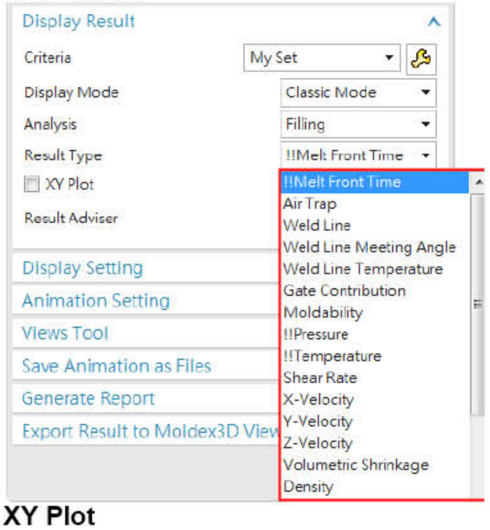


Criteria
Check if the results are out of user defined limit range.

Display Mode
User can change it to customized mode for only showing the result added to preference setting, or show all results by **Classic mode**.

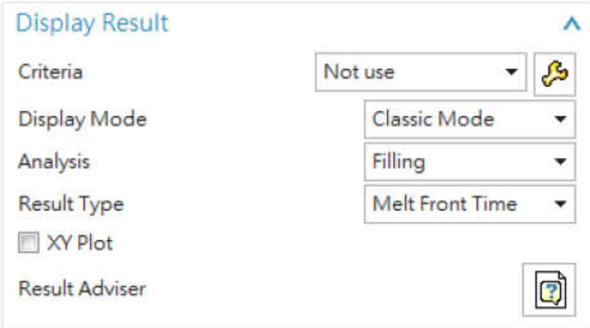
Analysis
By using the Filling/Packing option, users can view the result of filling or packing stage.

Result Type
Select the result to view from the pull down menu. There around 30 result types, including Melt Front Time, Air Trap, and Weld Line…ect.
If the result is out of the criterion used, there is a warning sign "!!" shown before result name.

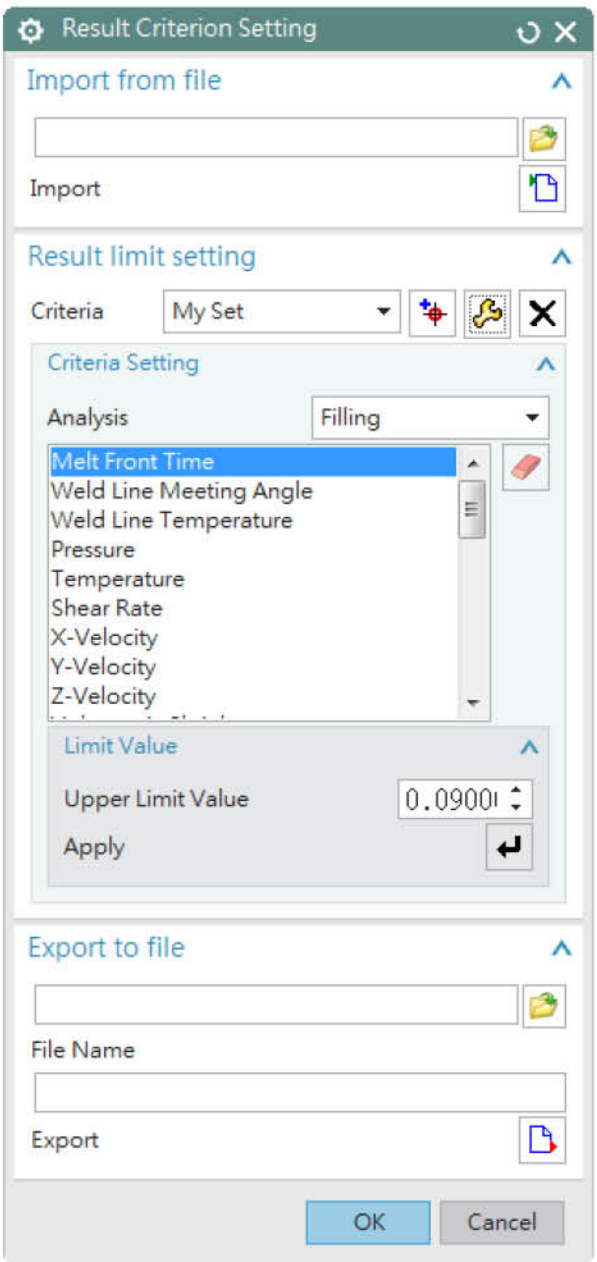


XY Plot
Check to activate XY Plot and select the result in the pull down menu of XY Curve Type.

5.2.8 Display Result - Criteria



Criteria
User can specify each result with limit range.
Expand the pull down menu to select the criterion you want to use.
Click the wrench button to call result criterion setting dialog and modify each result limit.

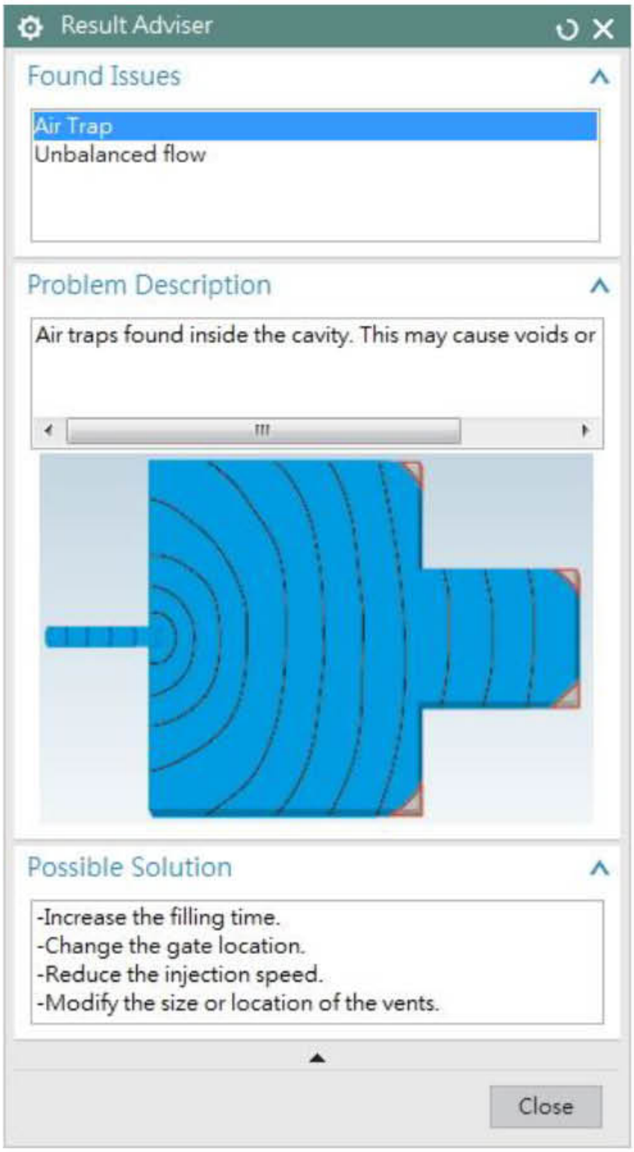


Result Criterion Setting
The Result Criterion Setting dialog box shows as the following:
Import from file—Import the criteria setting file *.xmrc.
Result limit setting—After clicking the result in list, the limit value group would be shown below. In limit value group, user can input upper/lower limit to be the constrain of the result.

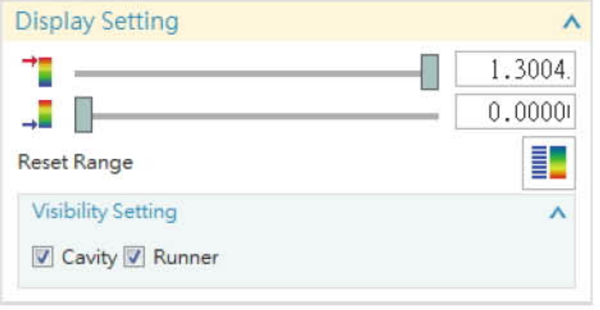
User also can remove setting by clicking the eraser button.
Export to file—Export the criteria setting file *.xmrc for other PC.

5.2.9 Display Result - Result Advisor

Result Advisor quickly coordinates possible issues for this run.



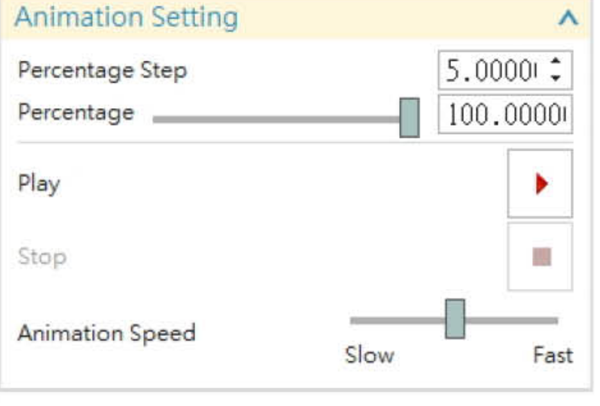
Result Advisor
Click **Result Advisor** to view the issues detected during the analysis, shown as the following:
Found Issues—Lists the issues detected in the simulation. Click an issue to view its description and solution.
Problem Description—Displays the description for the selected issue with a conceptual animation.
Possible Solution—Displays the recommended solutions for the selected issues.
Update the analysis as recommended in the solution and rerun the analysis.



Display Setting
This function allows users to adjust the upper and lower limits with a specific range for analysis results such as the part temperature and shear stress.

Upper limits of display range

Lower limits of display range
Use the slider to adjust the upper or lower limits of color legend.
Click **Reset Range** to reset the upper and lower limits as your preference.
Click **Cavity** to hide/show part.
Click **Runner** to hide/show runner.

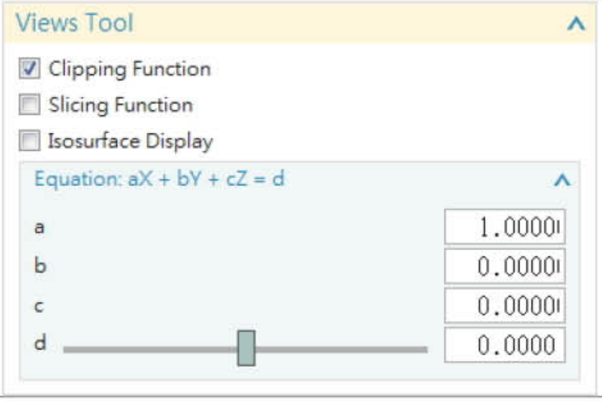


Animation Setting
Animation tools controls the play speed and display step. It helps to inspect results during the filling process or at specific filling percentage both on post and XY Curve.
Animation tools support only some results and XY Plot:

Percentage Step
Click the arrow to increase or reduce the step percentage of each frame. The animation displays smoothly with low value of percentage step.

Percentage
Pull the control bar changes the melt front display during filling process. The post and the color legend will be synchronized when pulling control bar.

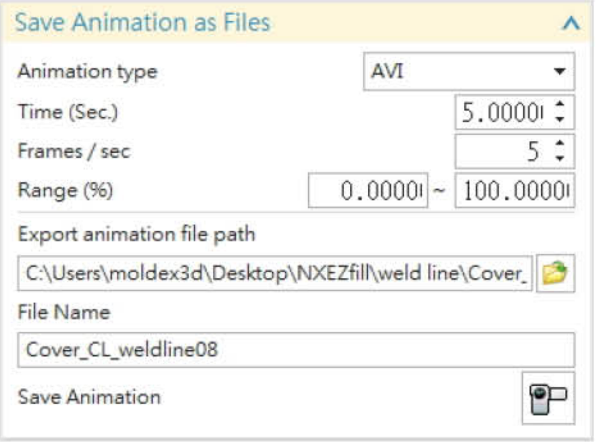
View Tool
Click a result item and check View Tools to enable view tools function.
There are three options, Clipping Function, Slicing Function, and Iso-surface Display.
Clipping Function
Referring to the plane equation: aX + bY + cZ = d, user can specify each parameter to define the clipping plane.
Slicing Function



Referring to the plane equation: aX + bY + cZ = d, user can specify each parameter to define the slicing plane.

Iso-surface Display

User can use this feature to view a specific value of result by adjusting the color legend.



Animation type

Select the format of the animation, AVI or GIF.

Time

Specify the total length of the animation.

Frames / sec

Define frames per second. The larger number makes the smoother animation but longer processing time.

Range

Users can make a partial animation by adjusting the range.

Export animation file path

Enter file name and specify the file path.

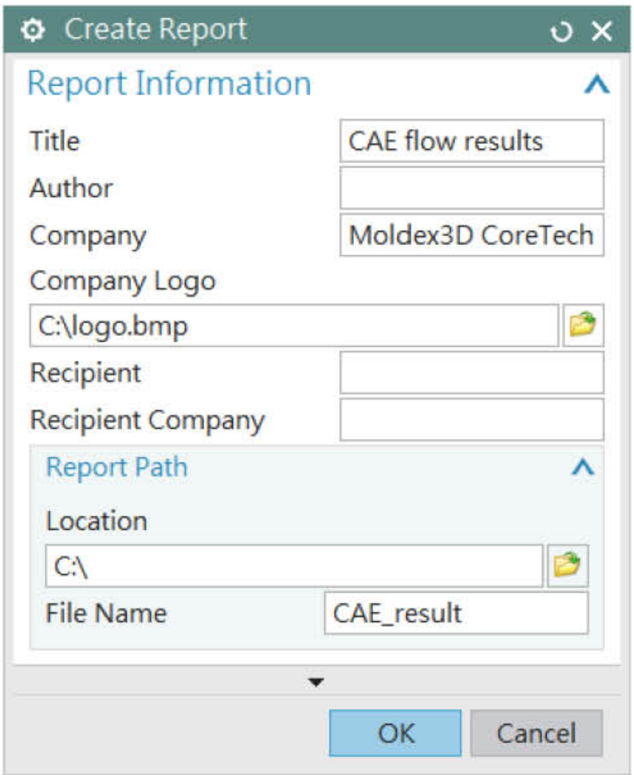
Save Animation

Click **Make Animation** to start recording.

5.2.10 Generate Report

After analyzing, analysis results such as melt front time animation, temperature distribution, pressure distribution can be coordinated as a Powerpoint file.

Click **Generate PowerPoint Report** .



Create Report – Report Information.
Fill the information to provide in the report, and click **OK**. The report will then be generated automatically.
The information items to be filled are:
Title
The title of the report.
Author
The author of the report.
Company
The company name that generates the report.
Company Logo
The company logo can be insert into the report, supporting the *.bmp file format.
Recipient
The recipient of the report.
Recipient Company
The recipient company of the report.
Report Path
Specify the report path
Location
The location of the report.
File Name
The filename of the report..

The report will be done automatically in several minutes.

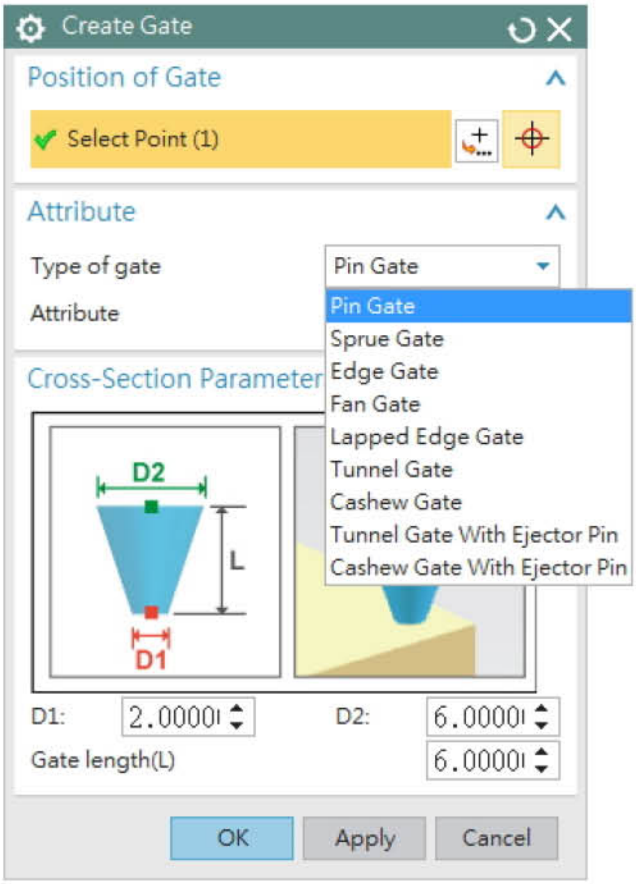
5.2.11 Export Result to Moldx3D Viewer

After analyzing, user can export *.rsv file which can be opened by Modex3D Viewer.

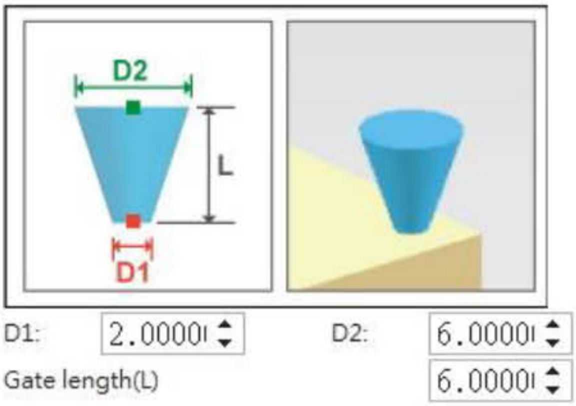
Click **Export RSV** button to output *.rsv file to indicated folder.

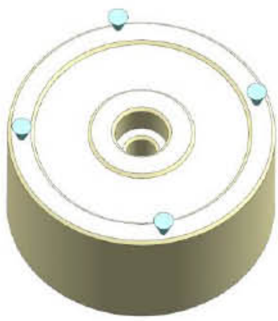
5.2.12 Gate Wizard

Click on **Gate Wizard** in the ribbon. A dialog will pop up for users to set the gate types and other corresponding parameters.



NX EasyFill Advanced supports many gate types: Pin Gate, Sprue Gate, Edge Gate, Fan Gate, Lapped Edge Gate, Tunnel Gate, Cashew Gate, Tunnel Gate With Ejector Pin, and Cashew Gate With Ejector Pin.
Each gate can be set to Cold Runner Gate or Hot Runner Gate. Following are the step-by-step procedures for setting each gate:



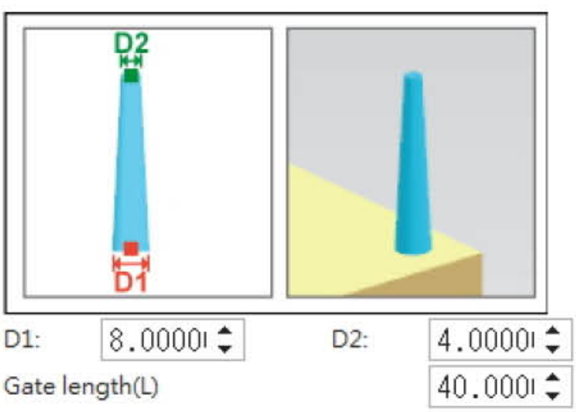


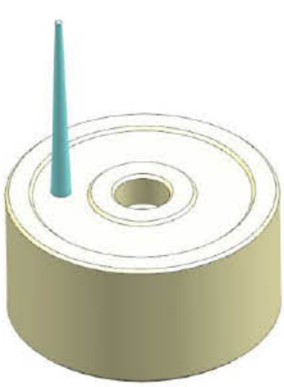
Pin Gate

Step 1: Select **Pin Gate**.

Step 2: Click Select Point  and select the point on model surface as the gating location.

Step 3: A gate parameter setting panel will pop up as left. Set related parameters, such as diameter of cross-section and gate length. After setting, click **OK** to confirm and exit the setting or click **Apply** to set next gate.





Sprue Gate

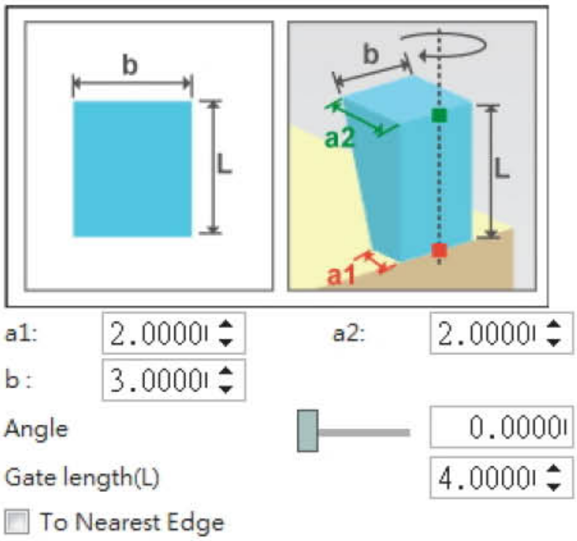
Step 1: Select Sprue Gate.

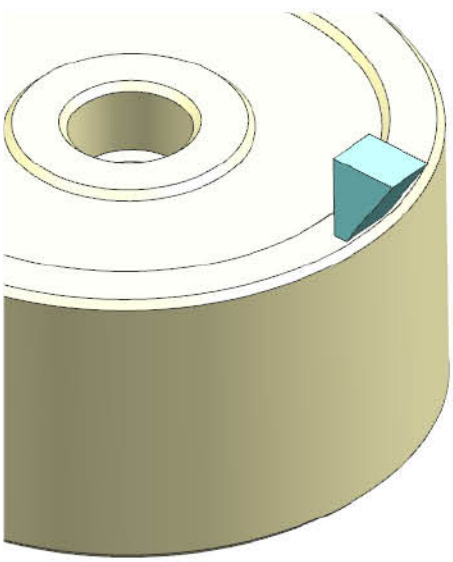
Step 2: Click Select Point  and select the face or point on model surface as the gating location.

Step 3: A gate parameter setting panel will pop up as shown. Set related parameters, such as diameter of cross-section and gate length. After setting, click **OK** to confirm and exit the setting or click **Apply** to set next gate.

Edge Gate

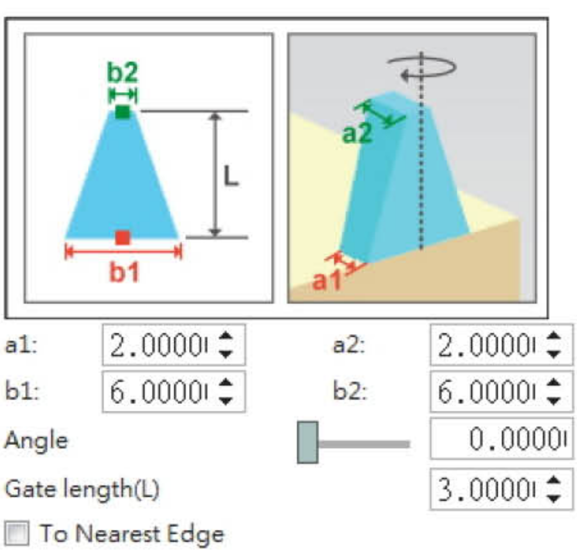
Step 1: Select Edge Gate.





Step 2: Click Select Point  and select the face or point on model surface as the gating location.

Step 3: A gate parameter setting panel will pop up as shown. Set related parameters, such as the width and length of cross-section, orientation vector and gate length. After setting, click **OK** to confirm and exit the setting or click **Apply** to set next gate.

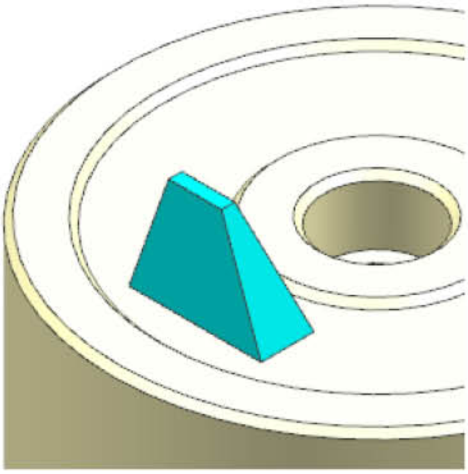


Fan Gate

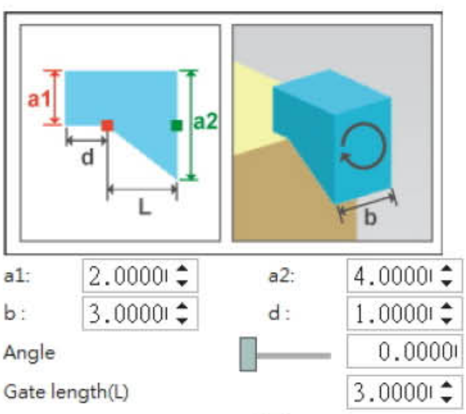
Step 1: Select Fan Gate.

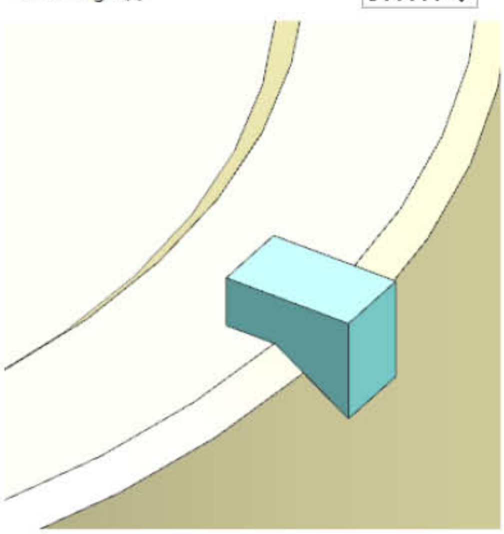
Step 2: Click Select Point  and select the face or point on model surface as the gating location.

Step 3: A gate parameter setting panel will pop up as shown. Set corresponding parameters, such as the width and length of cross-section, orientation vector and gate length. After setting, click **OK** to confirm and exit the setting or click **Apply** to set next gate.



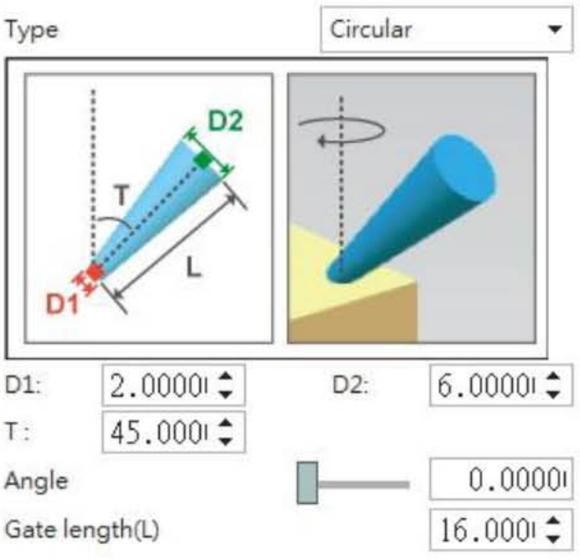
Note: While selecting Fan Gate, there is an additional option, **To Nearest Edge**, shown at the bottom of setting panel.
Check **To Nearest Edge**, and the gate will automatically move to the nearest edge from the location you selected.





Lapped Edge Gate
Step 1: Select Lapped Edge Gate.

Step 2: Click Select Point and select the face or point on model surface as the gating location.
Step 3: A gate parameter setting panel will pop up as shown. Set related parameters, such as the width and length of cross-section, orientation vector and gate length. After setting, click **OK** to confirm and exit the setting or click **Apply** to set next gate.

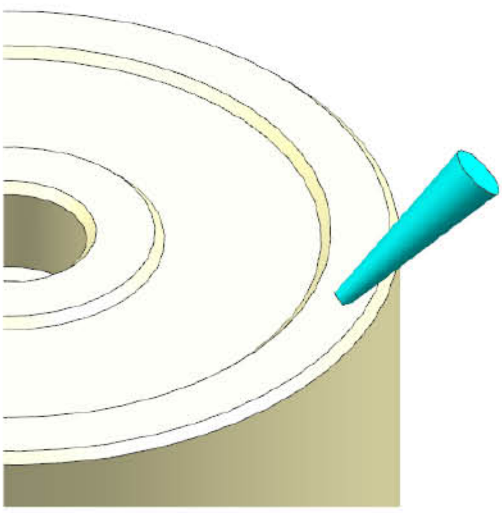


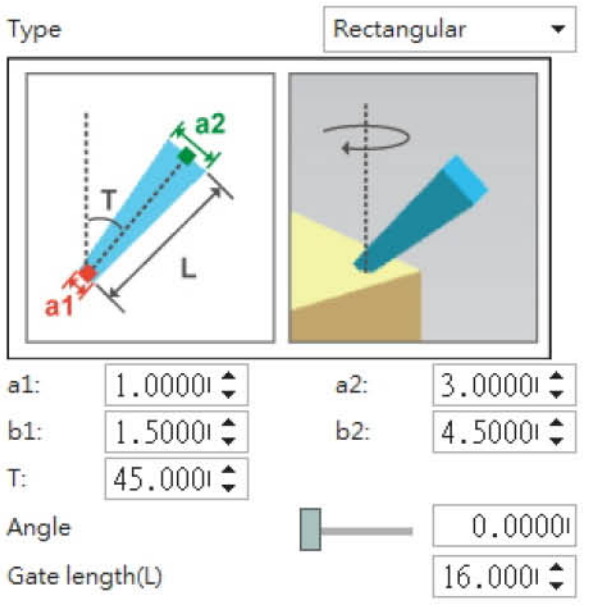
Tunnel Gate

Step 1: Select Tunnel Gate.

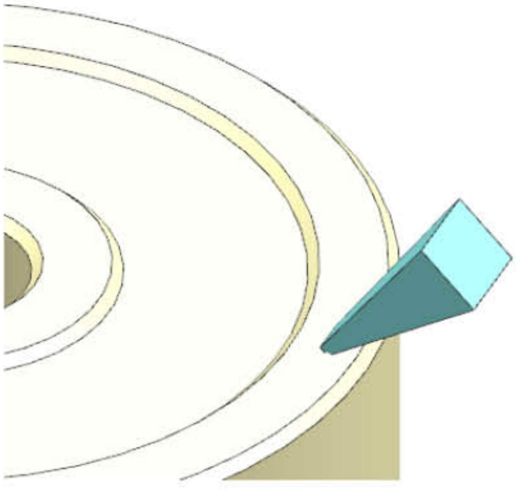
Step 2: Click Select Point  and select the face or point on model surface as the gating location.

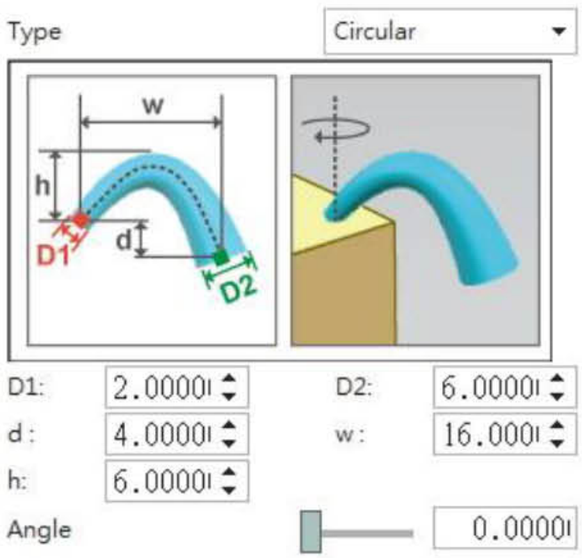
Step 3: A gate parameter setting panel will pop up as shown. Set related parameters, such as diameter of cross-section, orientation vector and gate length. After setting, click **OK** to confirm and exit the setting or click **Apply** to set next gate.

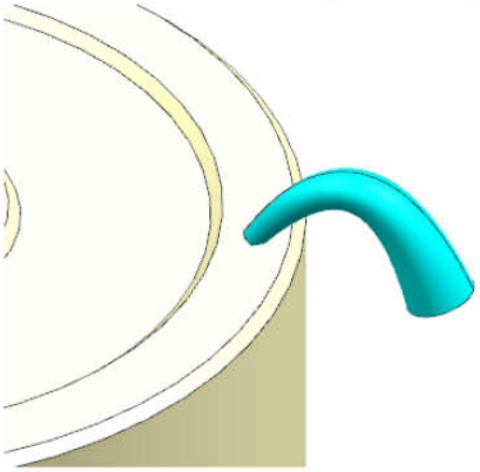




Note: The cross-section can be designed as circular or rectangular by selecting from **Type** drop down menu in the setting panel. Set corresponding parameters, such as dimensions of cross-section, orientation vector and gate length as in circular type.





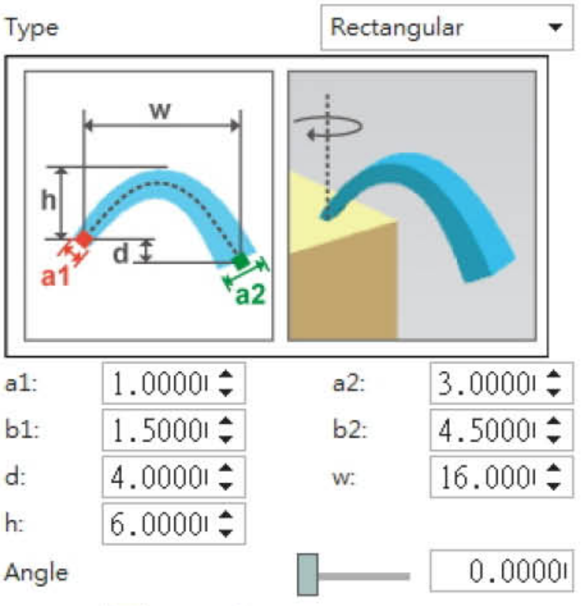


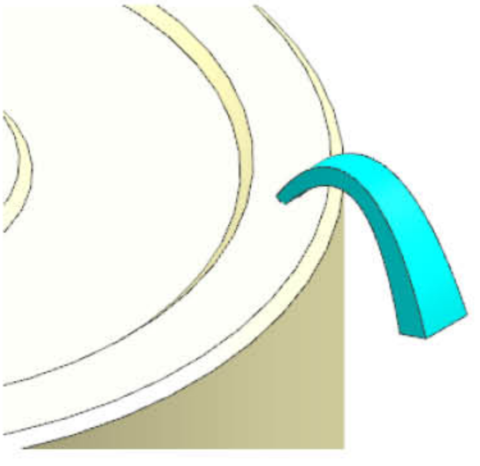
Cashew Gate

Step 1: Select Cashew Gate.

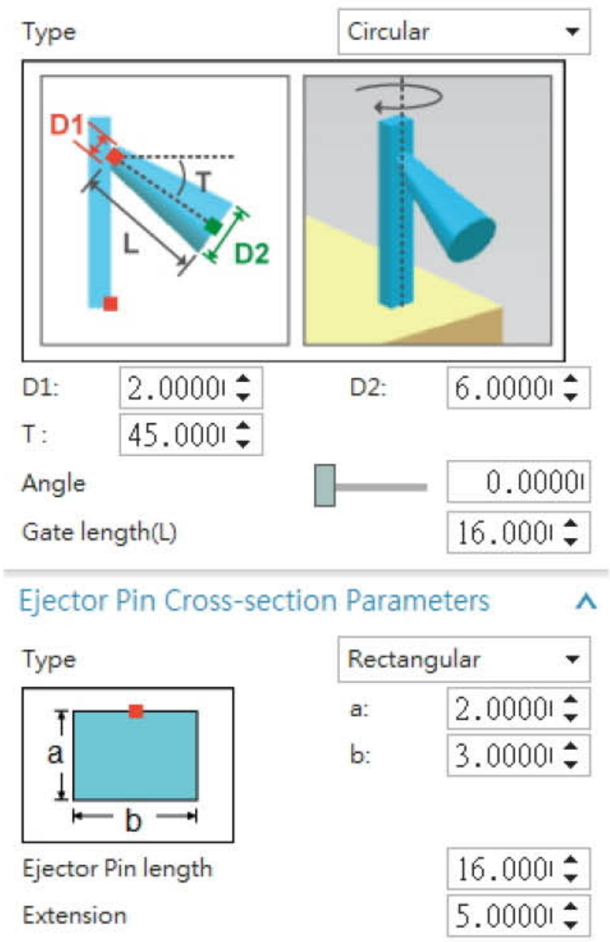
Step 2: Click Select Point  and select the face or point on model surface as the gating location.

Step 3: A gate parameter setting panel will pop up as shown. Set related parameters, such as the diameter of cross-section, orientation vector and gate length. After setting, click **OK** to confirm and exit the setting or click **Apply** to set next gate.





Note: The cross-section can be designed as circular or rectangular by selecting from **Type** drop down menu in the setting panel. Set corresponding parameters, such as dimensions of cross-section, orientation vector and gate length as in circular type.

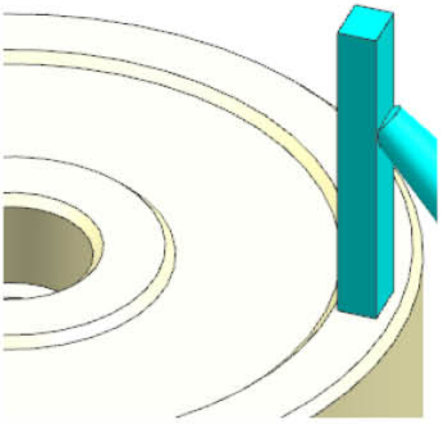


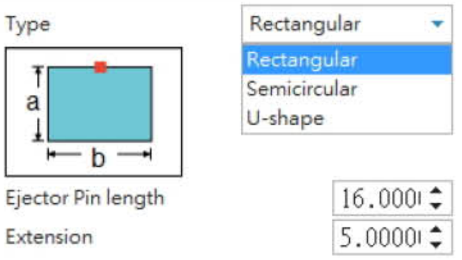
Tunnel Gate With Ejector Pin

Step 1: Select Tunnel Gate With Ejector Pin.

Step 2: Click Select Point  and select the face or point on model surface as the gating location.

Step 3: A gate parameter setting panel will pop up as shown. There will be two parts of setting parameters. The top part is for Tunnel Gate setting; the bottom is for Ejector Pin setting. After setting corresponding parameters, click **OK** to confirm and exit the setting or click **Apply** to set another gate.





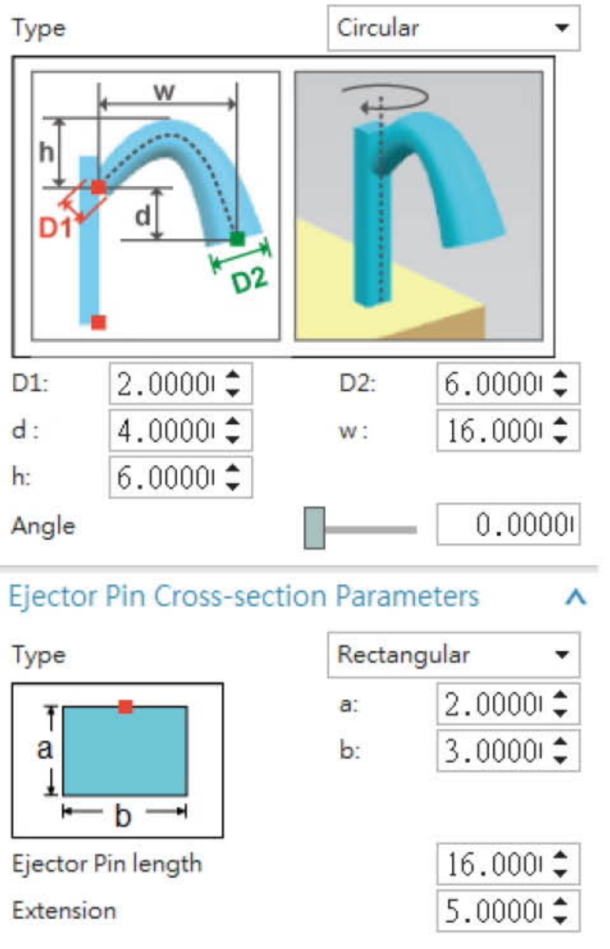
Note: There are three types of ejector for users to select: pin, Rectangular, Semicircular and U-shaped,

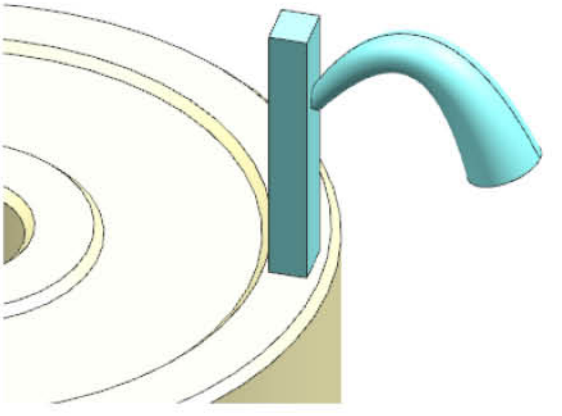
Cashew Gate with Ejector Pin

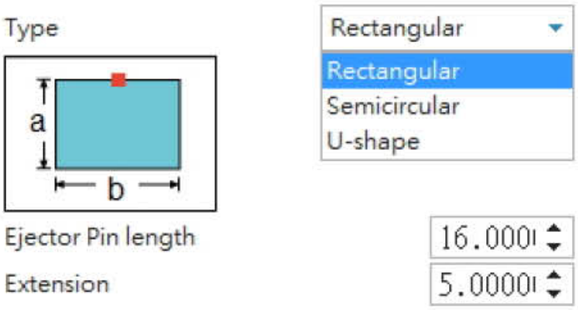
Step 1: Select Cashew Gate With Ejector Pin.

Step 2: Click Select Point  and select the face or point on model surface as the gating location.

Step 3: A gate parameter setting panel will pop up as shown. There will be two parts of setting parameters. The top part is for Cashew Gate Setting; the bottom is for Ejector Pin. After setting related parameters, click **OK** to confirm and exit the setting or click **Apply** to set next gate.



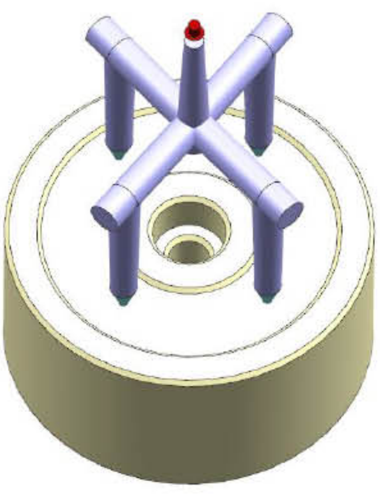




Note: There are three types of ejector pin for users to select: Rectangular, Semicircular and U-shaped,.

5.2.13 Runner Wizard

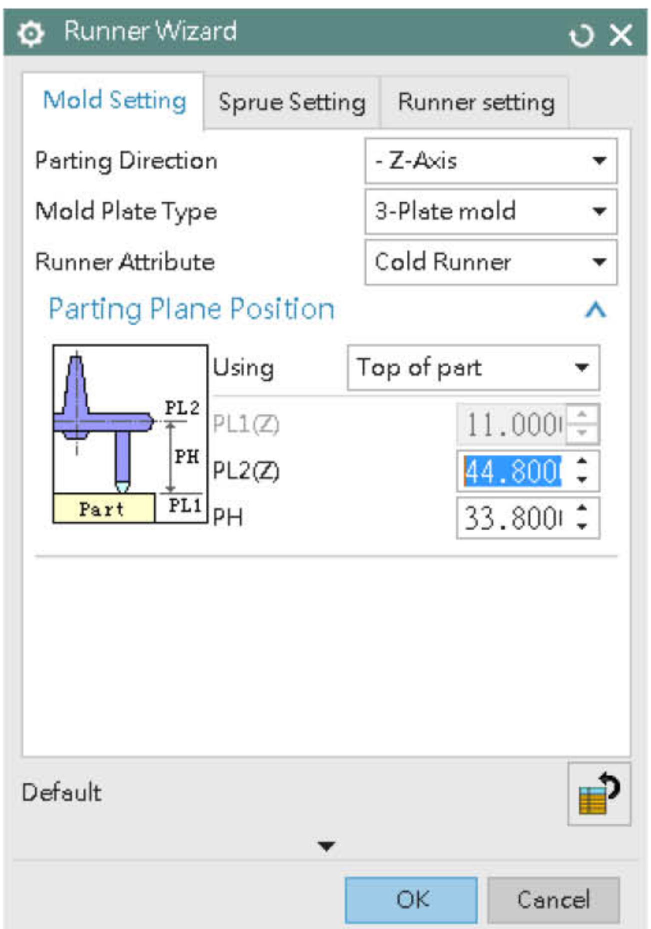
Runner Wizard automates the runner system design process by generating a knowledge built-in runner system. When designing the runner system, users have to set only gate location and dimension on the model, and the wizard will calculate for proper runner sizes, shapes and layout. It also supports to generate a complete runner system by specifying runners, sprue, and drops sequentially. With Runner Wizard, users can design runner systems more efficiently.



Click on **Runner Wizard** in the ribbon to launch Runner Wizard.

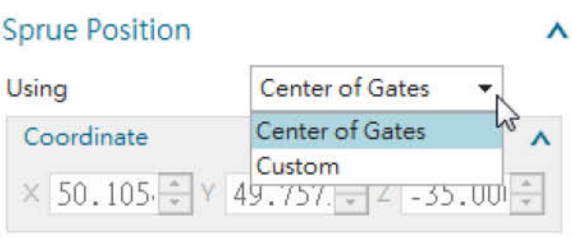
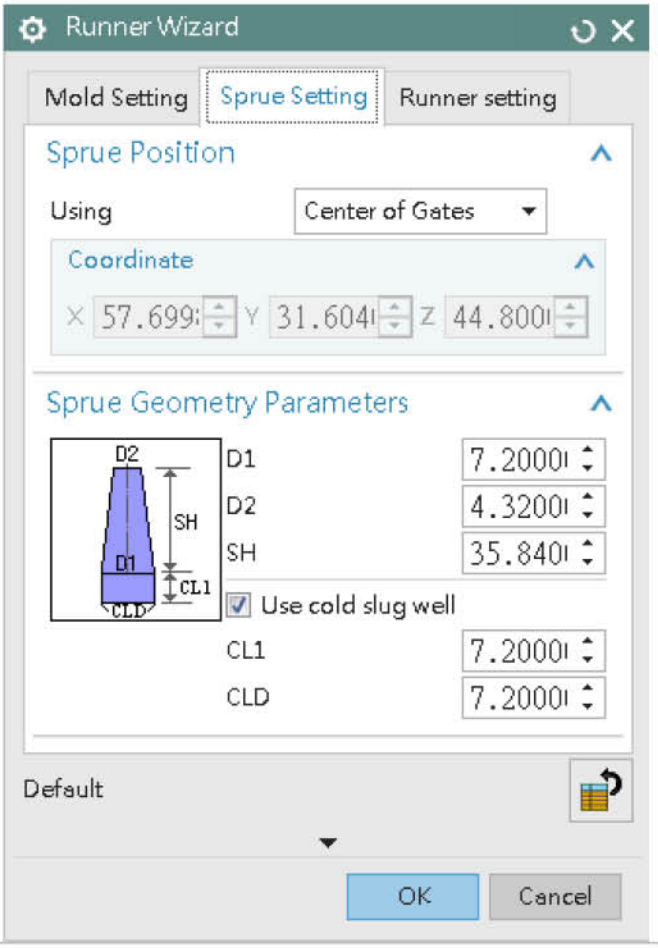
Note: it requires defining gates before running the wizard, or the following warning would be shown.

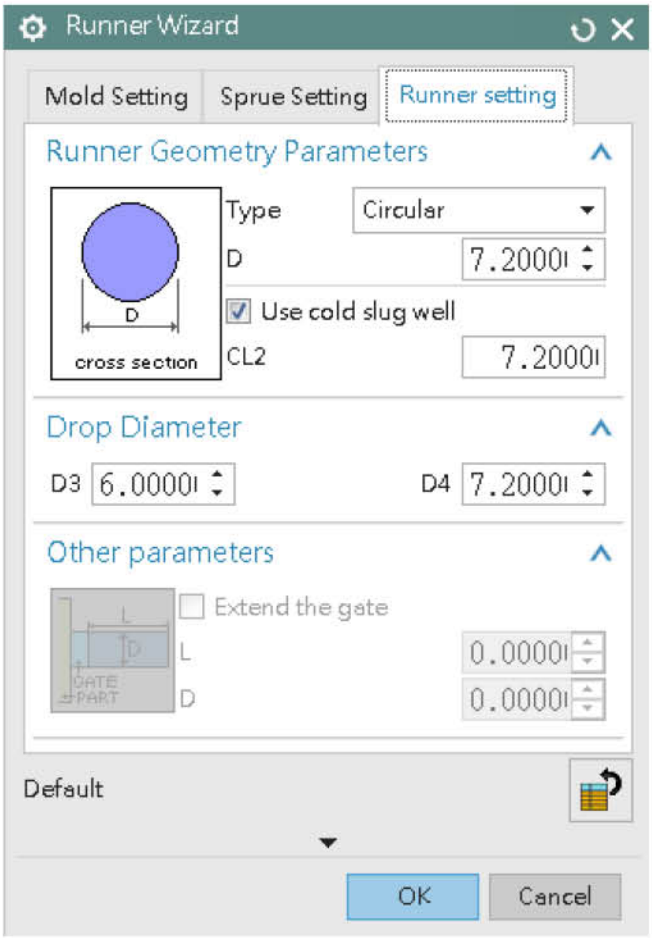
Click the Mold Setting tab. The explanation of each parameter is described below:



Parting direction:
Parting direction is available in positive and negative X, Y, and Z directions.
Mold Plate Type:
Select the mold plate type; there are 2-Plate mold and 3-Plate mold available.
Using:
The wizard offers four options to specify the position of main parting plane: Top of part, Bottom of part, Gate plane and Custom.
PL1:
Define the main parting plane position. (Modify only in Custom using type)
PL2:
Define the secondary parting plane position
PH:
Distance between main parting plane and secondary parting plane

The Sprue Settings tab of the wizard is for specifying the sprue position, sprue length and sprue diameter. The explanation of each parameter is described below:

	Select the sprue position from the drop-down list. There are two options available: Center of gates and Custom. Center of gates: Position the sprue at the center of gates. Custom: Specify a custom sprue location. Please enter values for sprue location in the X, Y and Z text box, or left-click in the graphic window to specify desired sprue location.
	**Sprue Geometry Parameters** **D1:** Specify the diameter of sprue start. **D2:** Specify the diameter of sprue end. **SH:** Specify the length of sprue. **CL1:** Specify the length of cold slug well. **CLD:** Specify the diameter of cold slug well. **Use cold slug well:** If users want to use the cold slug well for sprue, make sure this option is checked



Runner Setting

Type:
Define the cross section type of runners. There are Circular, Trapezoidal, Semicircular, and U-shaped available.

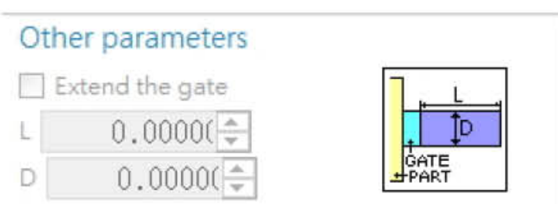
D:
Specify the diameter of runner.

Use cold slug well:
If having the cold slug well for runner, make sure this option is selected.

CL2:
Specify the length of cold slug well.

D3:
Specify the diameter of drop start.

D4:
Specify the diameter of drop end.



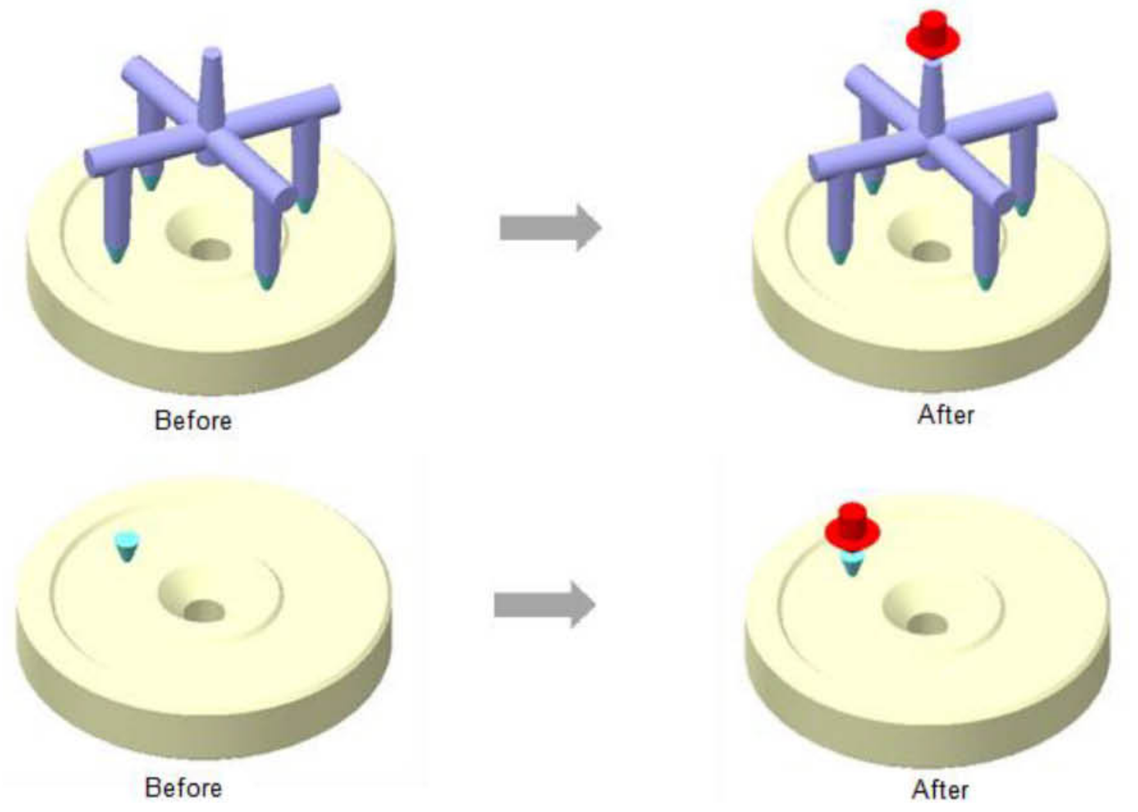
Extended Gate Parameters
In some specific locations, the runner setting has option to use extended gate, and specify the values of L (length) and D (Diameter) of the extended gate. Users can use extended gate function to avoid unreasonable runner creation.
An example is shown below:

5.2.14 Melt Entrance Wizard

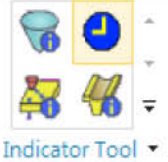
You can start auto melt entrance command in the ribbon.

Click **Melt Wizard** to start Melt Entrance Wizard.

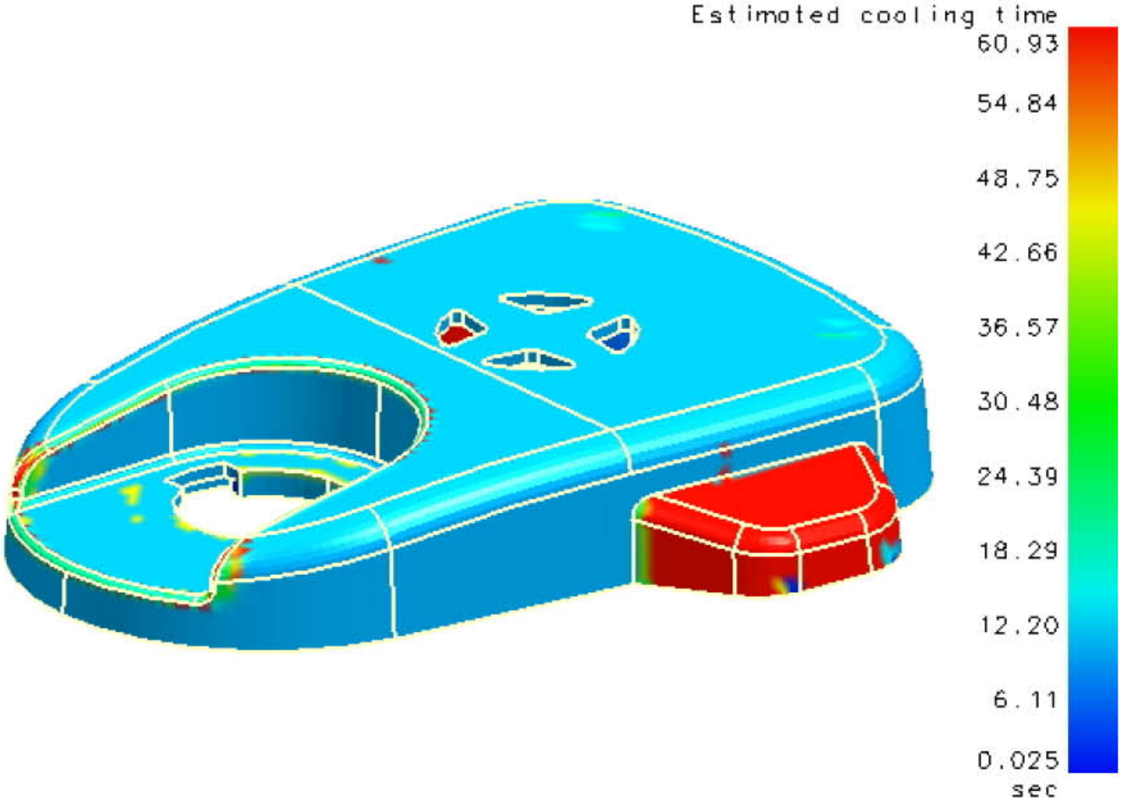
After clicking it, melt entrances will be added to the runner system automatically and the melt entrance can also be set directly on the gate as below.

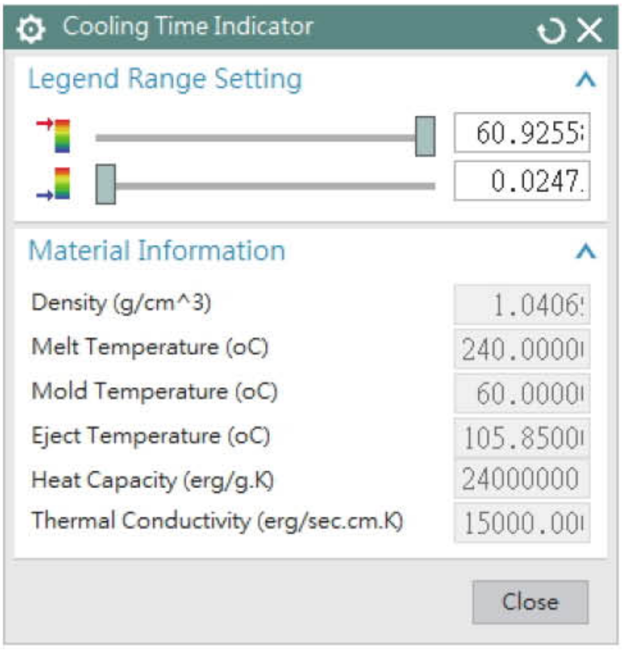


5.2.15 Cooling Time Indicator



This tool can estimate the cooling time of cavity.
After the part attribute setting process, you can find the **Cooling Time Indicator** icon enabled in the Tools menu. Within one click, the result of estimated cooling time will show up in the display window.

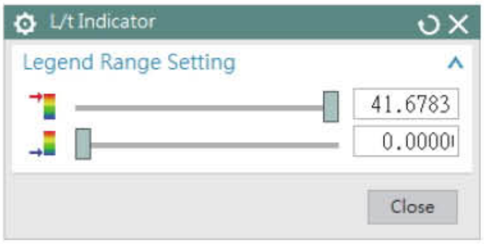


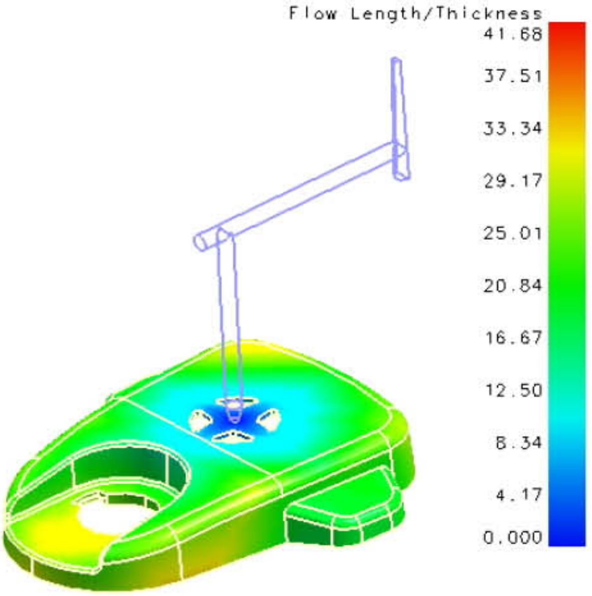


At the same time, Material Information and Parameter Setting will be displayed on left. The calculation of estimated cooling time is based on the associated information. Besides, users can use the slider to adjust upper and lower limits of color legend manually.

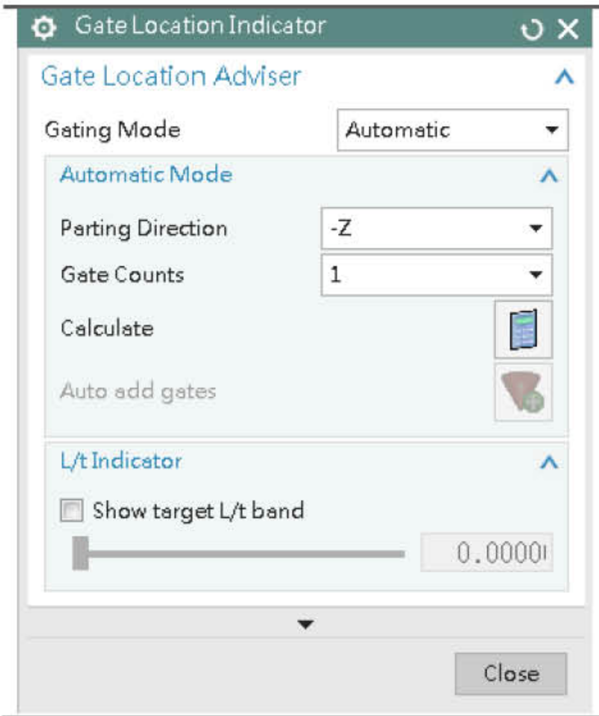
5.2.16 Flow Length to Thickness Ratio Indicator

After gate or melt entrance setting is done, click **L/t Indicator** in the Indicator Tool.
Pull the Upper/Lower Limit Slider to specify the range of the Flow Length/Thickness.

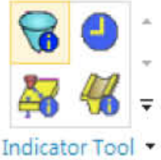




5.2.17 Gate Location Adviser



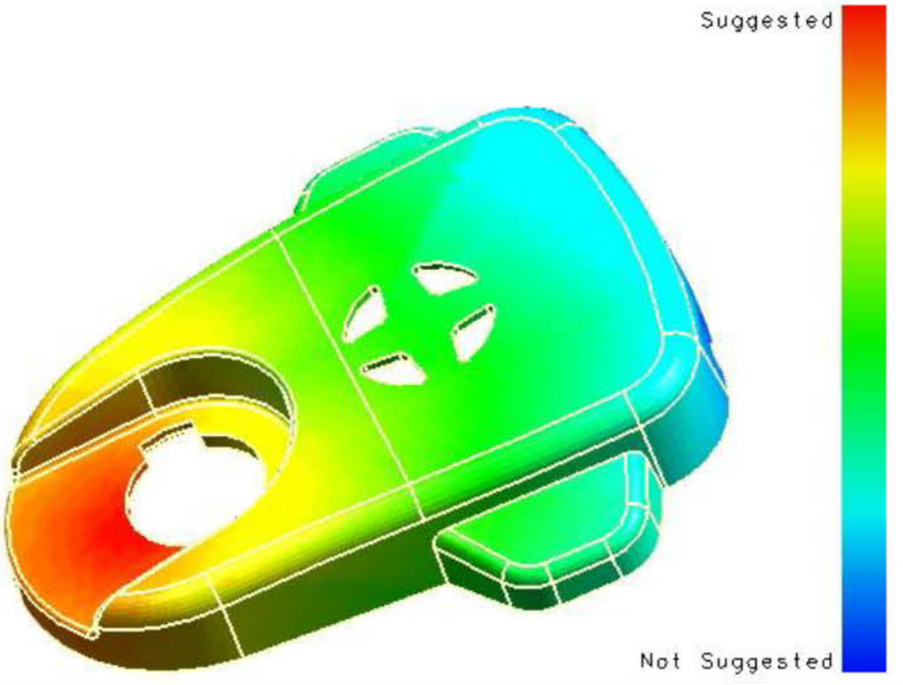
After part setting is done, click **Gate Location Advisor** in the category of indicator tool.

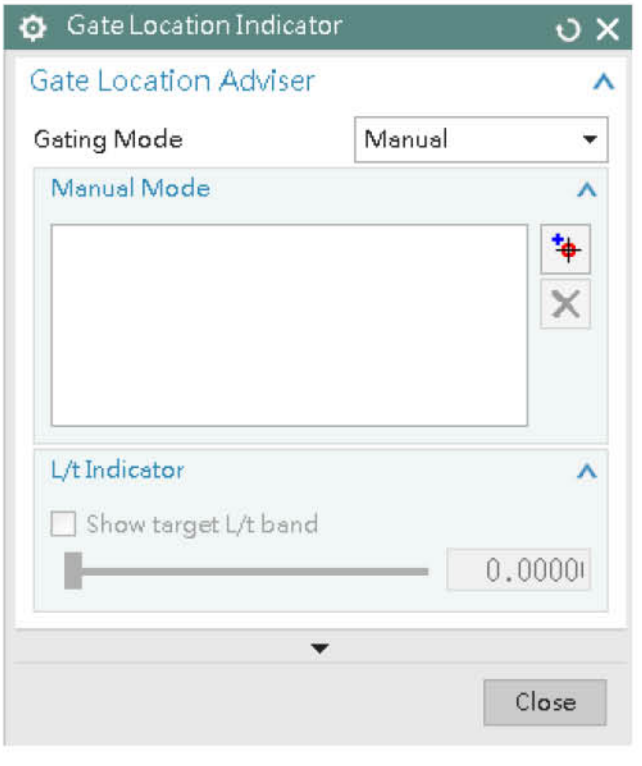


Automatic Mode
Select gate number (up to 6 gates).
Set parting direction of the mold base.

Click  to calculate the suggested gate location(s).

After calculation, click  to add pin gates on the suggested gate location(s).
Note: User can click **Show target L/t band** to show L/t immediately.

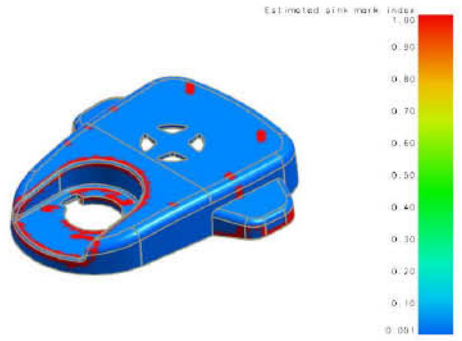




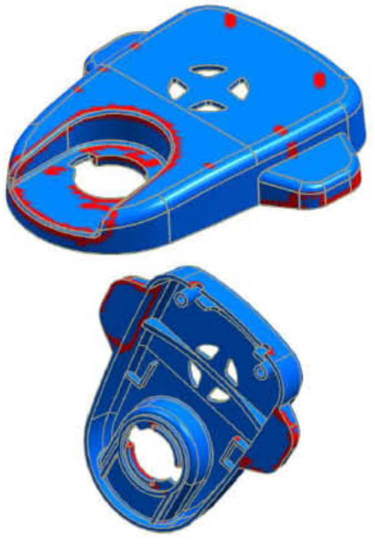
Manual Mode
Click **Add** button to call gate wizard for adding gates.
Note: User can click **Show target L/t band** to show L/t immediately.

5.2.18 Sink Mark Indicator

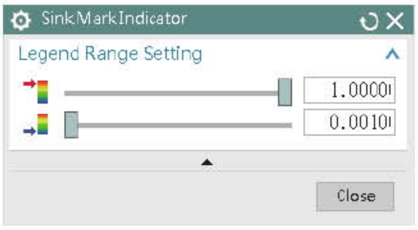
This tool is to estimate the location of sink mark.
To use this function, users must finish part attribute setting and make sure that both runner and melt entrance attribute exists (at least one attribute each).



After the attribute setting, click on **sink mark Indicator**. Within one click, the result of Sink Mark will show up on the display window.



Sink Mark usually occurs at rib or corners of cavity.



Legend Range Setting will be displayed on left. Users can use the slider to adjust upper and lower limits of color legend manually.

5.2.19 Upload to Teamcenter/Download from Teamcenter

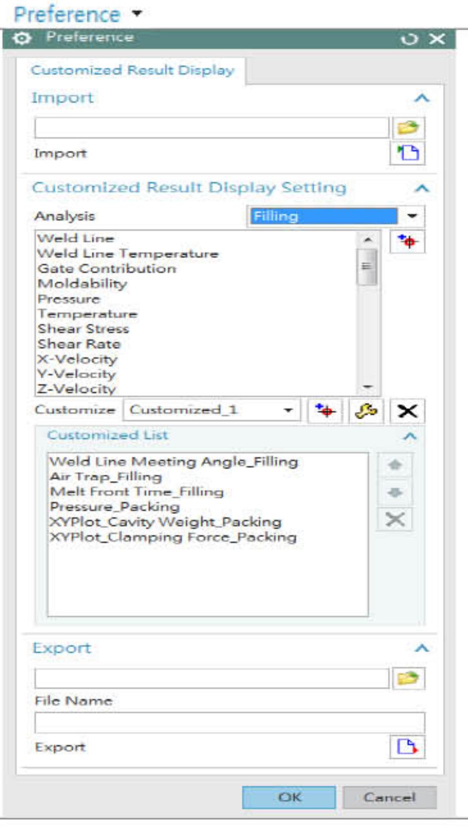
If NX is opened via Teamcenter, it is available to upload the analysis result to Teamcenter or download the analysis result from Teamcenter.
On the other hand, the analysis result opened directly from local disk and can only be saved to the local disk.
Note: The following functions will respond only if NX is opened via Teamcenter.

All result files will be placed in the local folder %Temp%\<item> after the analysis finished.
Click **Upload to Teamcenter** to collect the files into a XEDZ file and upload into Teamcenter as a dataset.

Click **Download from Teamcenter** to download the XEDZ file from Teamcenter back to the local folder %Temp%\<item>.

5.2.20 Set Preference

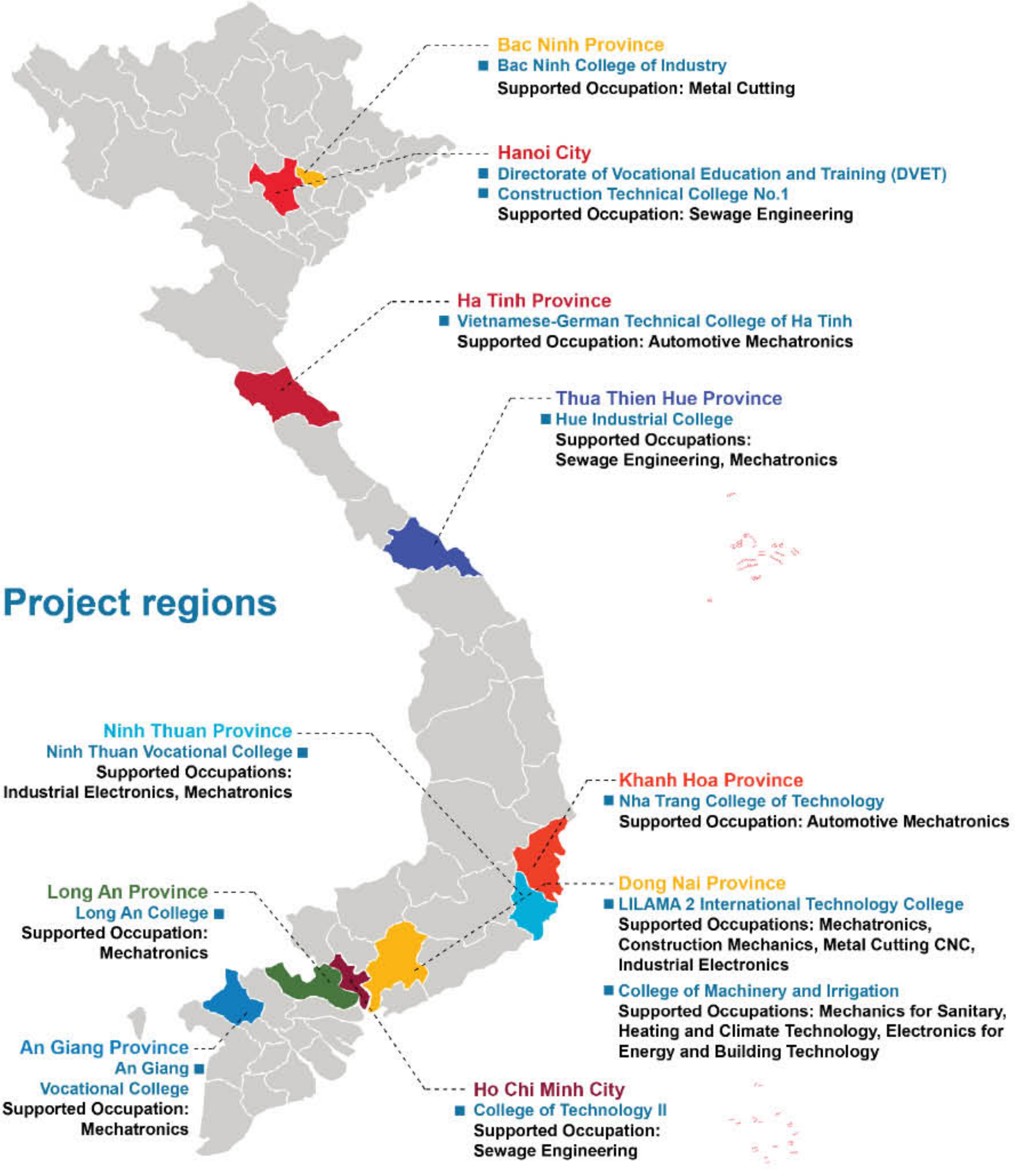
After click **Preference** in the ribbon, there will pop up a dialog to help filtering the result for users' need. It also provides that user can import *.xmrg file, or export setting file to other projects.



Customized Result Display
Users can set result list what they need to check ; after click OK button, result shown in **Show Result** dialog would be changed.
Import
User can import *.xmrg file in the group.
Customized Result Display Setting
User may change analysis type, select the result and click button, then the selected result would be added to customized list.
User also can set priority by button, and delete the result in customized list.
In addition, customization can modify by add button , rename button and delete button .
Export
After setting, user can export *.xmrg for other project.

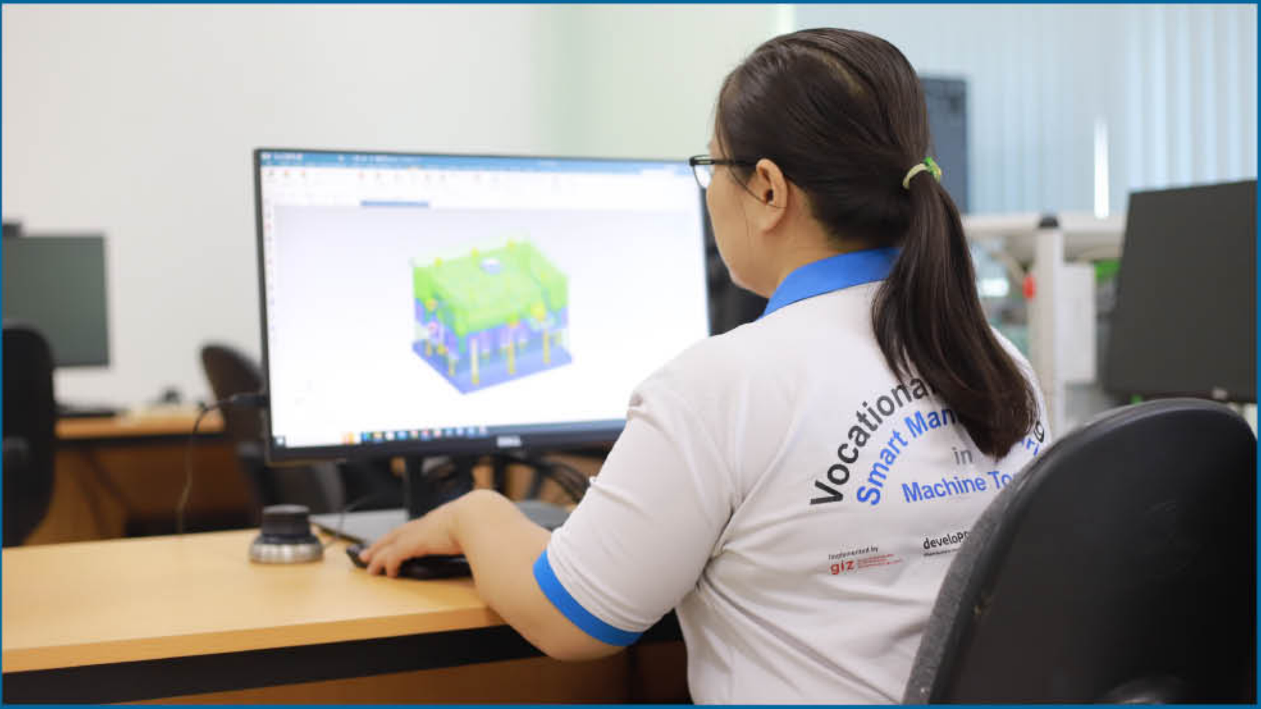
APPENDIX

1. PROJECT REGIONS AND PARTNERING TVET INSTITUTES



REFERENCES

1. Documents and drawings from mechanical faculty, Lilama2 college
2. Documents and pictures from websites on the internet
(https://docs.plm.automation.siemens.com/tdoc/nx/12.0.2/nx)
3. Inejction mould Design (R.G.W PYE)
4. Mould Design Using NX(Sham Tickoo)
5. NX Getting Start e Guide
6. NX Reference Guide
7. NX Help online
8. NX 12 for Engineering Design
Ming C. Leu, Wen Jin Tao, Amir Ghazanfari, Krishna Kolan
Department of mechanical and Aerospace Engineering
Missouri University of Science and Technology
9. Thiết kế khuôn NX Mould Winzard(4Ctech)
10. Thiết Kế Khuôn Với NX (Me-cad)



The training module was developed in the frame of the Development Partnership with the Private Sector (DPP) "Vocational Training for Smart Manufacturing in Machine Tools". The DPP is supported by the develoPPP.de programme of the German Federal Ministry for Economic Cooperation and Development (BMZ) and jointly implemented by the cooperation partners Siemens Vietnam, LILAMA 2 International Technology College and Deutsche Gesellschaft für Internationale Zusammenarbeit (GIZ) GmbH, represented by the Programme "Reform of Technical and Vocational Education and Training in Viet Nam".

Viet Nam, 12/2023