NX FOR ACADEMIC

NX for Engineering Design

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PREFACE

NX is one of the world's most advanced and tightly integrated CAD/CAM/CAE product development solution from Siemens Digital Industries Software. Spanning the entire range of product development, NX delivers immense value to enterprises of all sizes. It simplifies complex product development, thus speeding up the process of introducing products to the market.

The NX software integrates multidisciplinary principles, conceptual design, 3D modeling, documentation, engineering analysis, graphic simulation, and concurrent engineering. The software has powerful hybrid modeling capabilities by integrating constraint-based feature modeling and explicit geometric modeling. In addition to modeling standard geometry parts, it allows the user to design complex freeform shapes such as airfoils and manifolds. It also merges solid and surface modeling techniques into one powerful toolset.

This self-guided tutorial and hyperlinks (in underlined blue letters) to YouTube channel video provide a step-by-step approach for users to learn NX. It is intended for those with no previous experience with NX. However, users of previous versions of NX may also find this tutorial useful for them to learn the new user interfaces and functions. The user will be guided from starting a NX session to creating models and designs that have various applications. Each chapter has components explained with the help of various dialog boxes and screenshots. These components are later used in the assembly modeling, machining and finite element analysis. The files of components are also available online to download and use. To give NX user faster access to new enhancements and quality improvements, Siemens Digital Industries software begins delivering the NX software using a Continuous Release methodology.

Our previous efforts to prepare the NX self-guided tutorial were funded by the National Science Foundation's Advanced Technological Education Program and by the Partners of the Advancement of Collaborative Engineering Education (PACE) program.

If you have any questions or comments about this tutorial, please email Ming C. Leu at mleu@mst.edu or Chia-Hung Hung at hungch@mst.edu. The models and all the versions of the tutorial are available at http://web.mst.edu/~mleu.

CHAPTER 1 – INTRODUCTION

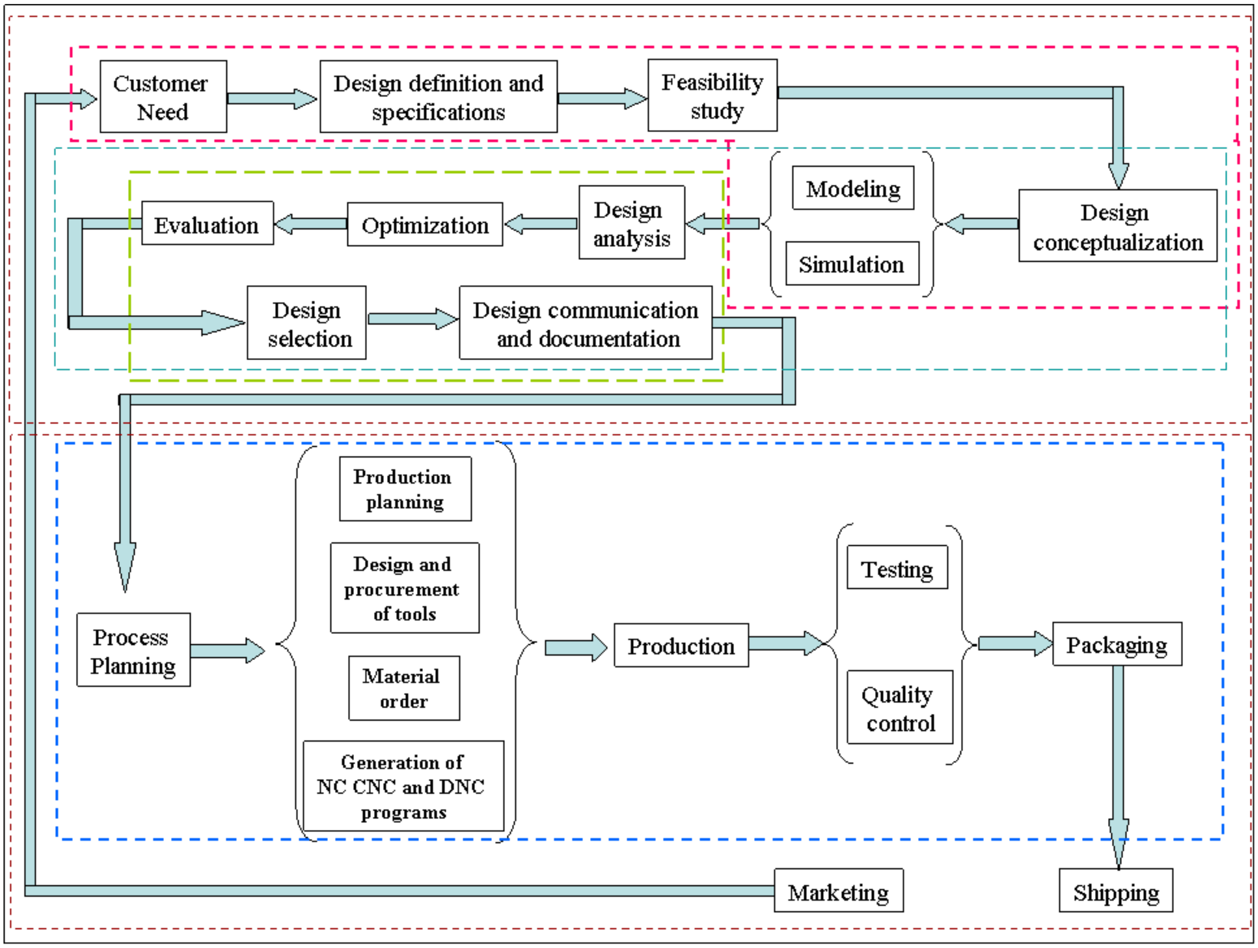
The modern manufacturing environment can be characterized by the paradigm of delivering products of increasing variety, smaller batches and higher quality in the context of increasing global competition. Industries cannot survive worldwide competition unless they introduce new products with better quality, at lower costs and with shorter lead-time. There is intense international competition and decreased availability of skilled labor. With dramatic changes in computing power and wider availability of software tools for design and production, engineers are now using *Computer Aided Design* (CAD), *Computer Aided Manufacturing* (CAM) and *Computer Aided Engineering* (CAE) systems to automate their design and production processes. These technologies are now used every day for all sorts of different engineering tasks. Below is a brief description of how CAD, CAM, and CAE technologies are being used during the product realization process.

1.1 PRODUCT REALIZATION PROCESS

The product realization process can be roughly divided into two phases: design and manufacturing. The design process starts with identification of new customer needs and design variables to be improved, which are identified by the marketing personnel after getting feedback from the customers. Once the relevant design information is gathered, design specifications are formulated. A feasibility study is conducted with relevant design information and detailed design and analyses are performed. The detailed design includes design conceptualization, prospective product drawings, sketches and geometric modeling. Analysis includes stress analysis, interference checking, kinematics analysis, mass property calculations and tolerance analysis, and design optimization. The quality of the results obtained from these activities is directly related to the quality of the analysis and the tools used for conducting the analysis.

The manufacturing process starts with the shop-floor activities beginning from production planning, which uses the design process drawings and ends with the actual product. Process planning includes activities like production planning, material procurement, and machine selection. There are varied tasks like procurement of new tools, NC programming and quality checks at various stages during the production process. Process planning includes planning for all the processes used in manufacturing of the product. Parts that pass the quality control inspections are assembled functionally tested, packaged, labeled, and shipped to customers.

A diagram representing the Product Realization Process (*Mastering CAD/CAM*, by Ibrahim Zeid, McGraw Hill, 2005) is shown below.



1.2 BRIEF HISTORY OF CAD/CAM DEVELOPMENT

The roots of current CAD/CAM technologies go back to the beginning of civilization when engineers in ancient Egypt recognized graphics communication. Orthographic projection practiced today was invented around the 1800s. The real development of CAD/CAM systems started in the 1950s. CAD/CAM went through four major phases of development in the last century. The 1950s was known as the era of interactive computer graphics. MIT's Servo Mechanisms Laboratory demonstrated the concept of numerical control (NC) on a three-axis milling machine. Development in this era was slowed down by the shortcomings of computers at the time. During the late 1950s the development of Automatically Programmed Tools (APT) began and General Motors explored the potential of interactive graphics.

The 1960s was the most critical research period for interactive computer graphics. Ivan Sutherland developed a sketchpad system, which demonstrated the possibility of creating drawings and altercations of objects interactively on a cathode ray tube (CRT). The term CAD started to appear with the word ‘design’ extending beyond basic drafting concepts. General Motors announced their DAC-1 system and Bell Technologies introduced the GRAPHIC 1 remote display system.

During the 1970s, the research efforts of the previous decade in computer graphics had begun to be fruitful, and potential of interactive computer graphics in improving productivity was realized by industry, government and academia. The 1970s is characterized as the golden era for computer drafting and the beginning of ad hoc instrumental design applications. National Computer Graphics Association (NCGA) was formed and Initial Graphics Exchange Specification (IGES) was initiated.

In the 1980s, new theories and algorithms evolved, and integration of various elements of design and manufacturing was developed. The major research and development focus were to expand CAD/CAM systems beyond three-dimensional geometric designs and provide more engineering applications.

The present-day CAD/CAM development focuses on efficient and fast integration and automation of various elements of design and manufacturing along with the development of new algorithms. There are many commercial CAD/CAM packages available for direct usages that are user-friendly and very proficient.

Below are some of the commercial packages in the present market.

- Solid Edge, AutoCAD, Inventor and TurboCAD are some affordable CAD software systems.
- NX, Creo, CATIA, SolidWorks, and Simcenter 3D are high-end modeling, designing software systems that are costlier but more powerful. These software systems also have computer aided manufacturing and engineering analysis capabilities.
- Onshape, NX (in their recent release), and Fusion 360 are cloud based CAD software, which provide CAD capabilities via user’s browser.

- ANSYS, ABAQUS, NASTRAN and COMSOL are packages mainly used for CAE purposes.

1.3 DEFINITION OF CAD/CAM/CAE

Following are the definitions of some of the terms used in this tutorial.

1.3.1 Computer Aided Design – CAD

CAD is technology concerned with using computer systems to assist in the creation, modification, analysis, and optimization of a design. Any computer program that embodies computer graphics and an application program facilitating engineering functions in design process can be classified as CAD software.

The most basic role of CAD is to define the geometry of design – a mechanical part or a contour surface part, a product assembly, an architectural structure, an electronic circuit, a building layout, etc. The greatest benefits of CAD systems are that they can save considerable time and reduce errors caused by otherwise having to redefine the geometry of the design from scratch every time it is needed.

1.3.2 Computer Aided Manufacturing – CAM

CAM technology involves computer systems that plan, manage, and control the manufacturing operations through computer interface with the plant's production resources.

One of the most important areas of CAM is numerical control (NC). This is the technique of using programmed instructions to control a machine tool, which cuts, mills, grinds, punches or turns raw stock into a finished part. Another significant CAM function is in the programming of robots. Process planning is also a target of computer automation.

1.3.3 Computer Aided Engineering – CAE

CAE technology uses a computer system to analyze the functions of a CAD-created product, allowing designers to simulate and study how the product will behave so that the design can be refined and optimized.

CAE tools are available for a number of different types of analyses. For example, kinematic analysis programs can be used to determine motion paths and linkage velocities in mechanisms. Dynamic analysis programs can be used to determine loads and displacements in complex assemblies such as automobiles. One of the most popular methods of analyses is using a Finite Element Method (FEM). This approach can be used to determine stress, deformation, heat transfer, magnetic field distribution, fluid flow, and other continuous field problems that are often too tough to solve with any other approach.

1.4. SCOPE OF THIS TUTORIAL

This tutorial is written for students and engineers who are interested in learning how to use NX for designing mechanical components and assemblies. Learning to use NX will also be valuable for learning how to use other CAD systems such as Creo and CATIA. This tutorial provides a systematic approach for learning NX.

Chapter 2 includes the NX essentials from starting a session to getting familiar with the NX layout by practicing basic functions such as Print, Save, and Exit. It also gives a brief description of the Coordinate System, Layers, various toolboxes and other important commands, which will be used in later chapters.

Chapter 3 presents the concept of sketching. It describes how to create sketches and to give geometric and dimensional constraints. This chapter is very important since present-day components are very complex in geometry and difficult to model with only basic features.

Chapter 4 begins actual designing and modeling of parts. It describes different features such as reference features, swept features and primitive features and how these features are used to create designs. Various kinds of feature operations are performed on the created features.

Chapter 5 teaches how to create a drawing from a part model. In this chapter, we show how to create a drawing by adding views, dimensioning the part drawings, and modifying various attributes in the drawing such as text size, arrow size and tolerance.

Chapter 6 teaches the concepts of Assembly Modeling and its terminologies. It describes Top-Down modeling and Bottom-Up modeling. We will use Bottom-Up modeling to assemble components into a product.

Chapter 7 introduces free-form modeling. The method of modeling smooth curves and surfaces with tangent, curvature or higher degree continuity will be demonstrated.

Chapter 8 is capsulated into a brief introduction to Design Simulations A.K.A. Simcenter 3D that is available in NX for the Finite Element Analysis.

Chapter 9 provides a real-time experience of implementing a designed model into a manufacturing environment for machining. This chapter deals with generation, verification and simulation of Tool Path to create CNC (Computer Numerical Control) codes to produce the designed parts from multiple axes and even advanced CNC machines.

Chapter 10 is a brief introduction of special modeling features and tools regarding Convergent Modeling, Process Automation, and Sheet Metal. A note of continuous release of NX is also provided.

The examples and exercise problems for part models in each chapter are so designed that they will be finally assembled in later chapters. Due to this distinctive feature, you should save all the design models that you have created in each chapter.

Note that the sample data used in this book are neither non-proprietary to the Missouri University of Science and Technology nor to Siemens Digital Industries Software.

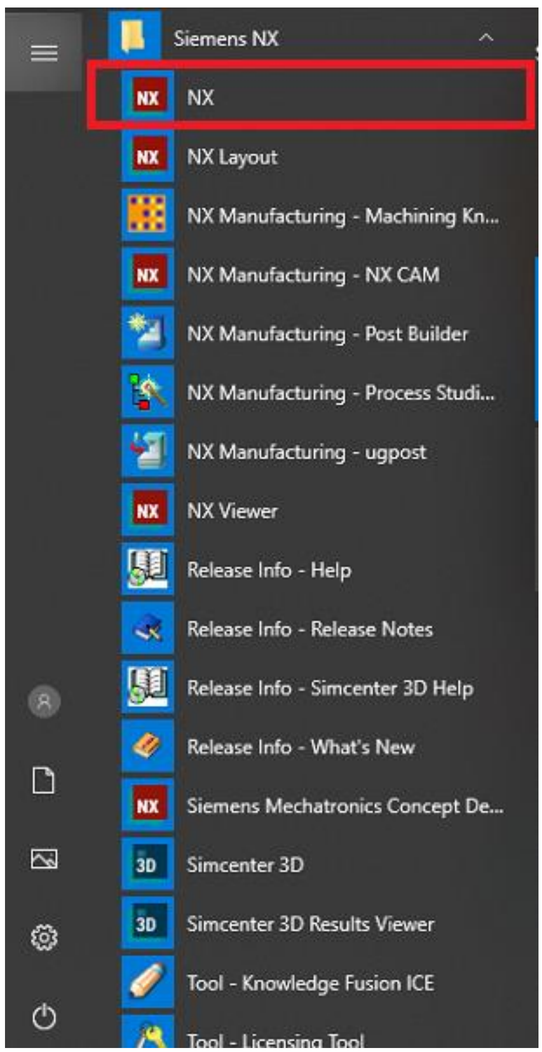
CHAPTER 2 – GETTING STARTED

Let us begin with starting of an NX session. This chapter will provide the basics required to use any CAD/CAM package. You will learn the preliminary steps to start, to understand and to use the NX package for modeling, drafting, etc. It contains five sub-sections a) Opening an NX session, b) Printing, saving, and closing part files, c) getting acquainted with the NX user interface d) Using layers and e) Understanding important commands and dialogs.

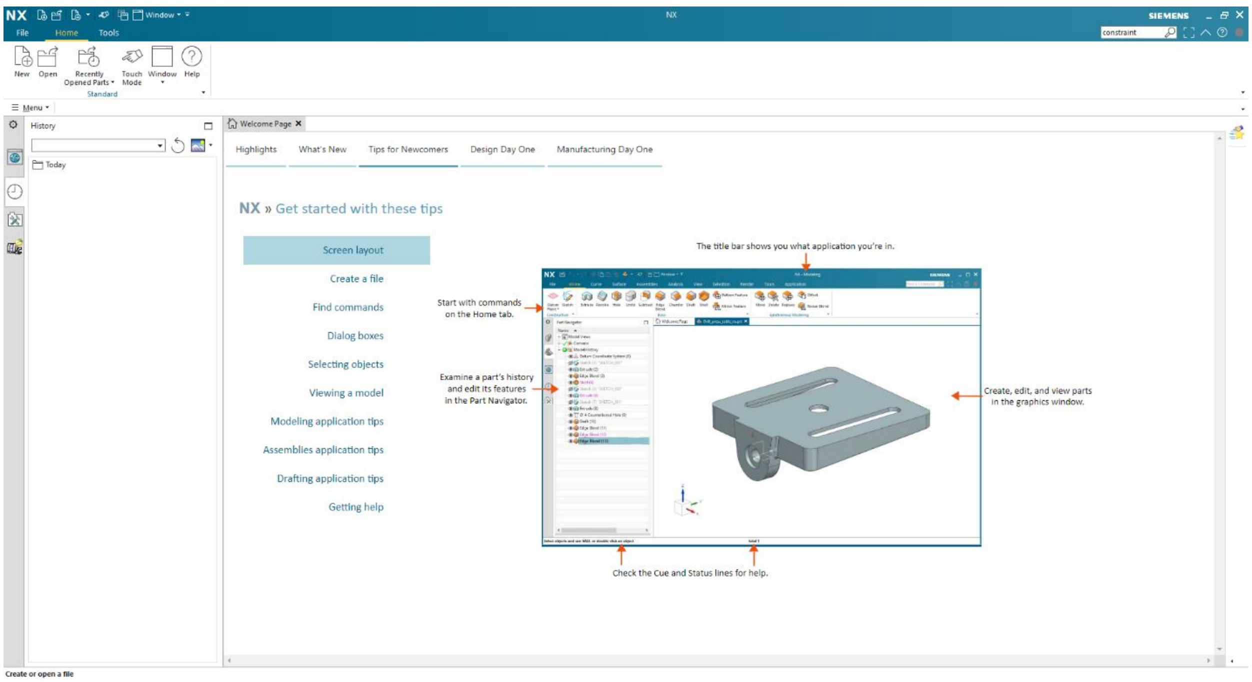
2.1 STARTING AN NX SESSION AND OPENING FILES

2.1.1 Start an NX Session

- Windows desktop screen, click on **Start** → **Program List** → **Siemens NX** → **NX**

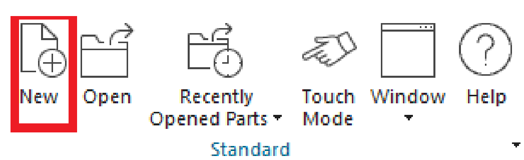


The main NX Screen will open. This is the Gateway for the NX software. The NX blank screen looks like the figure shown below. There will be several tips displayed on the screen about the special features of the current version. The *Gateway* also has the *Standard Toolbar* that will allow you to create a new file or open an existing file. On the left side of the *Gateway* screen, there is a toolbar called the *Resource Bar* that has menus related to different modules and the ability to define and change the *Role* of the software, view *History* of the software use and so on. This will be explained in detail later in this chapter.



2.1.2 Open a New File

Let's begin by learning how to [open a new part file in NX](). To create a new file there are three options.



- ➢ Click on the **New** button on left of Ribbon bar

OR

- ➢ Go through the **File** drop-down menu at the top-left of the screen and click **New**

OR

- ➢ Click **NEW** button at the top-left of the screen

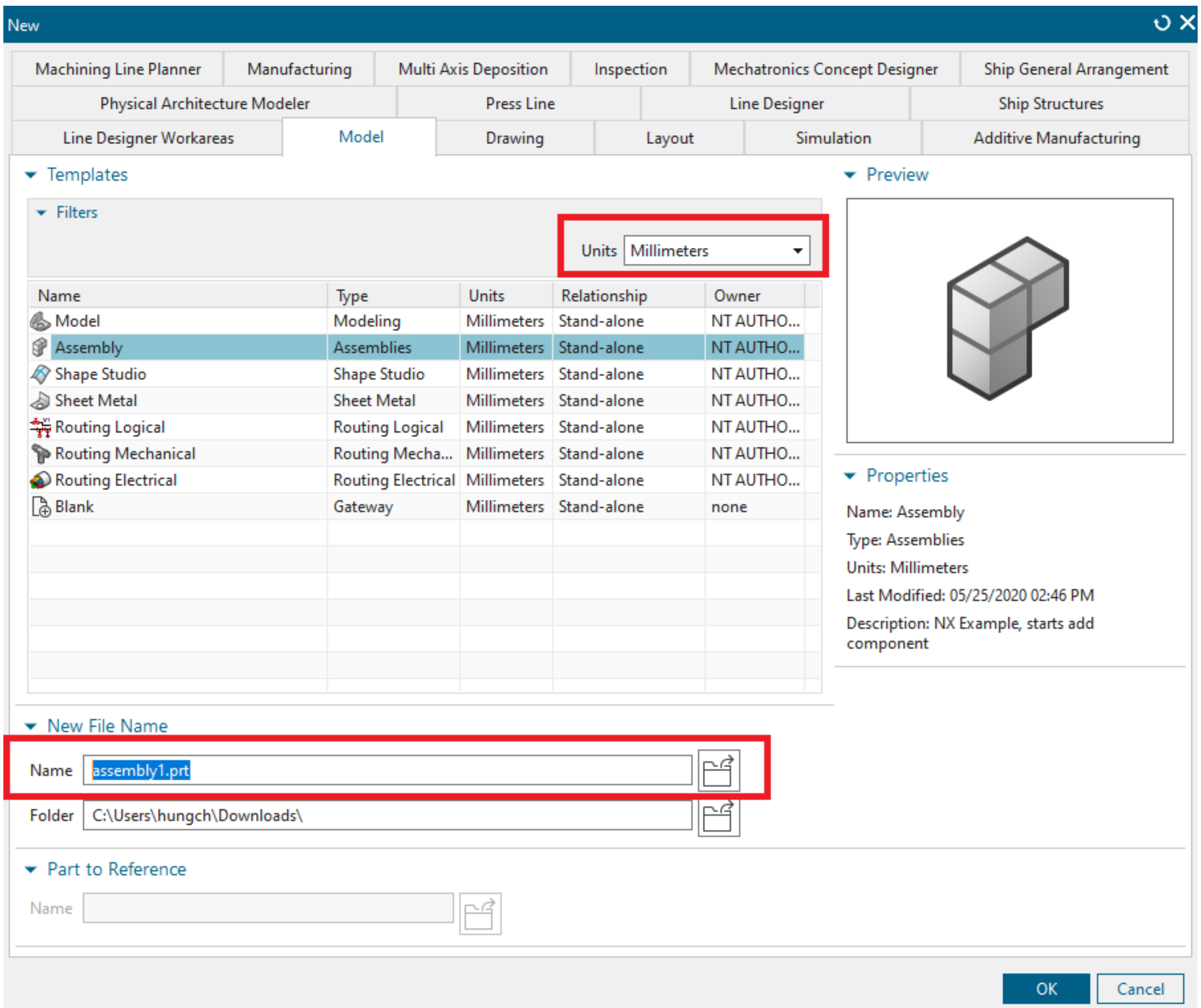
OR

- Press **<Ctrl> + N**

This will open a new session, asking for the type, name and location of the new file to be created.

There are numerous types of files in NX to select from the *Templates* dialogue box located at the center of the window. The properties of the selected file are displayed below the *Preview* on the right side. Since we want to work in the modeling environment and create new parts, only specify the units (inches or millimeters) of the working environment and the name and location of the file. The default unit is millimeters.

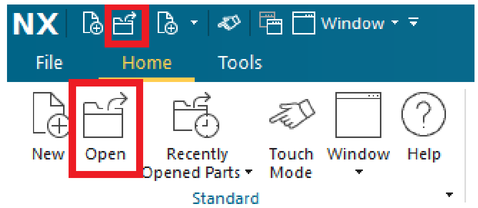
- Enter an appropriate name and location for the file and click **OK**



2.1.3 Open a Part File

There are several ways to open an existing file in NX.

➢ Click on the **Open** buttons at the top-left of the screen



OR

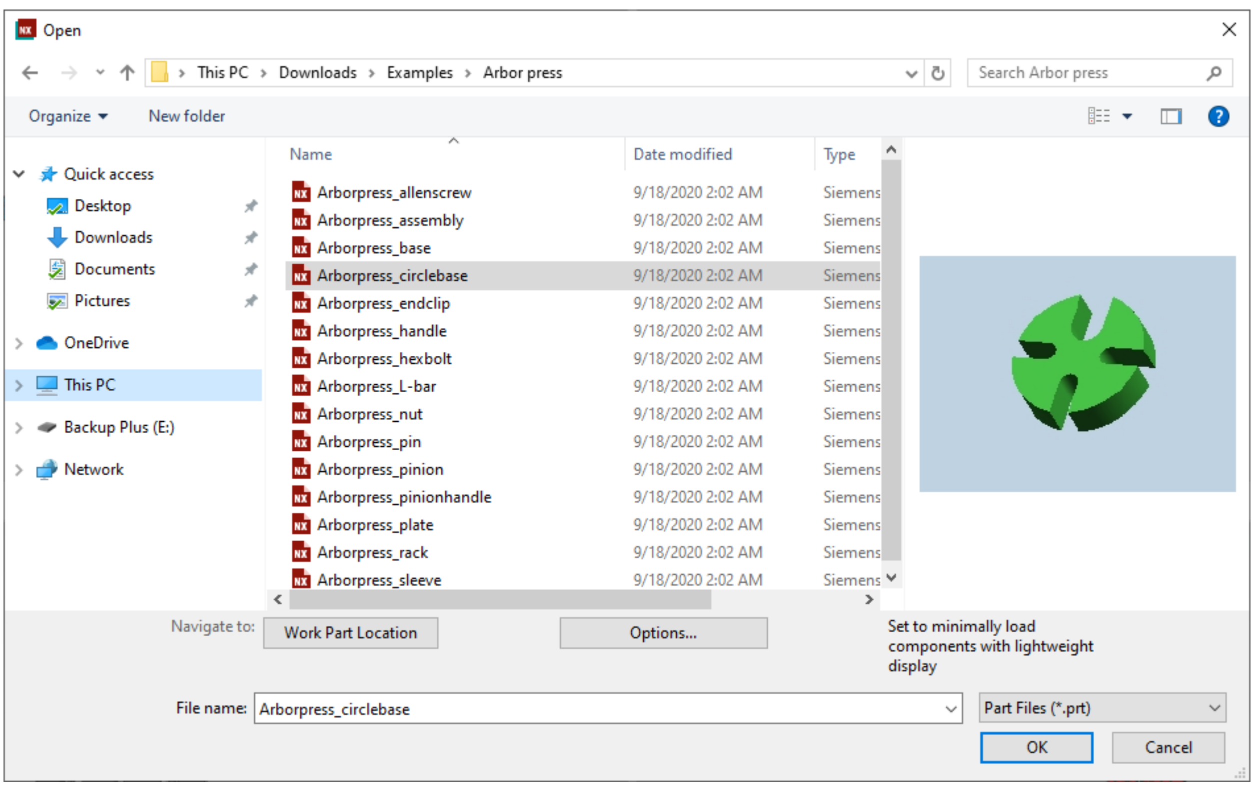
➢ Go through the **File** drop-down menu at the top-left of the screen and click **Open**

OR

➢ Press **<Ctrl> + O**

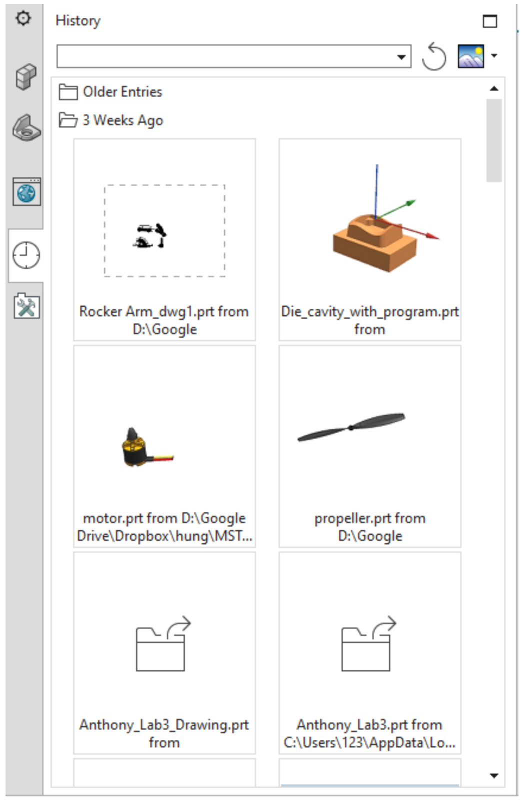
The Open Part File dialog will appear. You can see the preview of the files on the right side of the window. You can disable the *Preview* by un-clicking the box in front of the *Preview* button.

➢ Click **Cancel** to exit the window



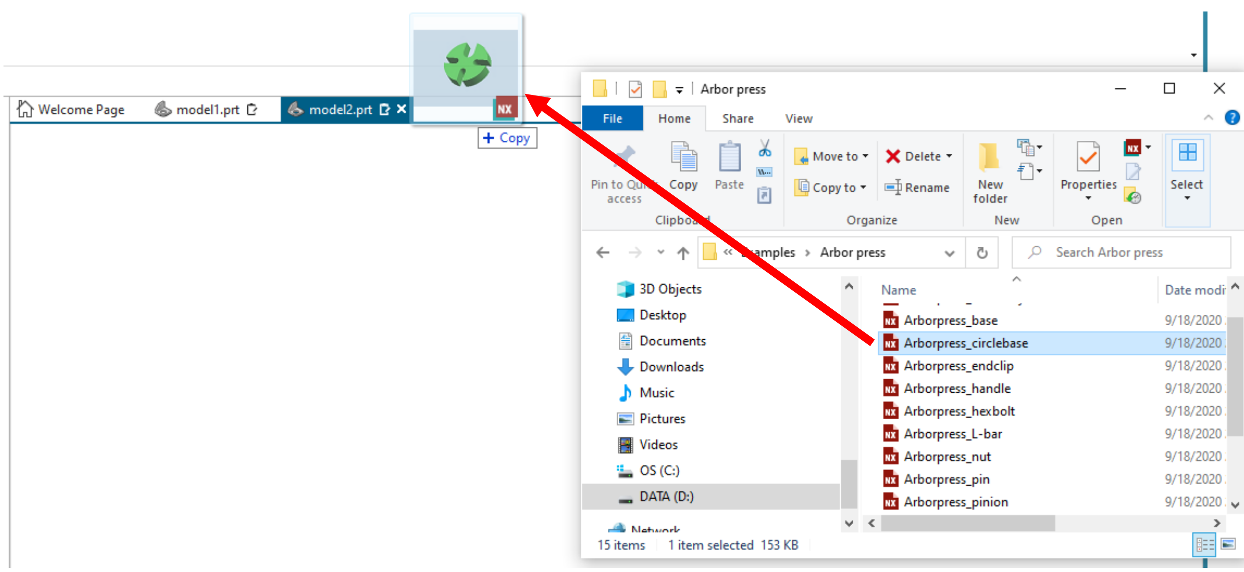
OR

➢ Open files in **History** tab in **Resource Bar**

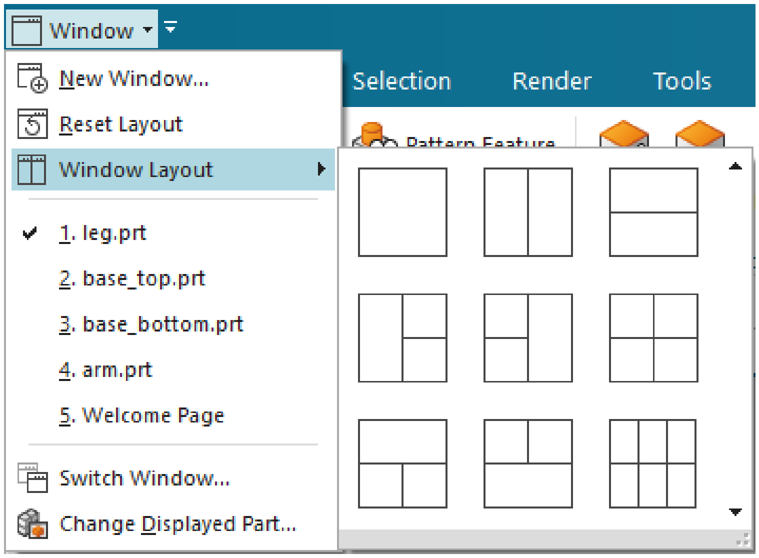


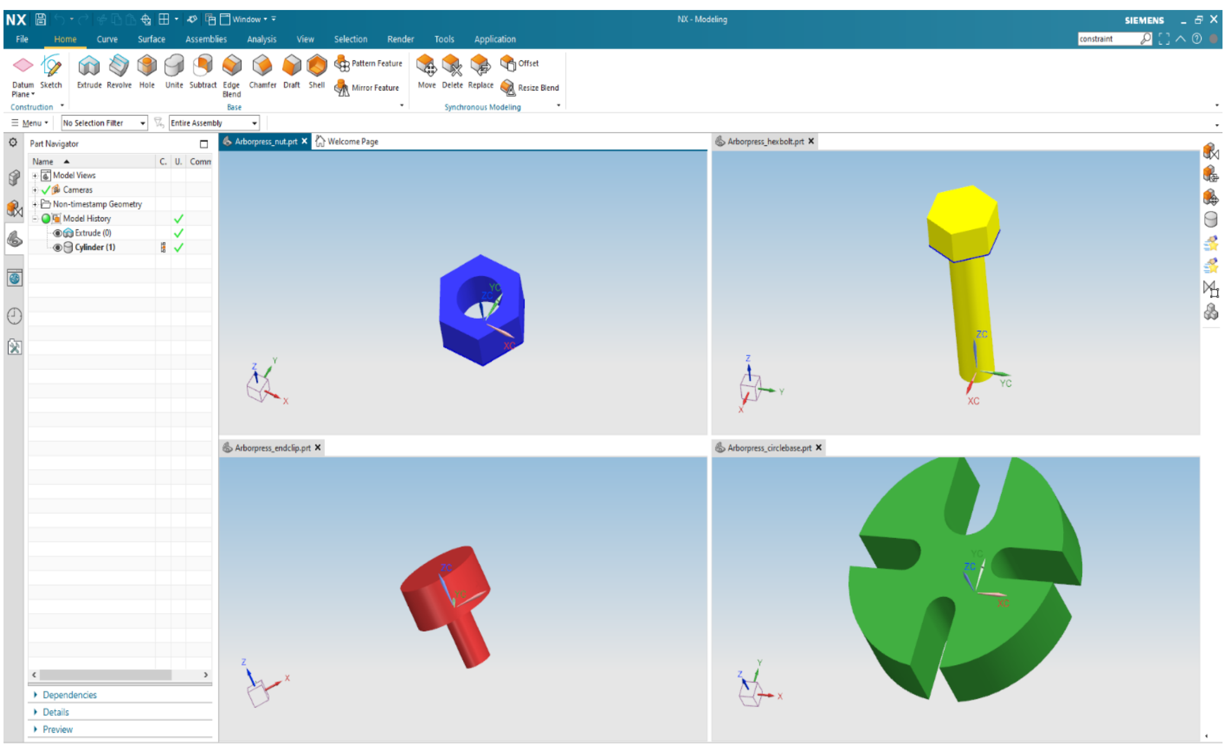
OR

➢ Left-click and drag file to window tab and drop it



When multiple parts are opened, you can simultaneously view multiple opened parts at the same time by clicking **Window Layout** in **Window** on the top of screen and choosing your preferred window layout, or dragging a selected display part to the center of the NX session, then you can select where and how to display multiple opened NX files.





2.2 PRINTING, SAVING AND CLOSING FILES

2.2.1 Print an NX Image

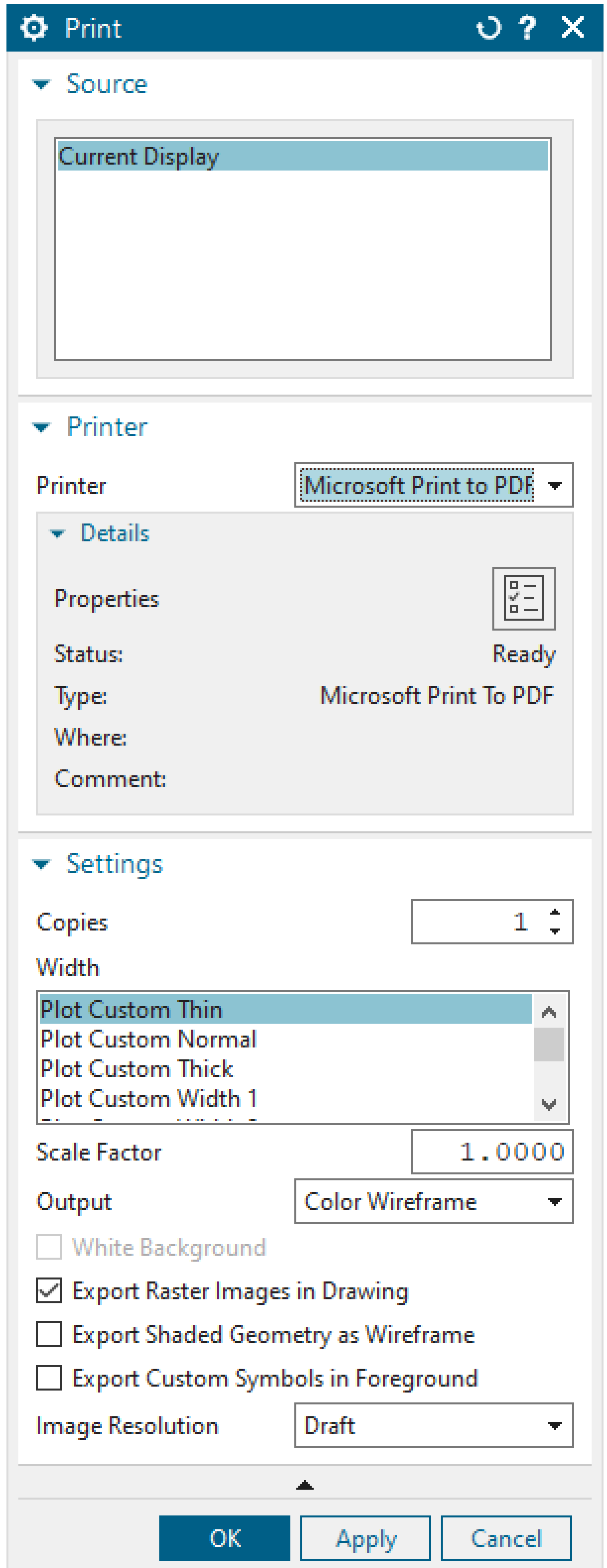
To print an image from the current display,

- Click **File** → **Print**

The following figure shows the Print dialog box. Here, you can choose the printer to use or specify the number of copies to be printed, size of the paper and so on.

You can also select the scale for all the three dimensions. You can also choose the method of printing, i.e. wireframe, solid model by clicking on the *Output* drop down-menu as shown in the Figure on right side

- Click **Cancel** to exit the window



2.2.2 Save Part Files

It is imperative that you save your work frequently. If for some reasons, NX shuts down and the part is not saved, all the work will be lost. To save the part files,

- Click **File** → **Save**

There are five options to save a file:

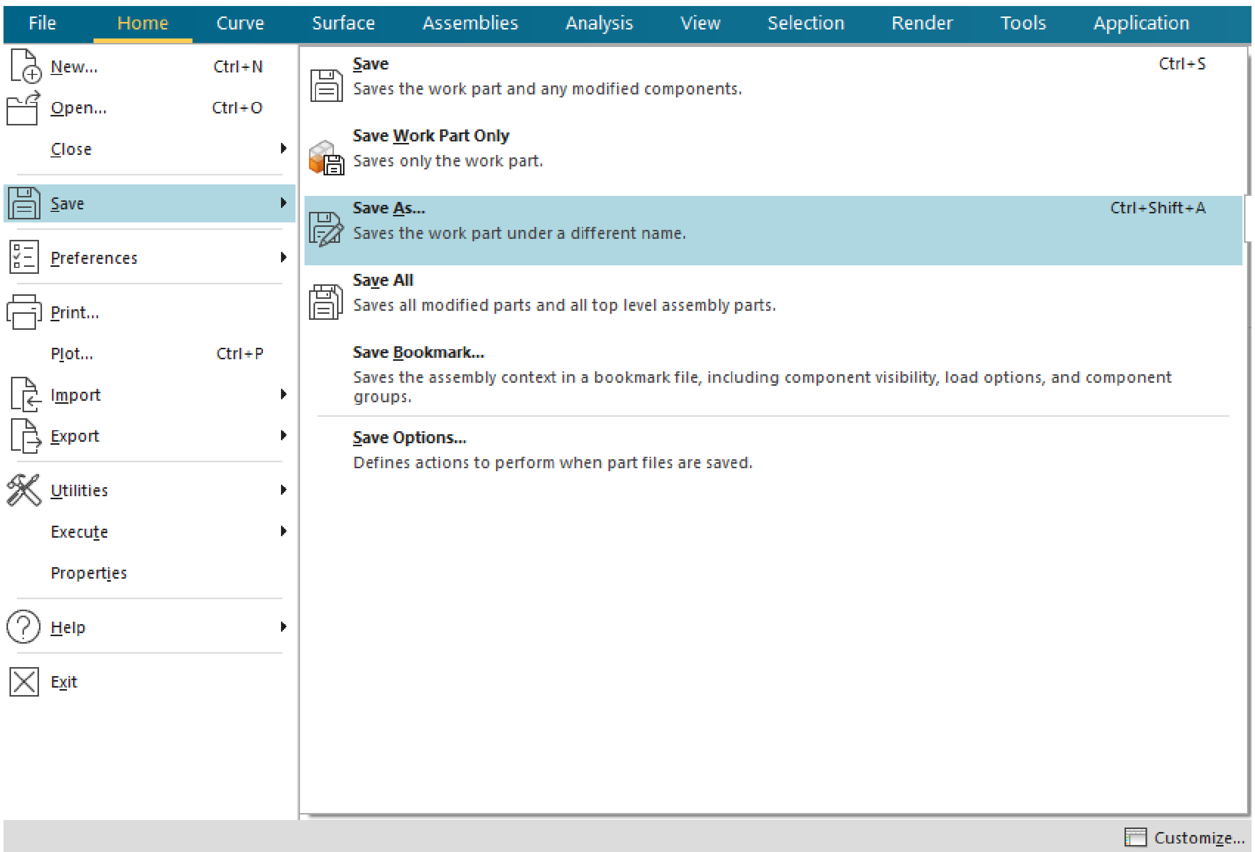
Save: This option will save the part on screen with the same name as given before while creating the part file.

Save Work Part Only: This option will only save the active part on the screen.

Save As: This option allows you to save the part on screen using a different name and/or type. The default type is *.prt*. However, you can save your file as IGES (.igs), STEP (.stp), AutoCAD DXF (.dxf), AutoCAD DWG (.dwg), CATIA Model (.model), CATIA V5 (.catpart), ACIS text (.sat), ACIS Binary (.sab), STEP242 compressed (.stpz), STEP242 XML (stpx), STEP242 compressed XLM (.stpxz) , Wavefront ASCII (stpxz) and IFC (.ifc).

Save All: This option will save all the opened part files with their existing names at their existing locations within your own computer filing system.

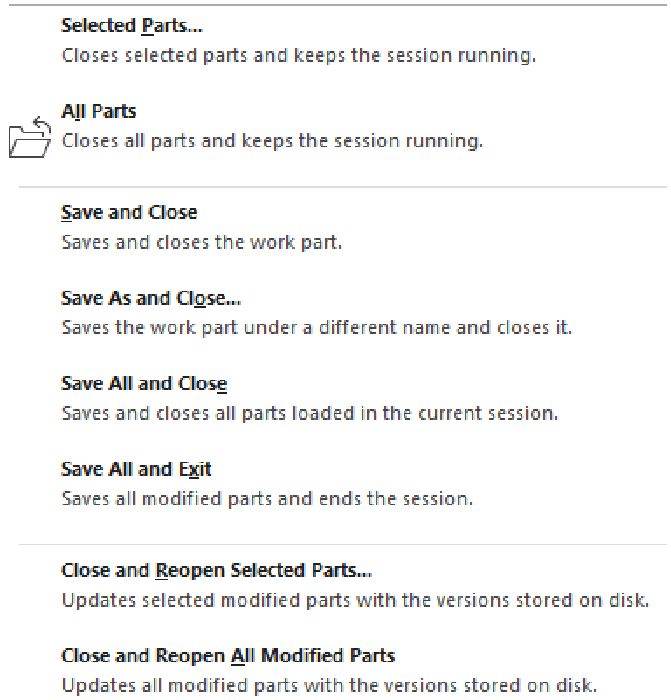
Save Bookmark: This option will save a screenshot and context of the present model on the screen as a .JPEG file and bookmarks.



2.2.3 Close Part Files

You can choose to close the parts that are visible on screen by

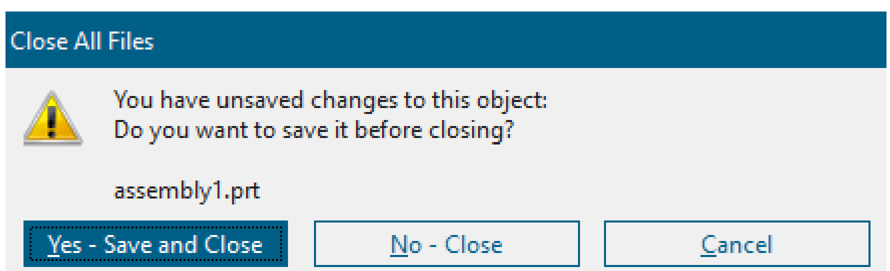
- ➢ Click **File** → **Close**



If you close a file, the file will be cleared from the working memory and any changes that are not saved will be lost. Therefore, remember to select *Save and Close*, *Save As and Close*, *Save All and Close* or *Save All and Exit*. In case of the first three options, the parts that are *selected* or *all parts* will be closed but the NX session keeps on running.

2.2.4 Exit an NX Session

- ➢ Click **File** → **Exit**



If you have files open and have made changes to them without saving, the message will ask you if you really want to exit.

- ➢ Select **No**, save the files and then **Exit**

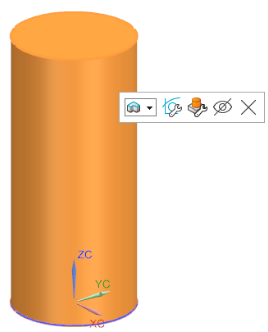
2.3 NX INTERFACE

The user interface of NX is made very simple through the use of different icons and ribbons. Most of the commands can be executed by navigating the mouse around the screen and clicking on the icons. The keyboard entries are mostly limited to entering values and naming files.

2.3.1 Computer Mouse Functionality

2.3.1.1 Left Mouse Button (MB1)

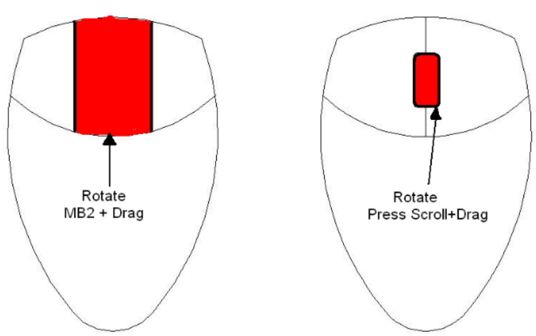
The left mouse button, named Mouse Button 1 (MB1) in NX, is used for *Selection* of icons, menus, and other entities on the graphic screen. Double clicking MB1 on any feature will automatically open the *Edit* Dialog box. Clicking MB1 on an object enables the user to have quick access to several options shown below. These options will be discussed in next chapters.



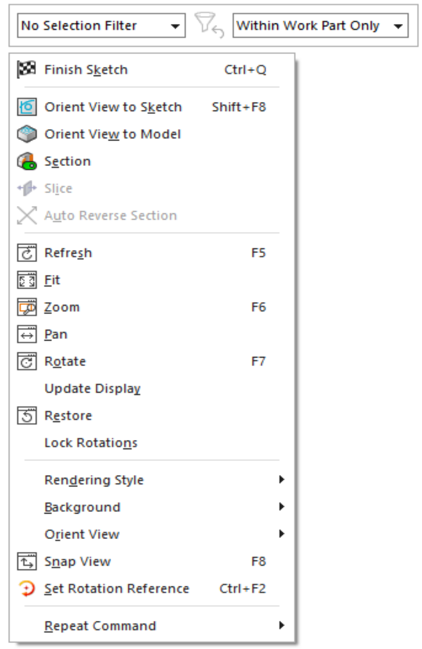
2.3.1.2 Middle Mouse Button (MB2)

The middle mouse button (MB2) or the scroll button is used to *Rotate* the object by pressing, holding and dragging. The model can also be rotated about a single axis. To rotate about the axis horizontal to the screen, place the mouse pointer near the right edge of the graphic screen and rotate. Similarly, for the vertical axis and the axis perpendicular to the screen, click at the bottom edge and top edge of the screen respectively and rotate. If you keep pressing the MB2 at the same position for a couple of seconds, it will fix the point of rotation (an orange circle symbol appears) and you can drag around the object to view.

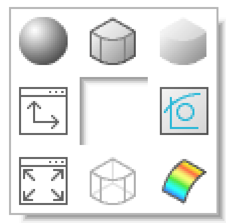
If it is a scroll button, the object can be zoomed in and out by scrolling. Clicking the MB2 will also execute the *OK* command if any pop-up window or dialog box is open.



2.3.1.3 Right Mouse Button (MB3)



MB3 or Right Mouse Button is used to access the user interface pop-up menus. You can access the subsequent options that pop up depending on the selection mode and ***Application***. The figure shown below is in ***Sketch Application***. Clicking on MB3 when a feature is selected will give the options related to that feature (Object/Action Menu).



Clicking MB3 and holding the button will display a set of icons around the feature. These icons feature the possible commands that can be applied to the feature.

2.3.1.4 Combination of Buttons

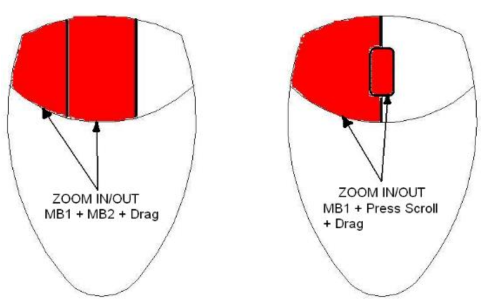
Zoom In /Out:

- Press and hold both **MB1** and **MB2** simultaneously and drag (or use wheel)

OR

- Press and hold **<Ctrl>** button on the keyboard and then press and drag the **MB2**

OR

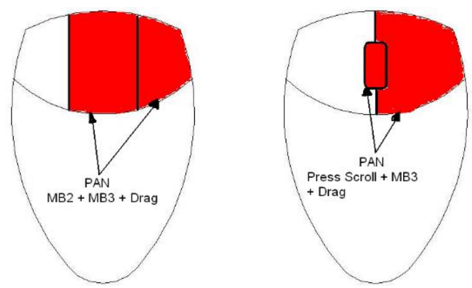


Pan:

- Press and hold both the **MB2** and **MB3** simultaneously and drag

OR

- Press and hold **<Shift>** button on the keyboard and press and drag the **MB2**

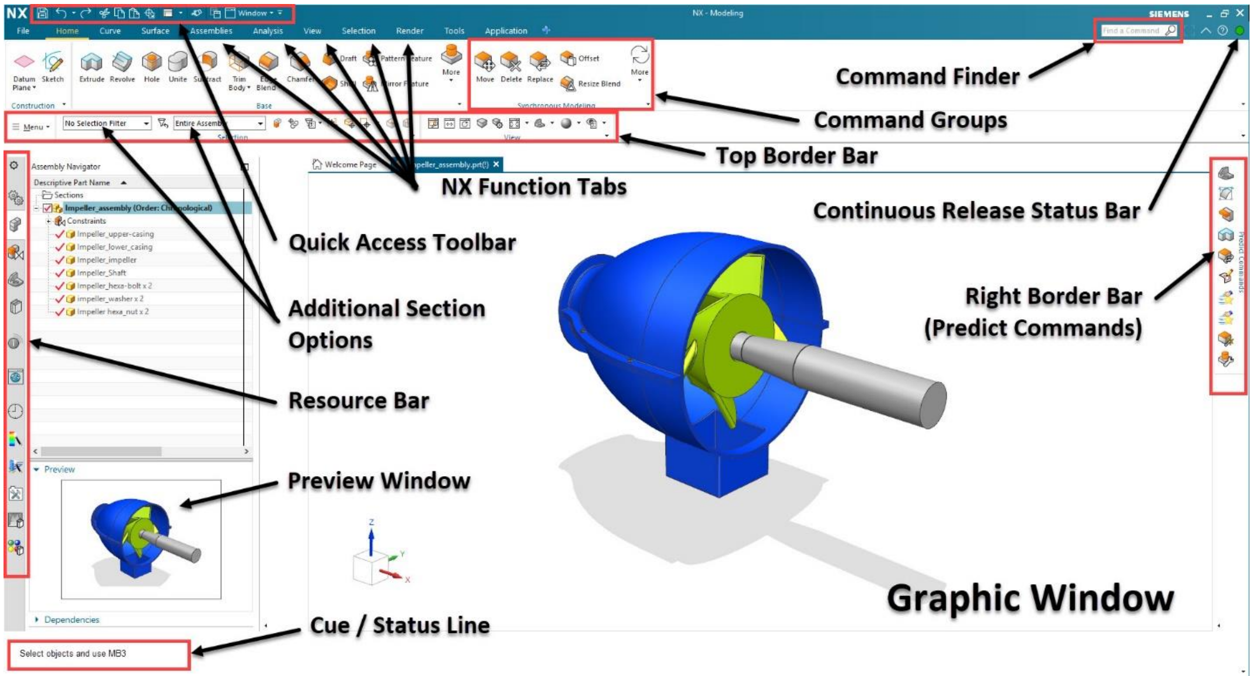


Shortcut to menus:

- Press and hold **<Ctrl> + <Shift>** and **MB1**, **MB2** and **MB3** to see shortcuts to **Feature**, **Create Sketch**, and **Synchronous Modeling** groups, respectively

2.3.2 NX Gateway

The following figure shows the typical layout of the NX window when a file is opened. This is the Gateway of NX from where you can select any module to work on such as modeling, manufacturing, etc. It has to be noted that these toolbars may not be exactly on the same position of the screen as shown below. The toolbars can be placed at any location or position on the screen. Look out for the same set of icons.



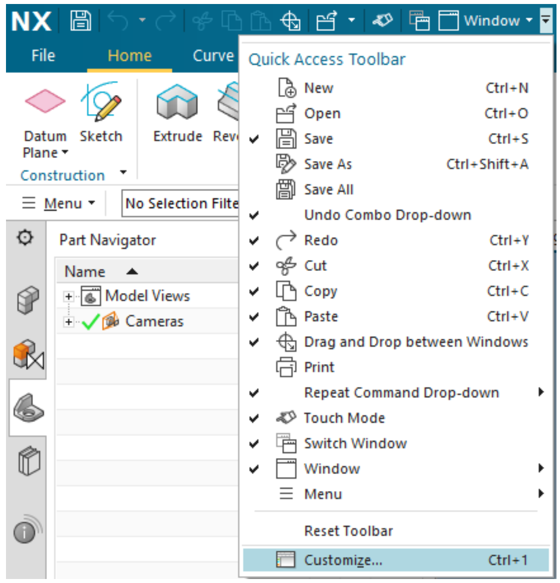
2.3.2.1 Ribbon Bar

The ribbon bar interface gives the user the ability to access the different commands easily without reducing the graphics window area. Commands are organized in ribbon bars under different tabs and groups for easy recognition and accessibility.

For example, in the ribbon bar shown in the figure above, we have home, curve, etc. tabs. In the home tab, we have direct sketch, feature, synchronous modeling and surface groups. And in each group, we have a set of featured commands.

2.3.2.2 Quick Access Toolbar

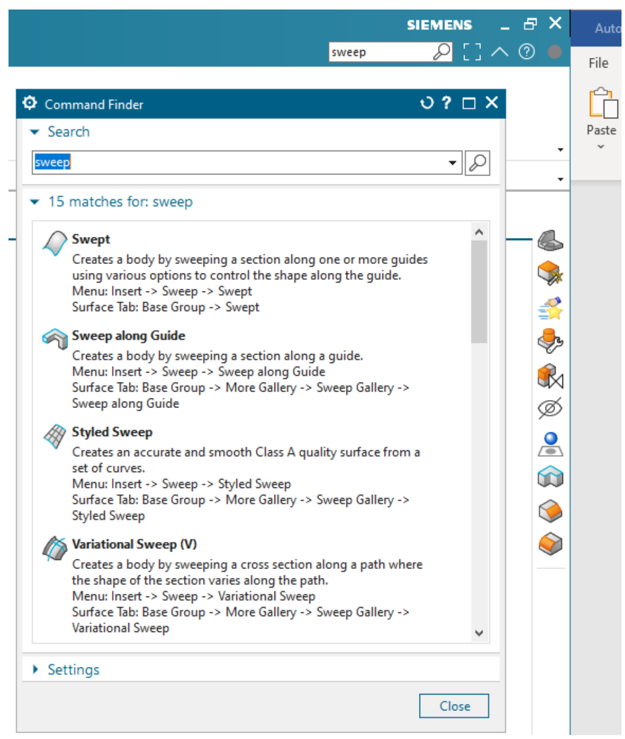
The quick access toolbar has most commonly used buttons (Save, Undo, Redo, Cut, Copy, Paste, Drag and Drop between Windows, Repeat Command Drop-down, Touch Mode and Switch Window) to expedite the modeling process. You may easily customize these buttons as shown in the figure below.



2.3.2.3 Command Finder

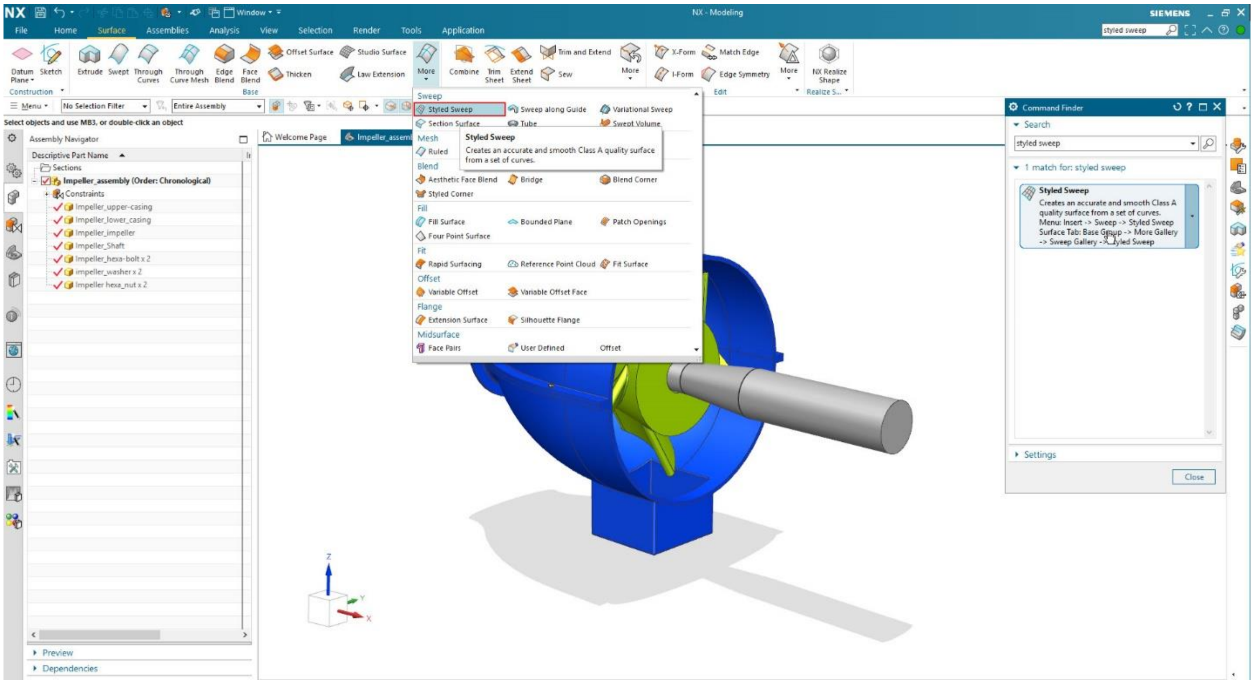
If you do not know where to find a command, use *Command Finder*. Let's say we have forgotten where the *Styled Sweep* is.

- Type **sweep** in the *Command Finder*
- Hover the mouse over **Styled Sweep**
- NX will show you the path to the command: **Menu → Insert → Sweep → Styled Sweep**



Type **sweep** in the *Command Finder* → Click on **Styled Sweep** in the *Command Finder* window

Command Finder also supports commands from other CAD systems, i.e. CATIA, Creo, and Solidworks. Users can type in commands in those systems and NX will find similar commands in NX.



2.3.2.4 Top-border

The most important button in the top-border is the *menu* button. Most of the features and functions of the software are available in the *menu*. The *Selection Bar* displays the selection options. These options include the *Filters, Components/Assembly,* and *Snap Points* for selecting features. Most common buttons in the *View* tab are also displayed in the *Top-border*.

2.3.2.5 Right-border

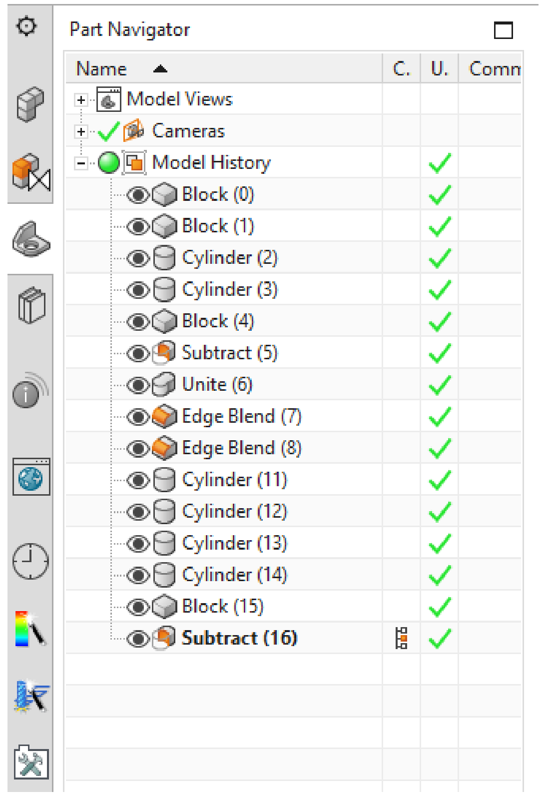
The common features and functions of the software you frequently used are available in the right-border. The NX also predicts and lists some features and functions you may need to use in the next step based on your current NX working task in the right-border.

2.3.2.6 Resource Bar

The Resources Bar consists of multiple tabs in one place using very little user interface space. NX places all navigator windows (*Assembly, Constraint* and *Part*) in the *Resource Bar*, as well as the *Reuse Library, HD3D Tools, Web Browser, History Palette, Process Studio, Manufacturing Wizards*, and *Roles*. Users can create their own custom Resource Bar in these specific tabs (History, Process Studio, Manufacturing Wizard, etc.) by moving your mouse to blank space of Resource Bar, then MB3 (right-click), and then select a New Palette. Two of the most important widows are explained below.

Part Navigator

- Click on the **Part Navigator** icon, the third icon from the top on the **Resource bar**



The *Part Navigator* provides a visual representation of the parent-child relationships of features in the work part in a separate window in a tree type format. It shows all the primitives, entities used during modeling. It allows you to perform various editing actions on those features. For example, you can use the *Part Navigator* to *Suppress* or *Unsuppress* the features or change their parameters or positioning dimensions. Removing the green tick mark will 'Suppress' the feature. The software will give a warning if the parent child relationship is broken by suppressing any particular feature.

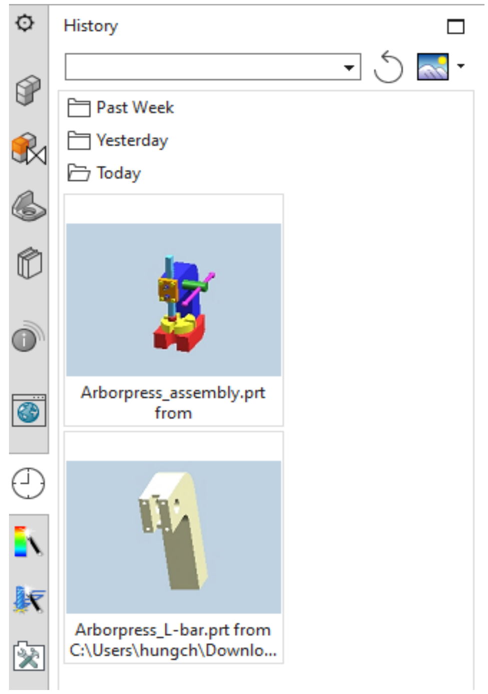
The *Part Navigator* is available for all NX applications and not just for modeling. However, you can only perform feature-editing operations when you are in the Modeling module. Editing a feature in the *Part Navigator* will automatically update the model. Feature editing will be discussed later.

History

- Click on the **History** icon, the seventh from the top on the **Resource bar tab**

The *History Palette* provides fast access to recently opened files or other palette entries. It can be used to reload parts that have been recently worked on or to repeatedly add a small set of palette items to a model.

The *History Palette* remembers the last palette options that were used and the state of the session when it was closed. NX stores the palettes that were loaded into a session and restores them in the next session. The system does not clean up the *History Palette* when parts are moved.



To re-use a part, drag and drop it from the History Palette, or just select it from that display window to the Graphics Window. To reload a part, click on a saved session bookmark.

2.3.2.7 Cue Line

The *Cue Line* displays prompt messages that indicate the next action that needs to be taken. To the right of the Cue line, the *Status Line* is located which displays messages about the current options or the most recently completed function.

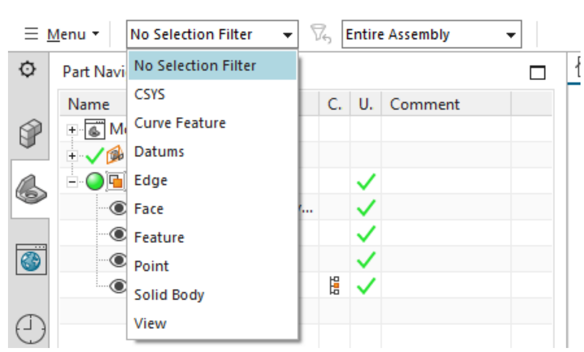
The *Progress Meter* is displayed in the *Cue Line* when the system performs a time-consuming operation such as loading a large assembly. The meter shows the percentage of the operation that has been completed. When the operation is finished, the system displays the next appropriate cue.

2.3.3 Geometry Selection

You can filter the selection method, which facilitates easy selection of the geometry in a close cluster. In addition, you can perform any of the feature operation options that NX intelligently provides depending on the selected entity. Selection of items can be based on the degree of the entity like, selection of *Geometric* entities, *Features* and *Components*. The selection method can be opted by choosing one of the icons in the *Selection Toolbar*.

2.3.3.1 Feature Selection

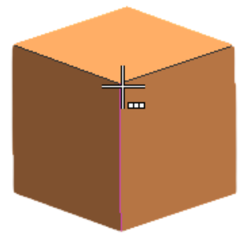
Clicking on any of the icons lets you select the features in the part file. It will not select the basic entities like edges, faces etc. The features selected can also be applied to a part or an entire assembly depending upon the requirement.



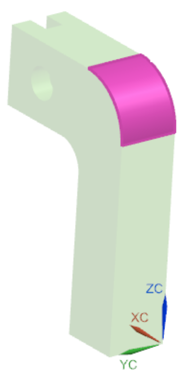
Besides that, the filtering of the features can be further narrowed down by selecting one of the desired options in the drop-down menu as shown in the figure. For example, selecting *Curve* will highlight only the curves in the screen. The default is *No Selection Filter*.

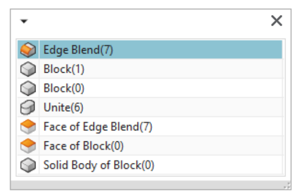
2.3.3.2 General Object Selection

Navigate the mouse cursor closer to the entity until it is highlighted with a magenta color and click the left mouse button to select any geometric entity, feature, or component.

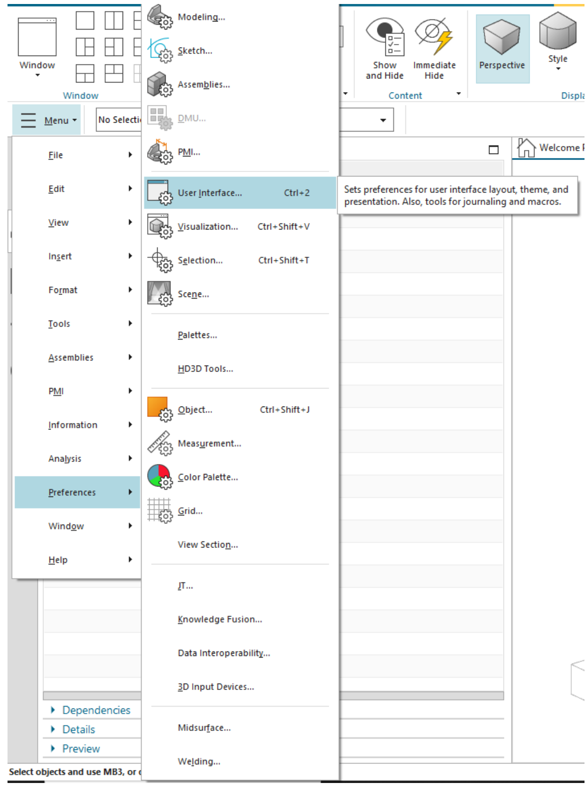


If you want to select an entity that is hidden behind the displayed geometry, place the mouse cursor roughly close to that area on the screen such that the cursor ball occupies a portion of the hidden geometry projected on the screen. After a couple of seconds, the ball cursor turns into a ***plus*** symbol (…) as shown in the image. Click the left mouse button (MB1) to get a *Selection Confirmation* dialog box as shown in the following figure below. This *QuickPick* menu consists of the list of entities captured within the ball of the cursor. The entities are arranged in ascending order of the *degree* of the entity. For example, edges and vertices are assigned lower numbers while solid faces are given higher numbers. By moving the cursor on the numbers displayed, NX will highlight the corresponding entity on the screen in a magenta color.





2.3.4 User Preferences



- ➢ Choose **Preferences** on the **Menu button** (located to top left of the main window) to find the various options available

User Preferences are used to define the display parameters of new objects, names, layouts, and views. You can set the layer, color, font, and width of created objects. You can also design layouts and views, control the display of object and view names and borders, change the size of the selection ball, specify the selection rectangle method, set chaining tolerance and method, and design and activate a grid. Changes that you make using the Preferences menu override any counterpart customer defaults for the same functions.

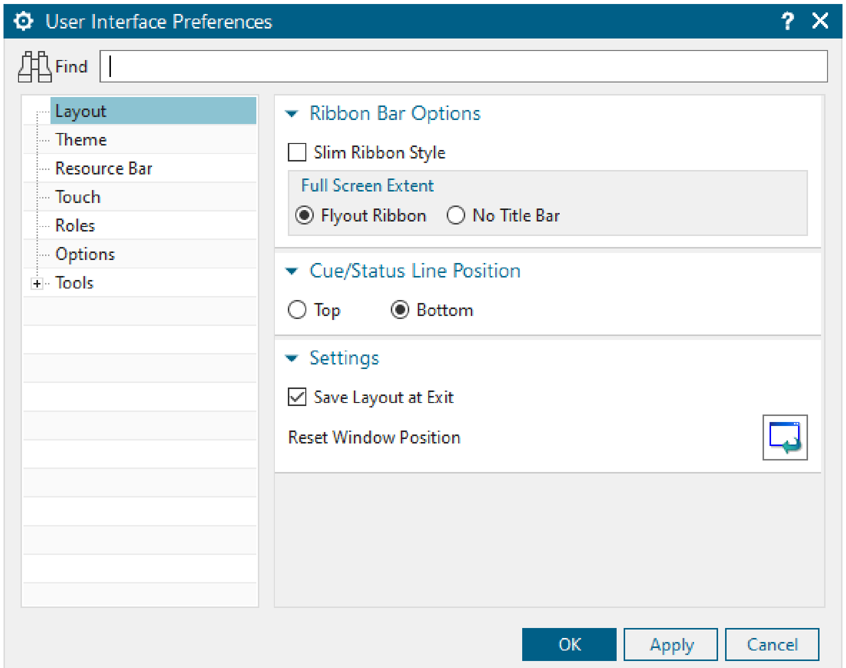
2.3.4.1 User Interface

- ➢ Choose **Preferences → User Interface** to find the options in the dialog box

The *User Interface* option customizes how NX works and interacts to specifications you set. You can control the location, size and visibility status of the main window, graphics display, and information window. You can set the number of decimal places (precision) that the system uses for both input text fields and data displayed in the information window. You can also specify a full or small dialog for file selection. You can also set macro options and enable a confirmation dialog for *Undo* operations.

- The *Layout* tab allows you to select the *User Interface Environment*
- The *Theme* tab allows you change NX Theme from Light (recommended) to Light Gray, Dark, or Classic
- The *Resource Bar* tab allows you change its display location
- The *Touch* tab lets you use touch screens

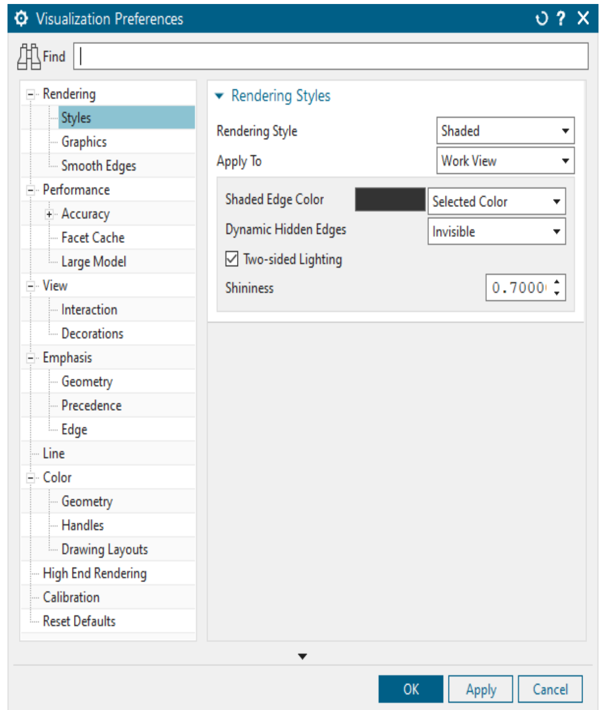
- The *Options* tab allows you, among others, to set the precision level (in the *Information Window*)
- The *Journal* tab in the *Tools* allows you to use several programming languages
- The *Macro* tab in the *Tools* allows you to set the pause while displaying animation



2.3.4.2 Visualization

- Choose **Preferences → Visualization** to find the options in the dialog box

This dialog box controls attributes that affect the display in the graphics window. Some attributes are associated with the part or with particular *Views* of the part. The settings for these attributes are saved in the part file. For many of these attributes, when a new part or a view is created, the setting is initialized to the value specified in the *Customer Defaults* file. Other attributes are associated with the session and apply to all parts in the session. The settings of some of these attributes are saved from session to session in the registry. For some session attributes, the setting can be initialized to the value specified by customer default, an environment variable.

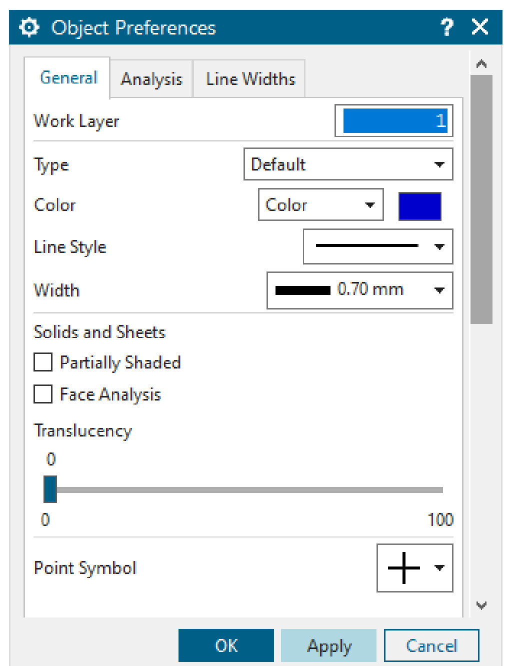


- ➢ Choose **Preferences → Color Pallet** to find the options in the dialog box

You can change and observe the *Color* and *Translucency* of objects.

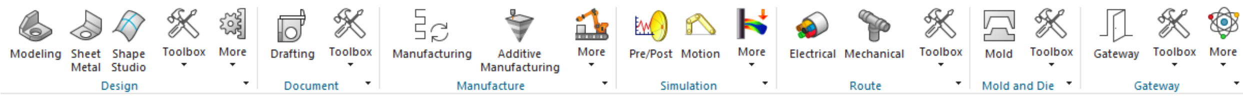
- ➢ Click **Preferences → Object**

This will pop up a dialog window *Object Preferences*. You can also apply this setting to individual entities of the solid. For example, you can click on any particular surface of the solid and apply the *Display* settings.



2.3.5 Applications

Applications can be opened using the *File* option located at the top left corner of the main window OR the *Applications* tab above the *Ribbon bar*. You can select the type of application you want to run. For example, you can select *Modeling*, *Drafting*, *Manufacturing*, and so on as shown in the figure. The default *Application* that starts when you open a file or start a new file is *Modeling*. We will introduce some of these *Application* in the next chapters.

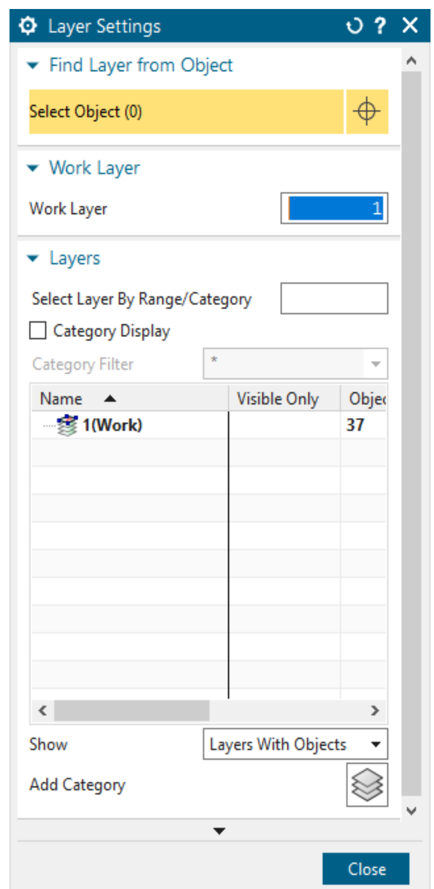


2.4 LAYERS

Layers are used to store objects in a file, and work like containers to collect the objects in a structured and consistent manner. Unlike simple visual tools like *Show* and *Hide*, *Layers* provide a permanent way to organize and manage the visibility and select-ability of objects in your file.

2.4.1 Layer Control

With NX, you can control whether objects are visible or selectable by using *Layers*. A *Layer* is a system-defined attribute such as color, font, and width that all objects in NX must have. There are 256 usable layers in NX, one of which is always the *Work Layer*. Any of the 256 layers can be assigned to one of four classifications of status.



- *Work*
- *Selectable*
- *Visible Only*
- *Invisible*

The *Work Layer* is the layer that objects are created ON and is always visible and selectable while it remains the *Work Layer*. *Layer 1* is the default *Work Layer* when starting a new part file. When the *Work Layer* is changed to another type of layer, the previous *Work Layer* automatically becomes *Selectable* and can then be assigned a status of *Visible Only* or *Invisible*.

The number of objects that can be on one layer is not limited. You have the freedom to choose whichever layer you want to create the object on and the status of that layer.

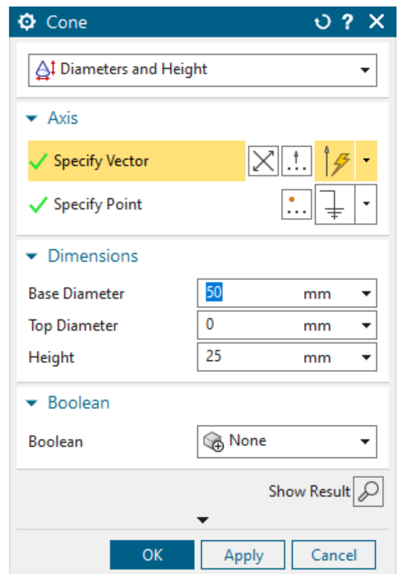
To assign a status to a layer or layers,

➢ Choose **Menu** → **Format** → **Layer Settings**

However, it should be noted that the use of company standards in regards to layers would be advantageous to maintain a consistency between users and their NX files.

2.4.2 Commands in Layers

We will follow simple steps to practice the commands in *Layers***.** First, we will create two objects (Solids) by the method as follows. The details of *Solid Modeling* will be discussed in the next chapter. The solids that we draw here are only for practice in this chapter.



➢ Choose **File** → **New**

Name the file and choose a folder in which to save it. Make sure you select the units to be *millimeters* in the drop-down menu. Choose the file type as *Model*

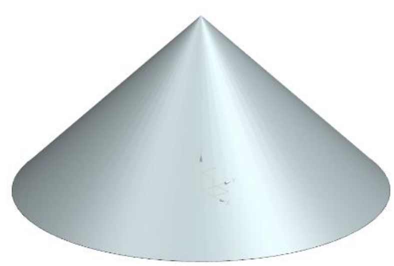
➢ Choose **Menu** → **Insert** → **Design Feature** → **Cone**

➢ Choose **Diameter and Height** under **Type**

➢ Click **OK**

➢ **Right-click** on the screen and choose **Orient View** → **Trimetric**

➢ **Right-click** on the screen and choose **Rendering Style** → **Shaded**



You will be able to see a solid cone similar to the picture on right.

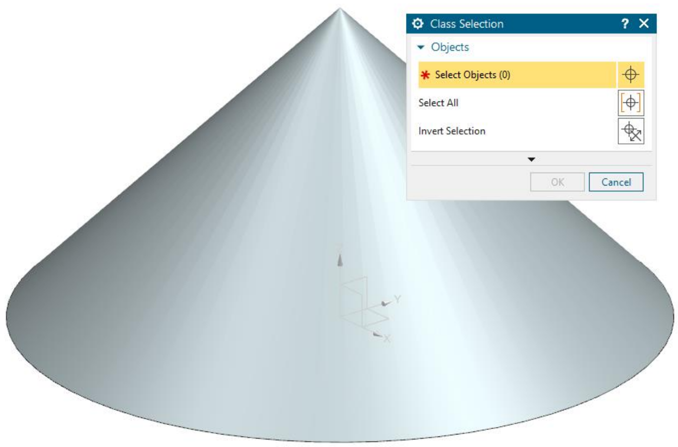
Now let us practice some *Layer* Commands.

➢ Choose **Menu** → **Format** → **Move to Layer**

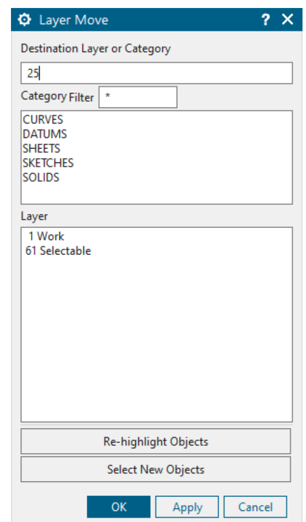
You will be asked to select an object

➢ Move the cursor on to the *Cone* and click on it so that it becomes highlighted

➢ Click **OK**



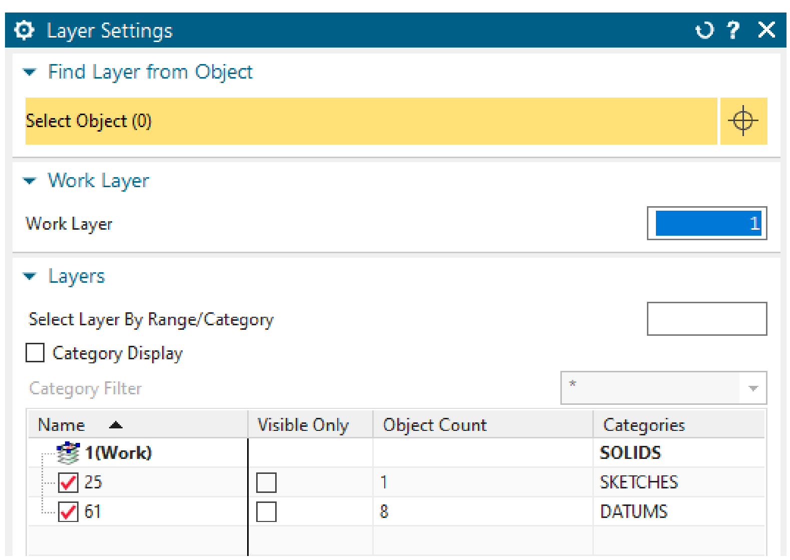
- In the **Destination Layer or Category** space at the top of the window, type 25 and Click **OK**



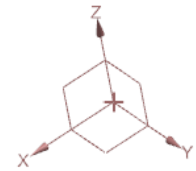
The *Cone* has now gone to the 25th layer. It can no longer be seen in Layer 1.

- To see the Cone, click **Menu → Format → Layer Settings**
- You can see that *Layer 25* has the object whereas the default *Work Layer 1* has no objects.

The Cone will again be seen on the screen. Save the file as we will be using it later in the tutorial.



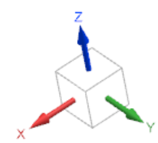
2.5 COORDINATE SYSTEMS



There are different coordinate systems in NX. A three-axis symbol is used to identify the coordinate system.

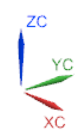
2.5.1 Absolute Coordinate System

The [*Absolute Coordinate System*]() is the coordinate system from which all objects are referenced. This is a fixed coordinate system and the locations and orientations of every object in NX modeling space are related back to this system. The *Absolute Coordinate System* (or Absolute CSYS) also provides a common frame of reference between part files. An absolute position at *X=1*, *Y=1*, and *Z=1* in one part file is the same location in any other part file.



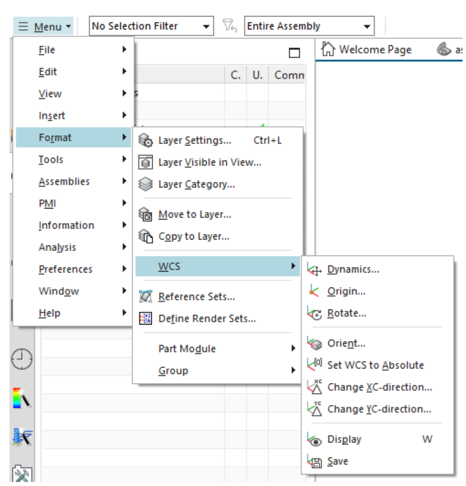
The *View Triad* on the bottom-left provides a visual indicator that represents the ORIENTATION of the *Absolute Coordinate System* of the model. Users can select any of the faces of the *View Triad* square block and NX will orient the view for that orientation.

2.5.2 Work Coordinate System



The *Work Coordinate System* (WCS) is what you will use for construction when you want to determine orientations and angles of features. The axes of the WCS are denoted XC, YC, and ZC. (The "C" stands for "current"). It is possible to have multiple coordinate systems in a part file, but only one of them can be the work coordinate system.

2.5.3 Moving the WCS



In this section, you will learn [how to move, translate and rotate the WCS](). To display the WCS (if WCS is not shown in NX graphics window), you can hit "w" key to turn on WCS and the WCS will turn off if you hit "w" key again. Then, by double click on the WCS, NX will display an active WCS with vectors and rotational balls, allowing you to translate and rotate the WCS.

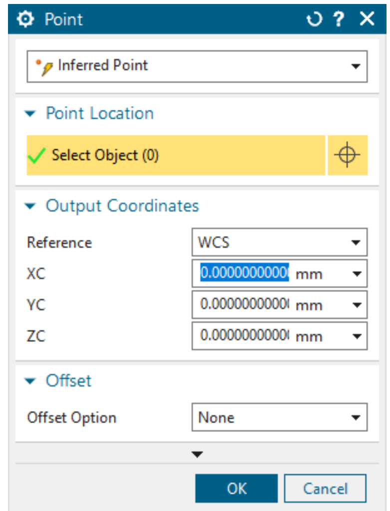
➢ Choose **Menu** → **Format** → **WCS**

2.5.3.1 Translate the WCS

This procedure will move the WCS origin to any point you specify, but the orientation (direction of the axes) of the WCS will remain the same.

- ➢ Choose **Menu → Format → WCS → Origin**

The *Point Constructor* dialog is displayed. You either can specify a point from the drop down menu at the top of the dialog box or enter the X-Y-Z coordinates in the XC, YC, and ZC fields.



The majority of the work will be in relation to the *Work Coordinate System* rather than the *Absolute Coordinate System*. The default is the WCS**.**

2.5.3.2 Rotate the WCS

You can also rotate the WCS around one of its axes.

- ➢ Choose **Menu → Format → WCS → Rotate**

The dialog shows six different ways to rotate the WCS around an axis. These rotation procedures follow the right-hand rule of rotation. You can also specify the angle to which the WCS be rotated.

You can save the current location and orientation of the WCS to use as a permanent coordinate system.

- ➢ Choose **Menu → Format → WCS → Save**

2.6 TOOLBARS

Toolbars contain icons, which serve as shortcuts for many functions. The figure on the right shows the main Toolbar items normally displayed. However, you can find many more icons for different feature commands, based on the module selected and how the module is customized.

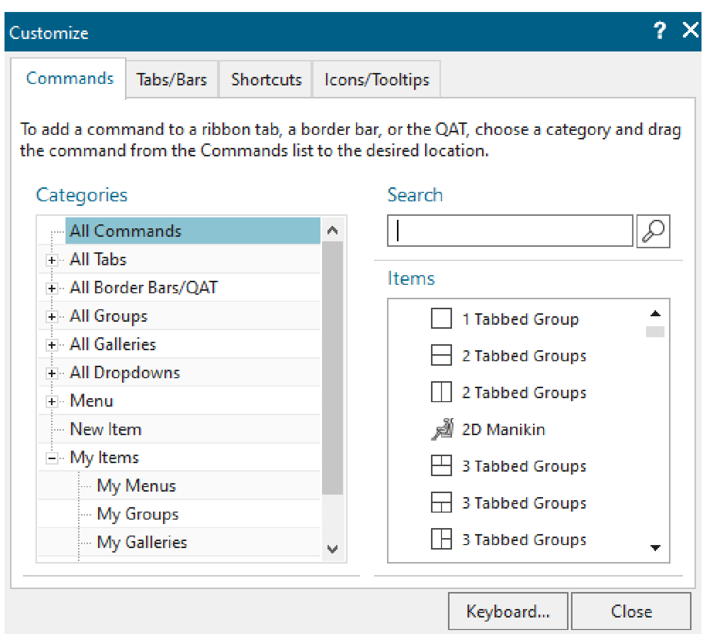
- ➢ **Right-Clicking** anywhere on the existing toolbars gives a list of other **Toolbars**. You can add any of the toolbars by checking them.

Normally, the default setting should be sufficient for most operations but during certain operations, you might need additional toolbars. If you want to add buttons pertaining to the commands and toolbars,

- ➢ Click on the **pull-down arrow** on any of the Toolbars and choose **Customize**.

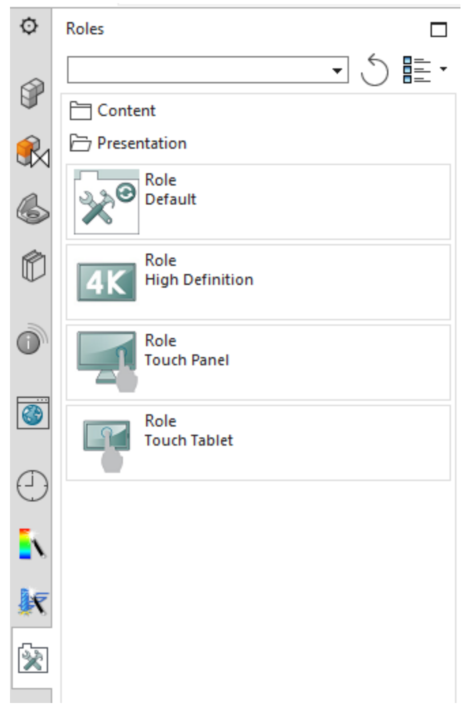
This will pop up a *Customize* dialog window with all the Toolbars and commands pertaining to each Toolbar under *Commands* tab. To add a command,

- ➢ Choose a category and drag the command from the *Commands list* to the desired location.



You can customize the settings of your NX interface by clicking on the *Roles* tab on the *Resource Bar*.

The *Roles* tab has different settings of the toolbar menus that are displayed on the NX interface. It allows you to customize the toolbars you desire to be displayed in the Interface.



CHAPTER 3 – TWO DIMENSIONAL SKETCHING

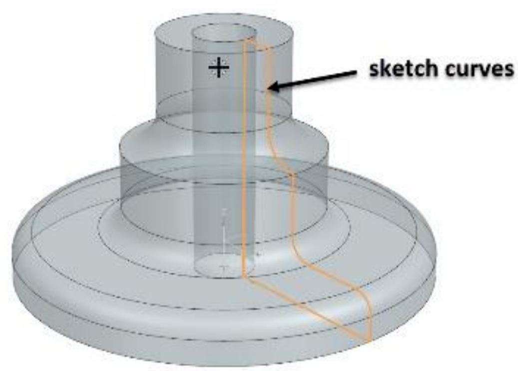
In this chapter, you will learn how to create and edit sketches in NX. You can directly create a sketch on a *Plane* in *Modeling* application. In most cases, *Modeling* starts from a 2D sketch and then *Extrude*, *Revolve* or *Sweep* the sketch to create solids. Many complex shapes that are otherwise very difficult to model can easily be drawn by sketching. In this chapter, we will see some concepts of sketching and then proceed to sketch and model some parts.

3.1 OVERVIEW

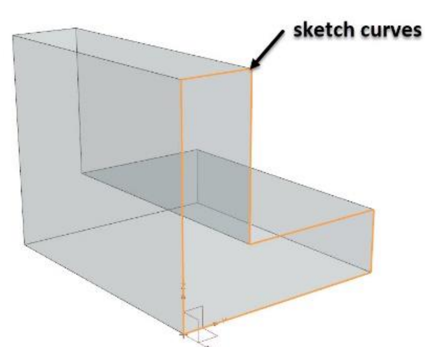
NX sketch is a feature with named set of curves joined in a string that when swept, form a solid. The sketch represents the outer boundary of that part. The curves are created on a plane in the sketcher. In the beginning, these curves are drawn without any exact dimensions. Then, *Dimensional Constraints* as well as *Geometric Constraints* are applied to fully constrain the sketch. These will be discussed in detail later in this chapter.

After sketching is completed, there are different ways to use them to generate 3D parts:

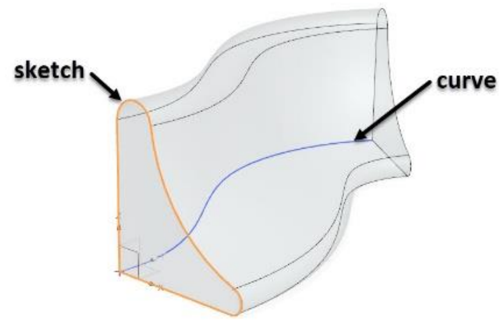
- A sketch can be revolved



- A sketch can be extruded



- A sketch can be swept along a guide (line):



Features created from a sketch are associated with it; i.e., if the sketch changes so do the features.

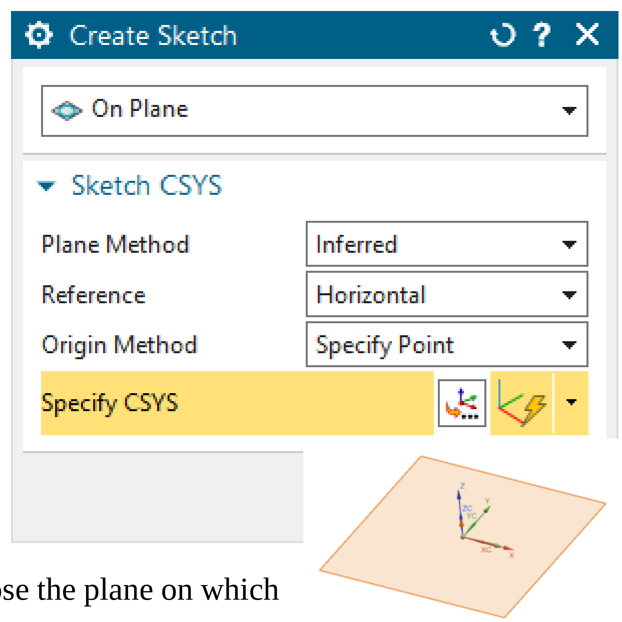
The advantages of using sketching to create parts are:

- The curves used to create the profile outline are very flexible and can be used to model unusual shapes.
- The curves are parametric, hence associative and they can easily be changed or removed.
- If the plane in which the sketch is drawn is changed, the sketch will be changed accordingly.
- Sketches are useful when you want to control an outline of a feature, especially if it may need to be changed in the future. Sketches can be edited very quickly and easily.

3.2 SKETCHING ENVIRONMENT

In NX you can create sketch using two ways. The first method creates the *Sketch* in the current environment and application. For this,

➢ Choose **Menu** → **Insert** → **Sketch**



In the other method you can create Sketch using

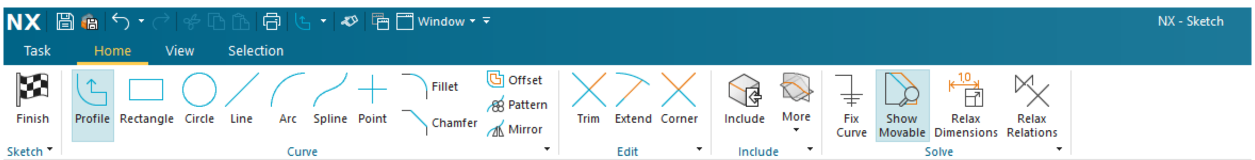
➢ Choose **Sketch** in **Home** toolbar

In either case, a dialog box pop-ups asking you to define the *Sketch Plane*. The screen will display the sketch options. You can choose the *Sketch Plane*, direction of sketching and type of plane for sketching. When you create a sketch using the *Create Sketch* dialog box, you can choose the plane on which the sketch can be created by clicking on the coordinate frame as shown. This will highlight the plane you have selected. The default plane selected is XC-YC. However, you can choose to sketch on another plane. If there are any solid features created in the model beforehand, any of the flat surfaces can also be used as a sketching plane.

- Choose the **XC-YC** plane and click **OK**

The sketch plane will appear, and the X-Y directions will be marked.

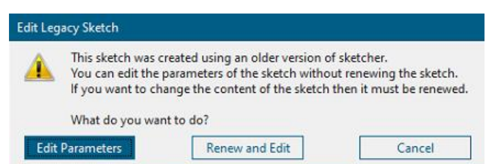
The main screen will change to the *Sketching Environment*. The XY plane is highlighted as the default plane for sketching. This is the basic sketch window.

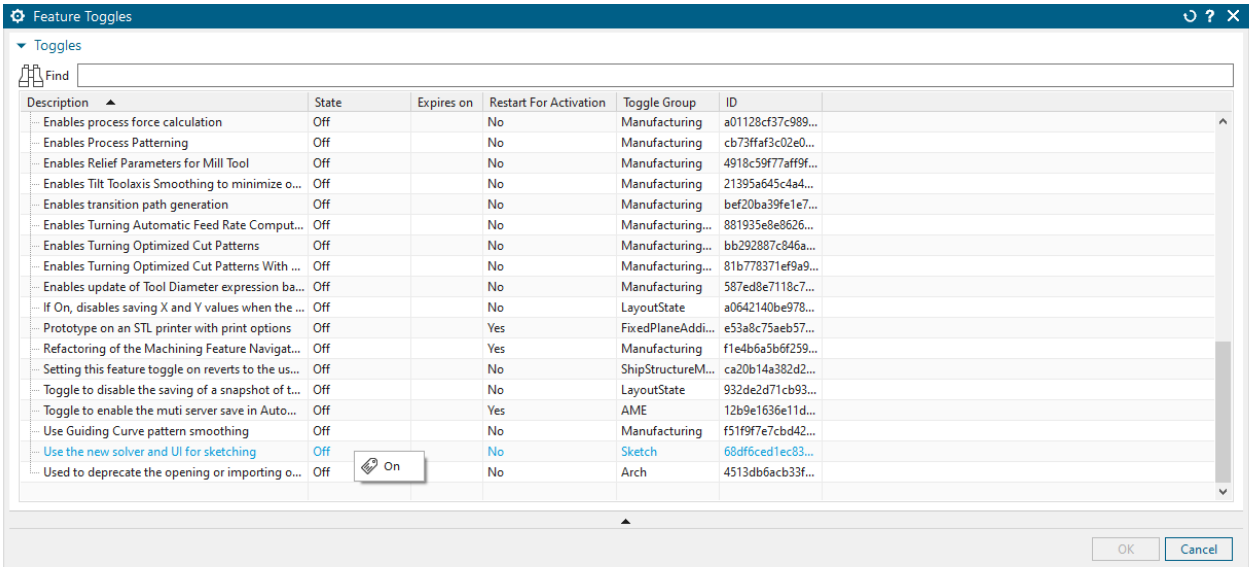


There are three useful options next to the *Finish Flag*. You can change the name of the sketch in the box. The next one is *Orient to Sketch* which orients the view to

the plane of the sketch. If the model file is rotated during the process of sketching, click on this icon to view the sketch on a plane parallel to the screen *Reattach* attaches the sketch to a different planar face, datum plane, or path, or changes the sketch orientation. It allows you to reattach the sketch to the desired plane without recreating all the curves, dimensions, and constraints.

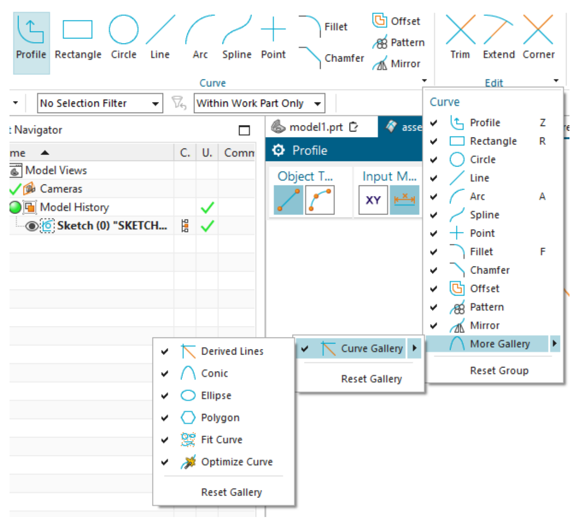
In the latest version of NX, if you want to open the sketch parts that you created from older version NX, **turn on** *"Use the new solver and UI for sketching"* in **File → Utilities → Feature Toggles** to edit parameters of the sketch without renewing the sketch. However, if you want to modify the design of the sketch, the sketch will need to be renewed.





Description	State	Expires on	Restart For Activation	Toggle Group	ID
Enables process force calculation	Off		No	Manufacturing	a01128cf37c989...
Enables Process Patterning	Off		No	Manufacturing	cb73ffaf3c02e0...
Enables Relief Parameters for Mill Tool	Off		No	Manufacturing	4918c59f77aff9f...
Enables Tilt Toolaxis Smoothing to minimize o...	Off		No	Manufacturing	21395a645c4a4...
Enables transition path generation	Off		No	Manufacturing	bef20ba39fe1e7...
Enables Turning Automatic Feed Rate Comput...	Off		No	Manufacturing...	881935e8e8626...
Enables Turning Optimized Cut Patterns	Off		No	Manufacturing...	bb292887c846a...
Enables Turning Optimized Cut Patterns With ...	Off		No	Manufacturing...	81b778371ef9a9...
Enables update of Tool Diameter expression ba...	Off		No	Manufacturing	587ed8e7118c7...
If On, disables saving X and Y values when the ...	Off		No	LayoutState	a0642140be978...
Prototype on an STL printer with print options	Off		Yes	FixedPlaneAddi...	e53a8c75aeb57...
Refactoring of the Machining Feature Navigat...	Off		Yes	Manufacturing	f1e4b6a5b6f259...
Setting this feature toggle on reverts to the us...	Off		No	ShipStructureM...	ca20b14a382d2...
Toggle to disable the saving of a snapshot of t...	Off		No	LayoutState	932de2d71cb93...
Toggle to enable the muti server save in Auto...	Off		Yes	AME	12b9e1636e11d...
Use Guiding Curve pattern smoothing	Off		No	Manufacturing	f51f9f7e7cbd42...
Use the new solver and UI for sketching	Off		No	Sketch	68df6ced1ec83...
Used to deprecate the opening or importing o...	Off		No	Arch	4513db6acb33f...

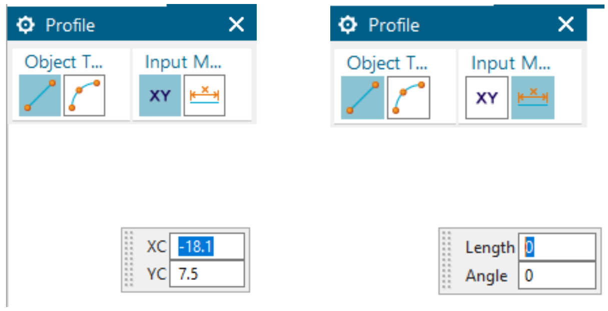
3.3 SKETCH CURVE TOOLBAR

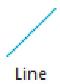


This toolbar contains icons for creating the common types of curves and spline curves, editing, extending, trimming, filleting etc. Each type of curve has different methods of selection and methods of creation. Let us discuss the most frequently used options.

Profile

This option creates both straight lines as well as arcs depending on the icon you select in the pop-up toolbar. You can pick the points by using the coordinate system or by entering the length and angle of the line as shown in the following figures.



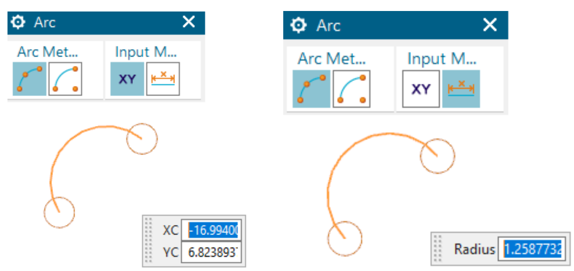


Line

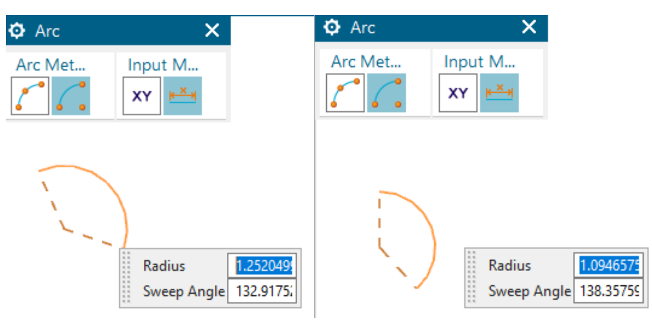
This option will selectively create only straight lines.

Arc

This option creates arcs by either of two methods. The first option creates arc with three sequential points as shown below.



The second option creates the arc with a center point, radius and sweep angle or by center point with a start point and end point. The illustration is shown below.



Circle

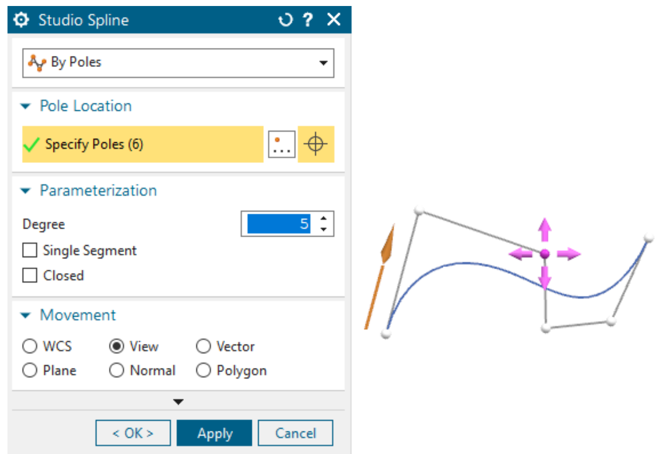
Creating a circle is similar to creating an arc, except that circle is closed.

Quick Trim

This trims the extending curves from the points of intersection of the curves. This option reads every entity by splitting them if they are intersected by another entity and erases the portion selected.

Studio Spline

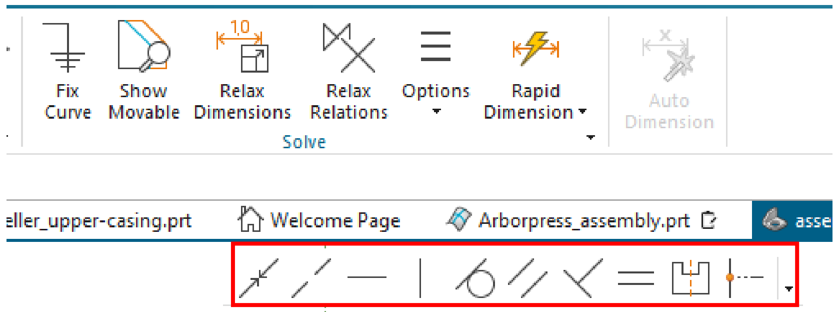
You can create basic spline curves (B-spline and Bezier) with poles or through points with the desired degree of the curve. The spline will be discussed in detail in the seventh chapter (Freeform Features).



3.4 CONSTRAINTS TOOLBAR

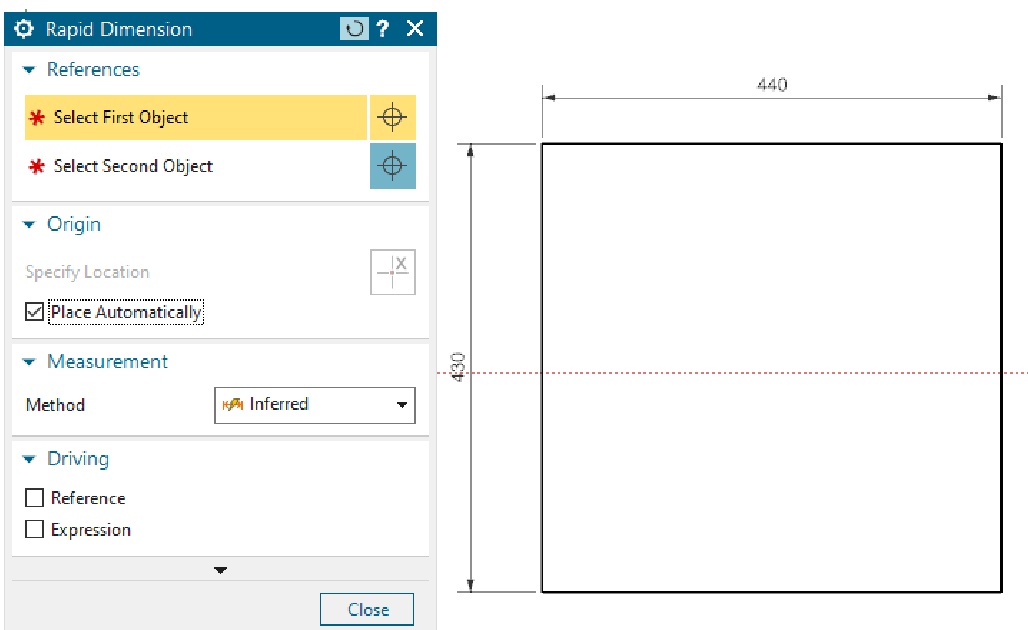
All the curves are created by picking points. For example, a straight line is created with two points. In a 2D environment, any point has two degrees of freedom, one along X and another along Y axis. The number of points depends on the type of curve being created. Therefore, a curve entity has twice the number of degrees of freedom than the number of points it comprises. These degrees of freedom can be removed by creating a constraint with a fixed entity. In fact, it is recommended that you remove all these degrees of freedom (making the sketch *Fully Constrained*) by relating the entities directly or indirectly to the fixed entities. It can be done by giving dimensional or geometric properties like *Parallelism*, *Perpendicularity*, etc.

In NX smart constraints are applied automatically, i.e. automatic dimensions or geometrical constraints are interpreted by NX. You can see this option when you enter the draft feature. The following paragraphs show how to manually apply constraints.



Dimensional Constraints

The degrees of freedom can be eliminated by giving dimensions with fixed entities like axes, planes, the coordinate system or any existing solid geometries created in the model. These dimensions can be linear, radial, angular etc. You can edit the dimensional values at any time during sketching by double-clicking on the dimension.

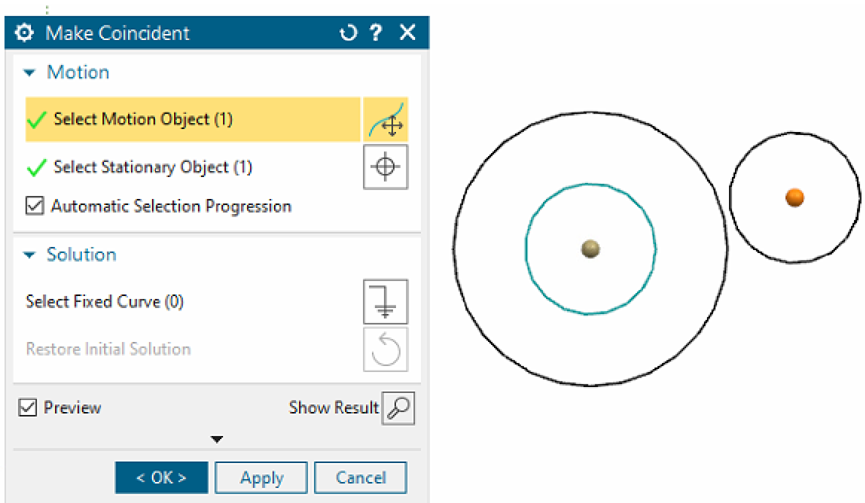


Geometric Constraints

Besides the dimensional constraints, some geometric constraints can be given to eliminate the degrees of freedom. They include coincident, collinear, horizontal, vertical, tangent, parallel, perpendicular, equal, symmetric and mid-point aligned. You could freely apply constraints on lines and curves depends on your needs.

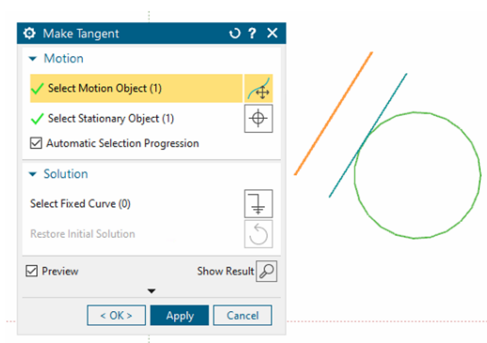
Coincident:

A coincident constraint is applied on two circles in the below picture to aligned small circle to the large circle.



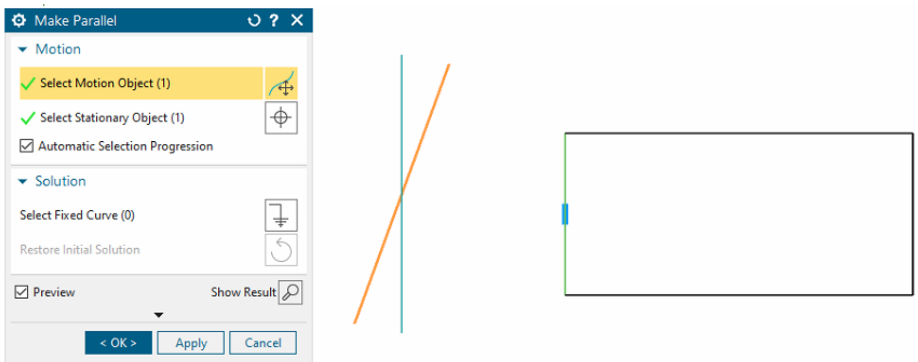
Tangent:

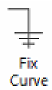
A tangent constraint is applied on the line (orange) and circle (green) to make the line is tangent to the circle.



Parallel:

A parallel constraint is applied on the line in the below picture to be parallel to the left side of the rectangle (the line was originally at an angle with the rectangle).



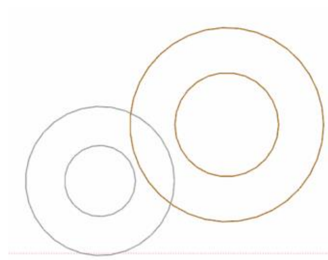


Fix Curve Constraints

You can apply **Fix Curve constraints** on curves by clicking this icon to fix curves.

Show Movable

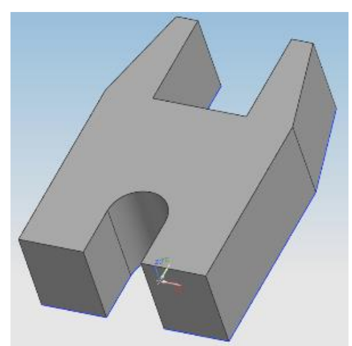
Show you the curves those are fixed (grey color) and movable (brown color).



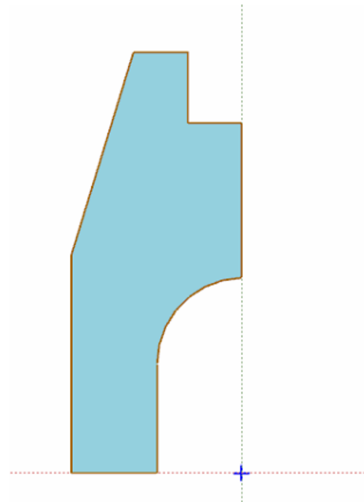
3.5 EXAMPLES

3.5.1 Arbor Press Base

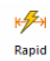
An arbor press base will be modeled in this example using NX Sketch as shown in the right figure.

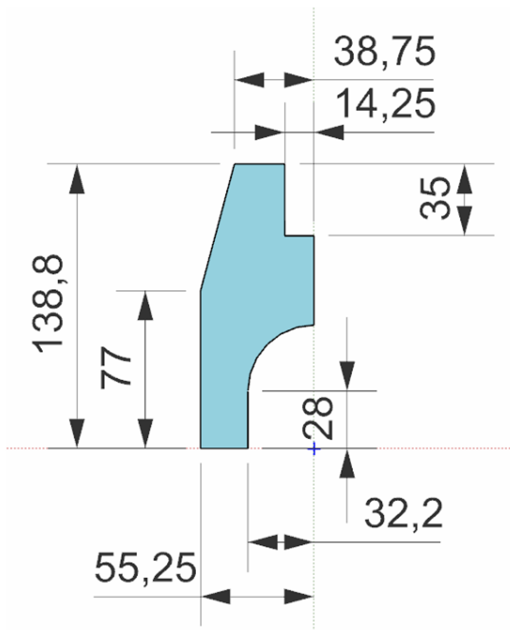


- Create a new file and save it as **Arborpress_base.prt**
- Click on the **Sketch** button and click **OK** (default X-Y plane)
- Draw a figure similar to the one shown on right. While making continuous sketch, click on the **Line** icon on the **Profile** dialog box to create straight lines and the **Arc** icon to make the semicircle. (Look at the size of the XY plane in the figure. Use that perspective for the approximate zooming). For those parts are symmetrical, you only need to create half of part and use the mirror feature to create the whole part.

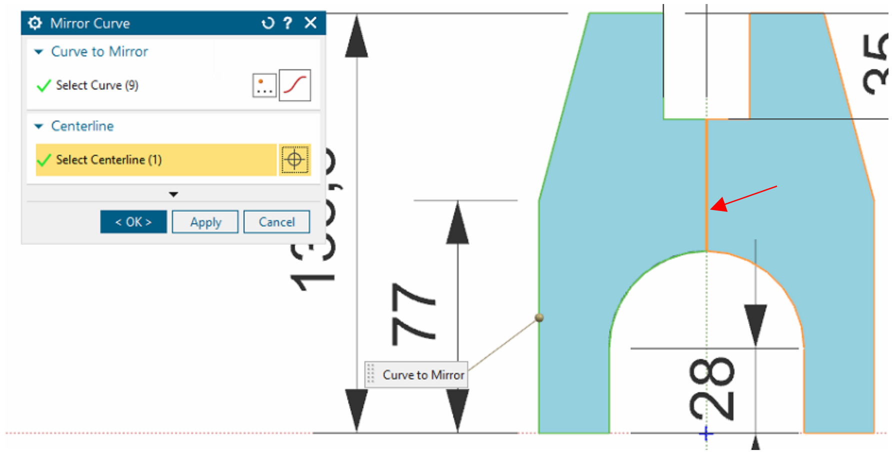


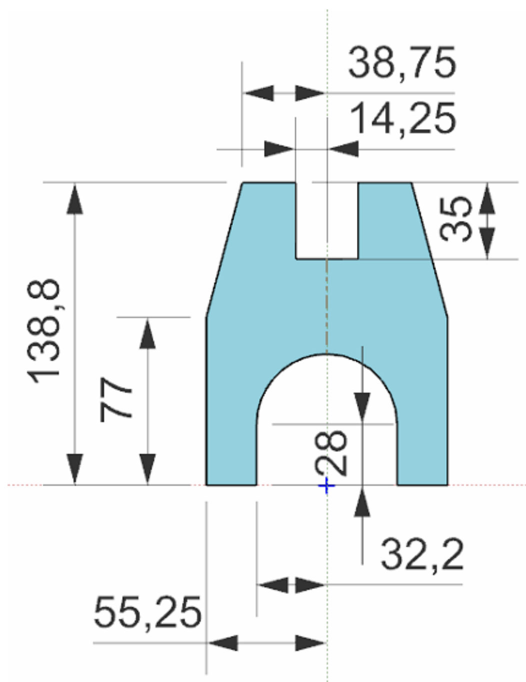
Now we have to create the *Dimensional* constraints.

- Choose the **Rapid Dimension** icon  in the **Solve** toolbar (right click) to add all the dimensions as shown in the following figure and specify the values by double-click left mouse bottom dimensions to change dimensions



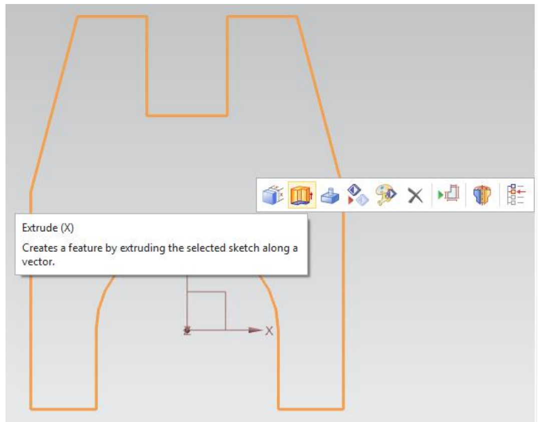
Now, we have finished a half of the part. The mirror feature will help create the other half efficiently. Click **Mirror** in the **Curve Box** → select the curves you draw → select Centerline (indicated by red arrow) to create the whole part.



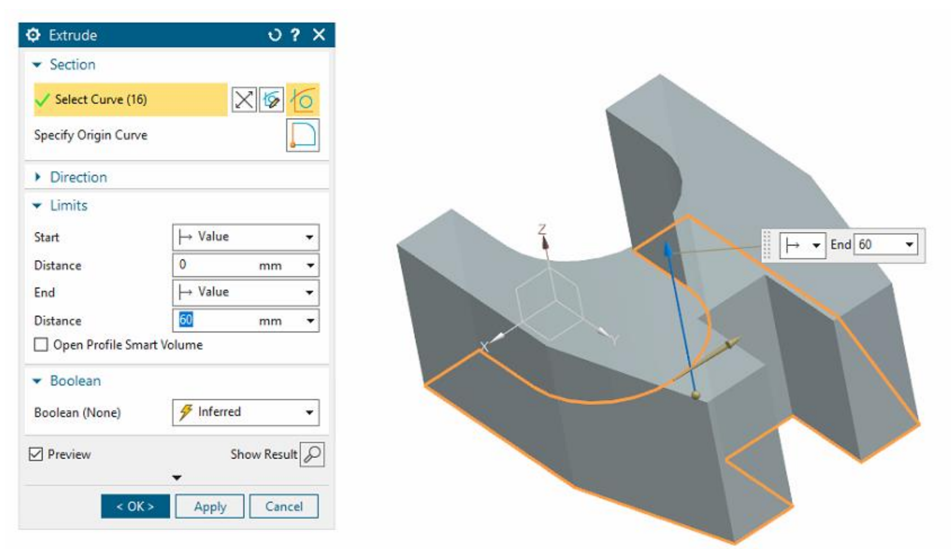


➢ Click on the **Finish Flag**  on the top left corner or bottom right of the screen when you are finished

➢ Click on the sketch and select **Extrude** (this **Feature** is explained in detail in the next sections)



➢ Extrude this sketch in the Z-direction by **60** mm



➢ Save and close the file

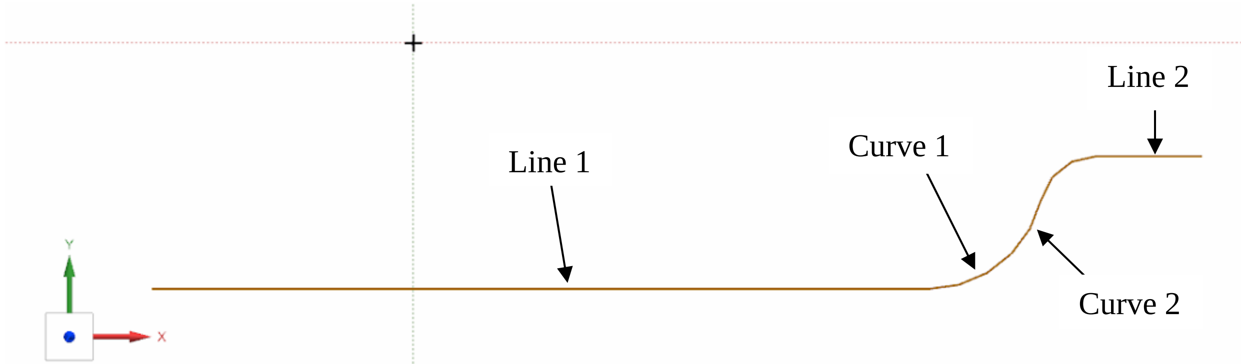
3.5.2 Impeller Lower Casing

➢ Create a new file in **inches** and save it as **Impeller_lower_casing.prt**

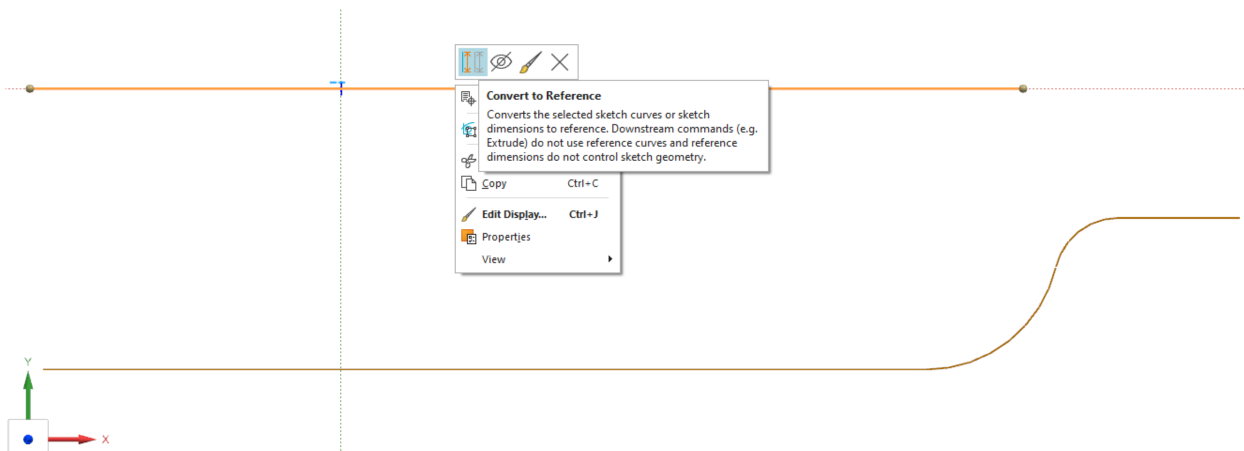
➢ Click **Menu** → **Insert** → **Sketch** or click **Sketch**  icon from the ribbon bar

➢ Set the sketching plane as the **XC-YC** plane and click **OK**

➢ Make sure the **Profile** window is showing and draw the following curve

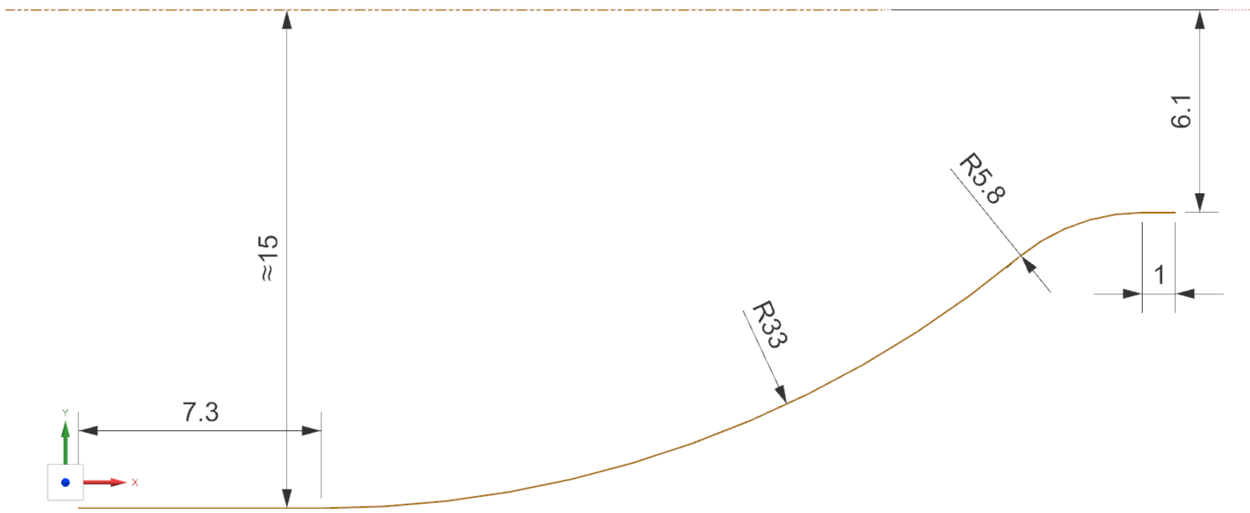


- Create a reference line at the X-axis: clicking the **line** in the **Curve** group and create a line aligned with X-axis. Then, click right-mouse bottom on the line to convert the solid line to reference line.

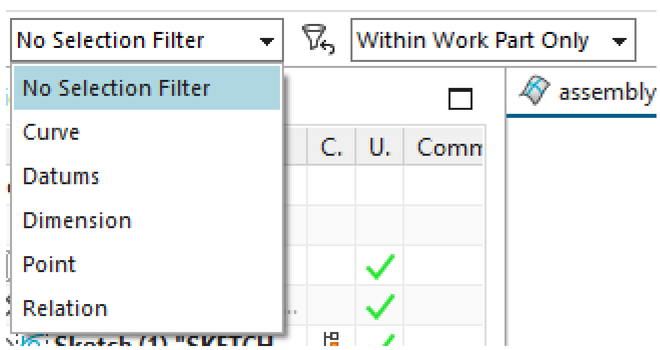


Next, we will constrain the curve

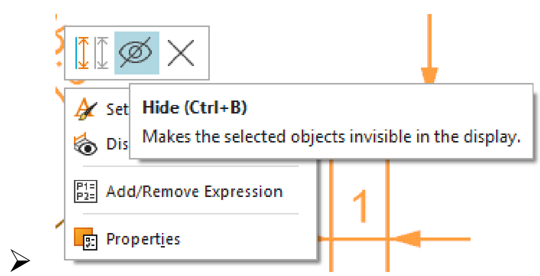
- Use **Tangent** constraint  to make all of the curve-lines and curve-curve joints Tangent
- Use **Rapid Dimension** to apply the dimensional constraints and change dimensions by double clicking left-mouse bottom, as shown in the figure below



- Quickly select all the dimensions: After change to **Dimension** in **No Selection Filter,** select all dimensions.

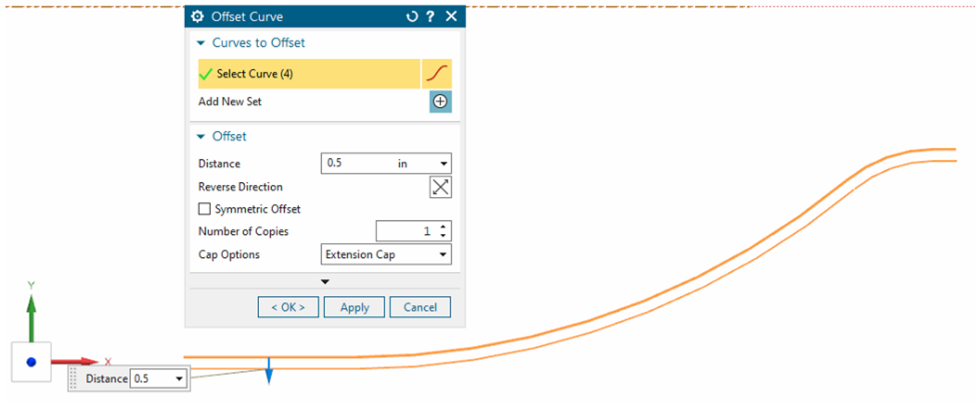


- Right click on any dimensions and **Hide** the dimensions

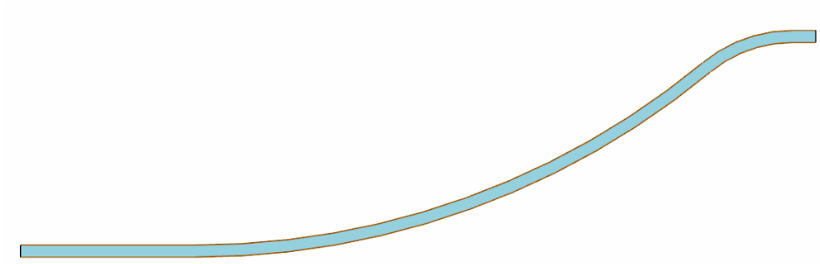


- Choose **Offset** in **Curve Group** from the ribbon bar
- Select all the curves. You should see 4 objects being selected in **Select Object**
- Enter the **Distance** to be **0.5** inch

➢ Click **OK**



➢ Then join the end-points at the two ends using the basic curves (**line**) to complete the sketch



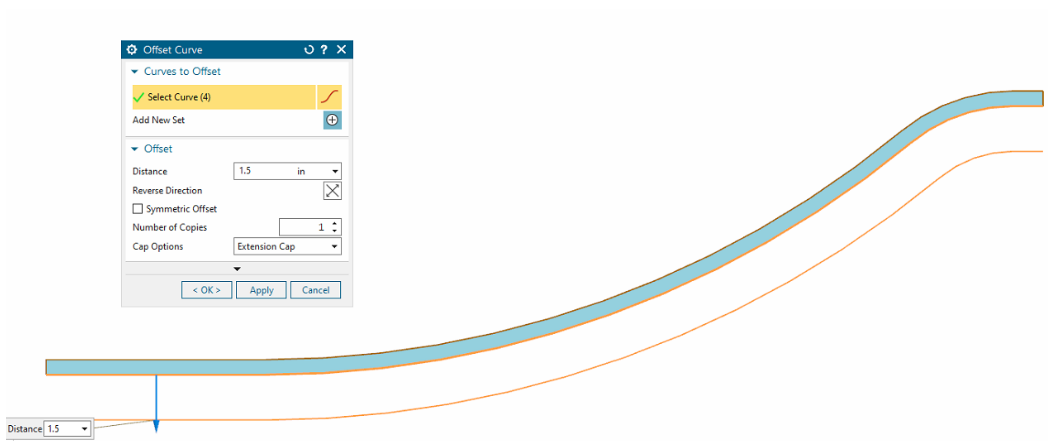
The sketch is ready.

➢ Select 4 curves on bottom side and Choose **Offset** in **Curve Group** from the ribbon bar

➢ You should see 4 objects being selected in **Select Object**

➢ Enter the **Distance** to be **1.5** inch

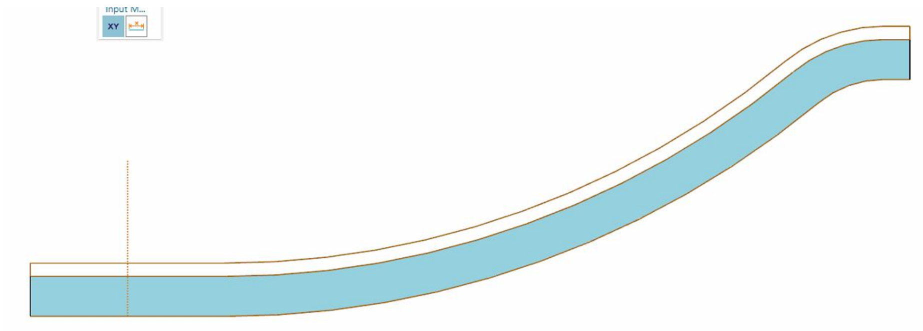
➢ Click **OK**



This will form a curve outside the casing.

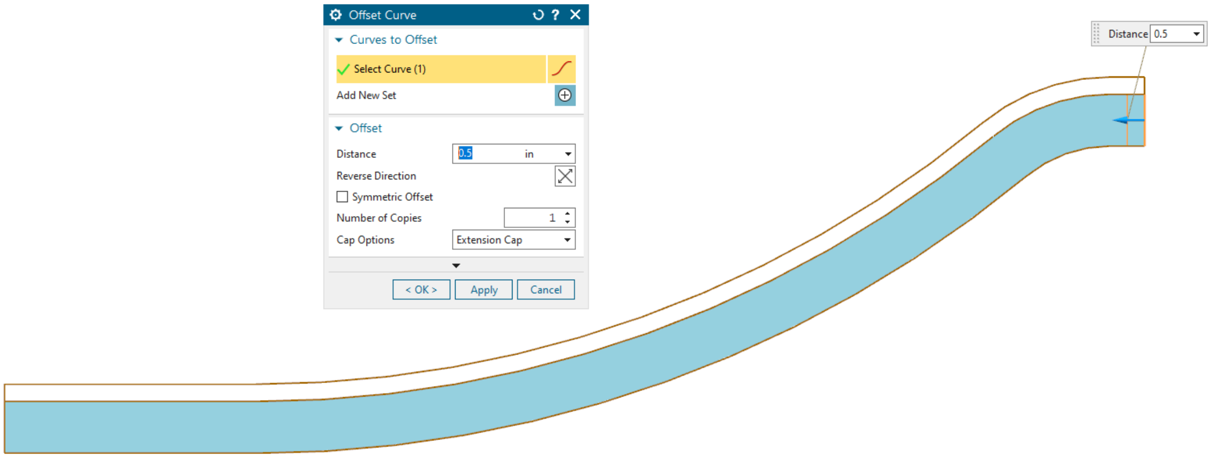
- ➢ Using **straight lines** join this curve with the inside curve of the casing

It will form a closed chain curve as shown.

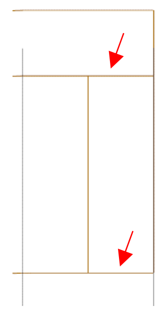


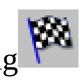
Now we will create the curve required for outside of the casing on the smaller side which will form the flange portion.

- ➢ Choose **Edit → Move Object**
- ➢ Select the outer line as shown in the figure below
- ➢ Select the outer line as shown in the figure below and Choose **Offset** in **Curve Group** from the ribbon bar
- ➢ You should see 1 object being selected in **Select Object**
- ➢ Enter the **Distance** to be **0.5** inch
- ➢ Click **OK**



- ➢ **Move** the lower curve in the **XC**-direction by **-0.5** inches. This is the same as translating it in the **negative XC-direction** by **0.5** inches
- ➢ Using **straight lines** join the two lines (red arrows) as shown in the figure on right side

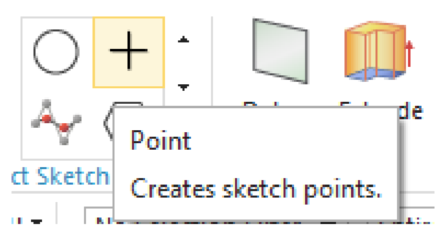


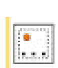
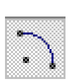
- ➢ Click on the Finish Flag 
- ➢ Save and Close the file

We will use this sketching in the next chapter to model the Impeller Lower Casing.

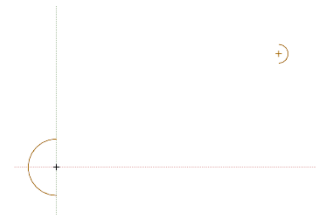
3.5.3 Impeller

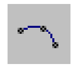
- ➢ Create a new file in **inches** and save it as **Impeller_impeller.prt**
- ➢ Click on **Sketch** to create the profile of the Impeller
- ➢ Set the sketching plane as the XC-YC plane and click **OK**
- ➢ Click on **Menu** → **Insert** → **Point** or click **Point** from **Curve** group in the ribbon bar

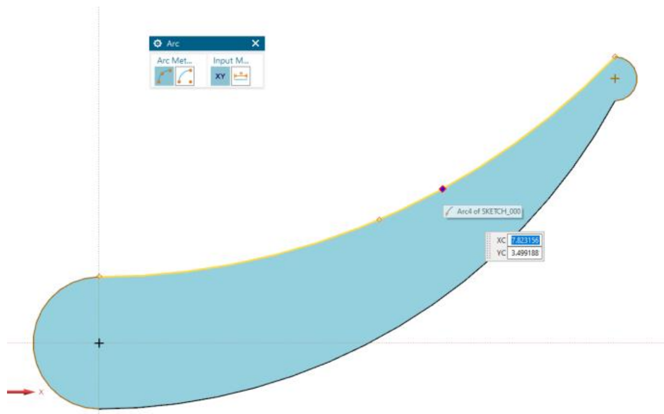


- ➢ Use point dialog  to create two **Points**, one at the origin **(0, 0, 0)** and one at **(11.75, 6, 0)**
- ➢ Click on the **Arc** icon on the side toolbar and click on the **Arc by Center and Endpoints** icon  in the pop-up toolbar

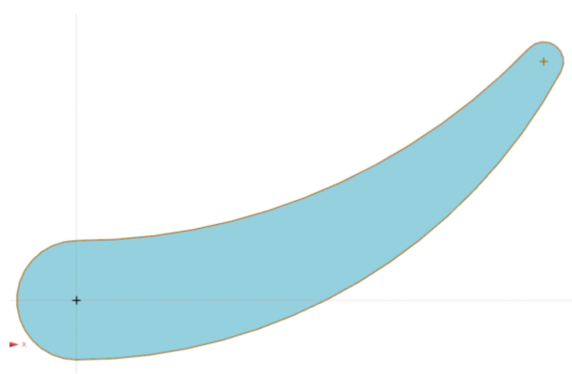
- Click on the point at the origin and create an arc with a radius of 1.5 and a sweep angle 180 degrees similar to the one shown in the figure below
- Click on the point at (11.75, 6, 0) and create an arc with a radius of **0.5**



- Click on the **Arc by 3 Points** icon  in the pop-up toolbar
- Select the top endpoints of the two arcs you just created and click somewhere in between to create another arc that connects them. Do the same for the bottom endpoints

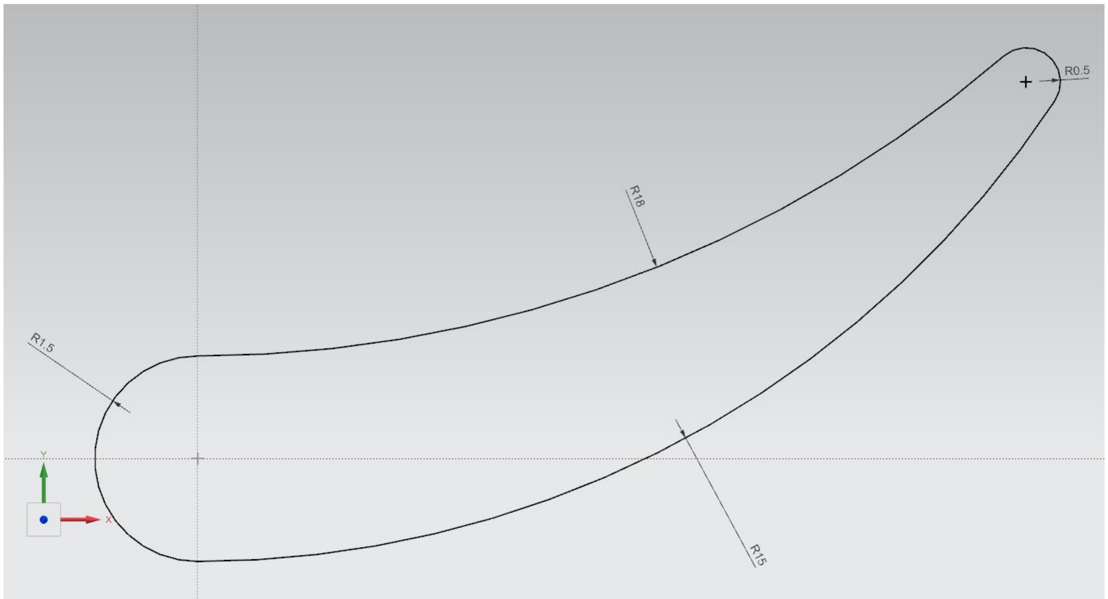


- Click on the **Tangent** Constraints icon  in the Ribbon toolbar and make sure that all the arcs are **Tangent** to one another at their endpoints



- Click on the point at the origin and click on the **Fix Curve** icon
- Then click on the **Rapid Dimension** icon

- Give the **Radius** dimensions for each arc. Edit dimensions so that the two arcs on the end are 1.5 and 0.5 inches and the two middle arcs are 18 and 15 inches as shown in the figure below



- Click on the **Finish Flag** 

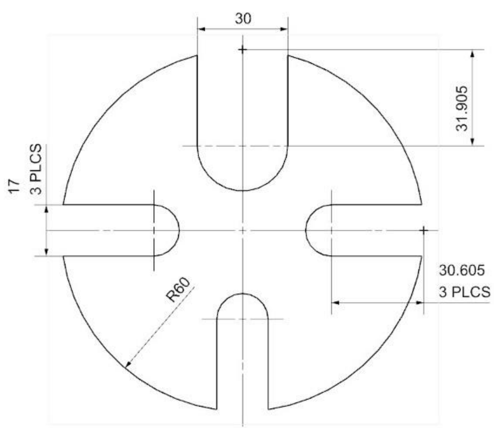
- Save and Close the file.

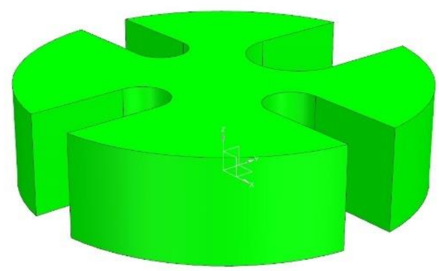
We will use this sketching in the next chapter to model the Impeller.

3.6 EXERCISES

3.6.1 Circular Base

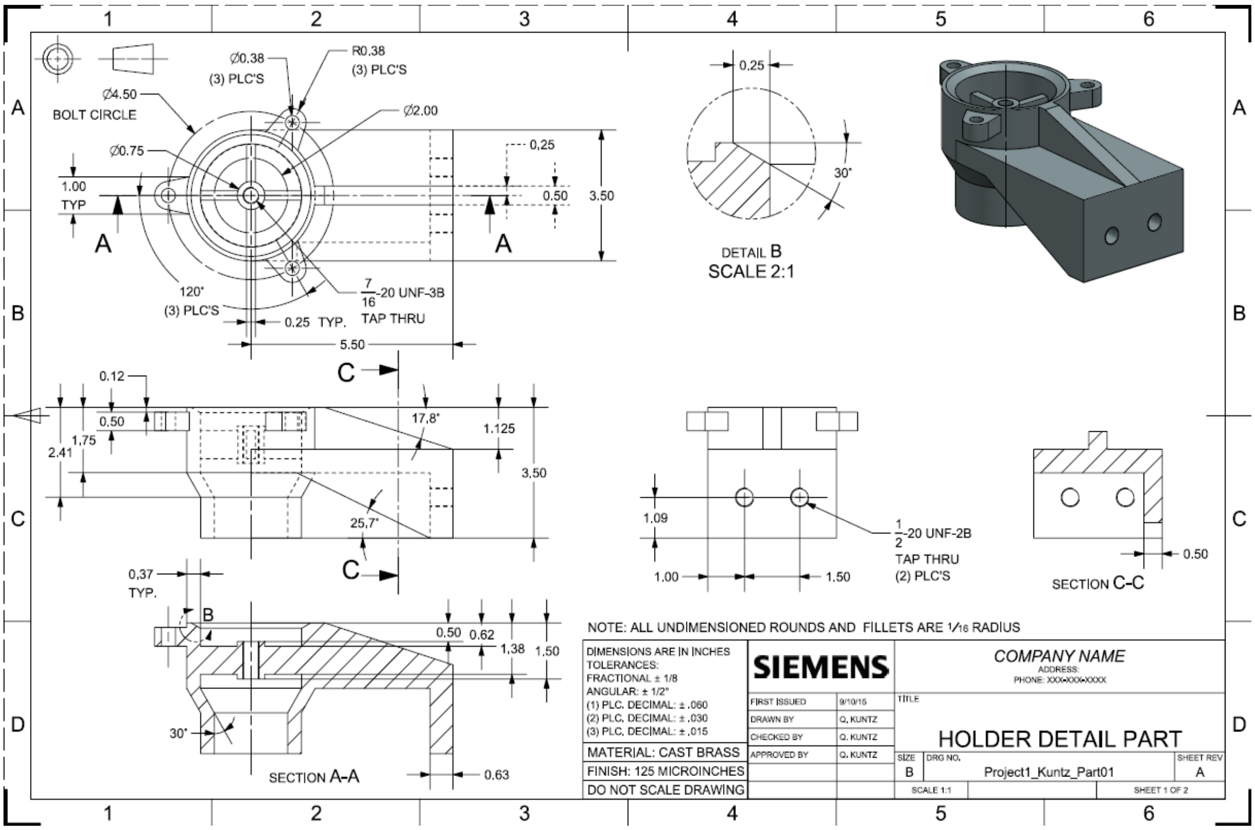
One of models that we will use in several chapters of this manual for various purposes is a simplified arbor press having more than ten components. We modeled one of these components in example 3.5.1 of this chapter. Another component is a circular base used to hold a workpiece under the press. Use the dimensions given below to model the circular base (dimension in millimeters). Note: thickness = 30. (review the Sketch of Circular Base)





3.6.2 Sketching of a Holder

In one of the exercises in chapter 4, you will be asked to model a holder. Multiple 2D sketches will be required to model the 3D holder. As a starting point for that exercise, model the 2D sketching shown in the figure below and save the file (all dimensions are in inches).



CHAPTER 4 – THREE DIMENSIONAL MODELING

This chapter discusses the basics of three-dimensional (3D) modeling in NX. We will discuss what a feature is, what the different types of features are, what primitives are, and how to model features in NX using primitives. This will give a head start to the modeling portion of NX and develop an understanding of the use of *Form Features* for 3D modeling. Once these features are introduced, we will focus on *Feature Operations* which are functions that can be applied to the faces and edges of a solid body or features you have created. These include taper, edge blend, face blend, chamfer, trim, etc. After explaining the feature operations, the chapter will walk you through some examples.

In NX, *Features* are a class of objects that have a defined parent. Features are associatively defined by one or more parents and the order of their creation and modification retain within the model, thus capturing it through the *History*. Parents can be geometrical objects or numerical variables. Features include primitives, surfaces and/or solids and certain wire frame objects (such as curves and associative trim and bridge curves). For example, some common features include blocks, cylinders, cones, spheres, extruded bodies, and revolved bodies.

Commonly *Features* can be classified as follows:

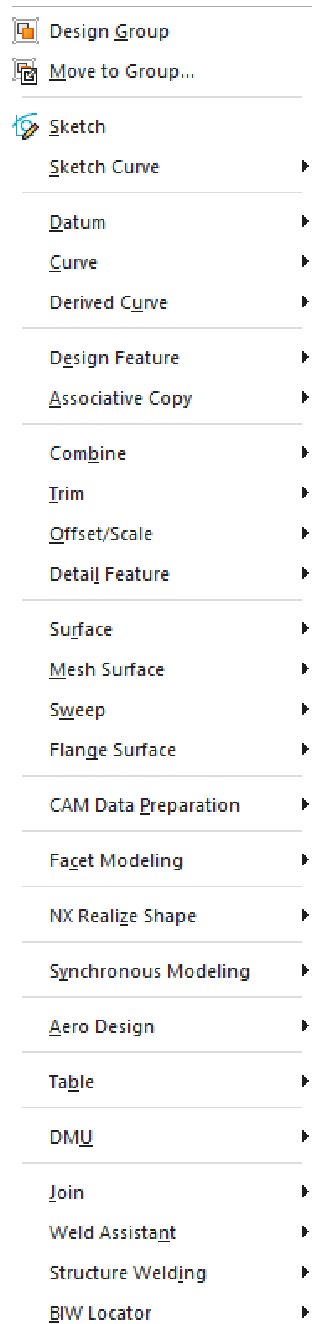
- **Body**: A class of objects containing solids and sheets
- **Solid Body**: A collection of faces and edges that enclose a volume
- **Sheet Body**: A collection of one or more faces that do not enclose a volume
- **Face**: A region on the outside of a body enclosed by edges

4.1 TYPES OF FEATURES

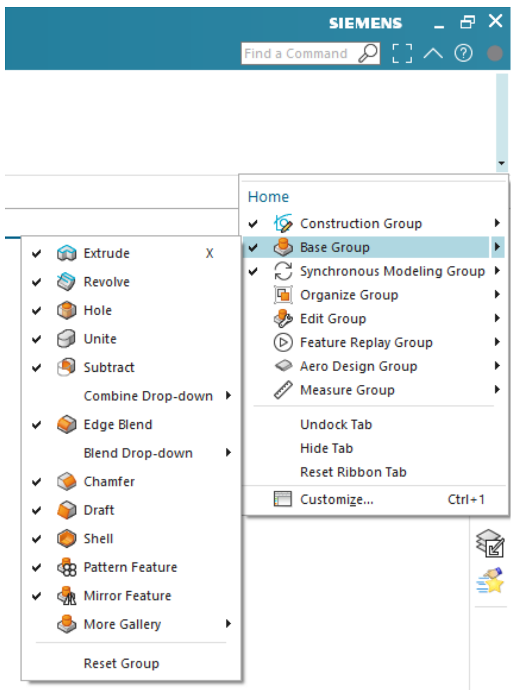
There are six types of *Form Features*: Primitives, Reference features, Swept features, Remove features, Extract features, and User-defined features. NX stores all the *Form Features* under the *Insert* menu option. The form features are also available in the *Form Features Toolbar*.

➢ Click **Insert** on the **Menu**

As you can see, the marked menus in the figure on the right side contain the commands of *Form Features*. The *Form Feature* icons are grouped in the *Home Toolbar* as shown below. You can choose the icons that you use frequently.



- ➢ Click on the drop down arrow at the bottom right of **Ribbon Toolbar**
- ➢ Choose **Base Group**



4.1.1 Primitives

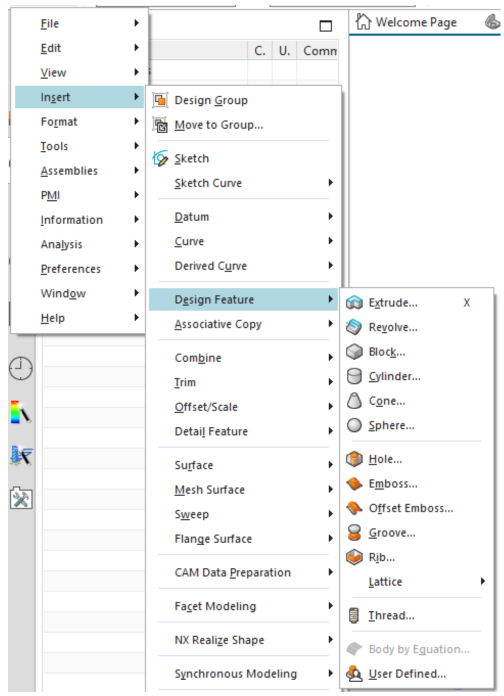
NX enable you create solid bodies in the form of generic building shapes. Primitives include:

- Block
- Cylinder
- Cone
- Sphere

Primitives are the primary entities. Hence, we will begin with a short description of primitives and then proceed to modeling various objects.

➢ To create primitives’ geometries:

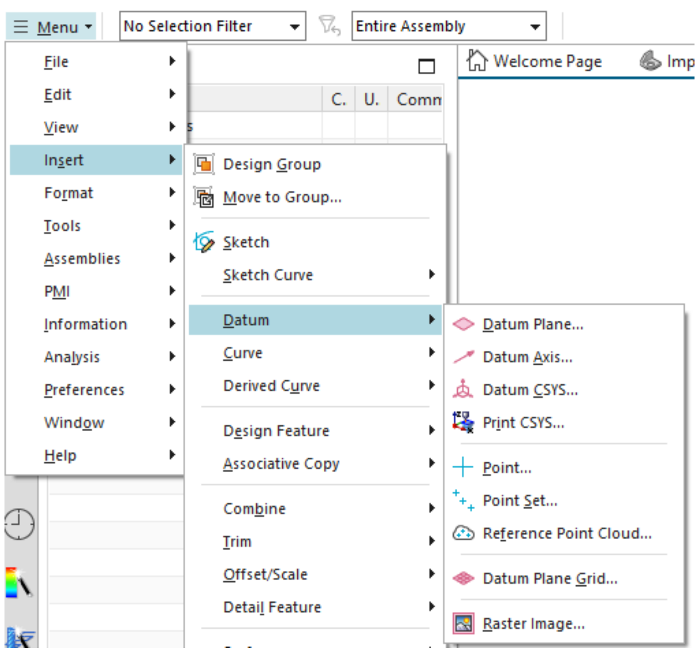
Menu → Insert → Design Feature as shown the right figure.

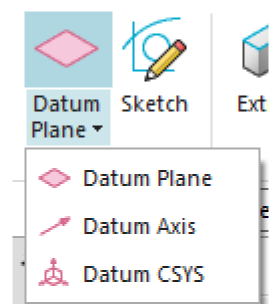


4.1.2 Reference Features

These features enable you to create reference points, reference axes, reference plane, or reference coordinates, which can be used for constructing other features.

- ➢ Click on **Menu → Insert → Datum** or click on **Datum Plane** in **Construction** group in the ribbon bar to view the different **Reference Feature** options: **Datum Plane**, **Datum Axis**, and **Datum CSYS**



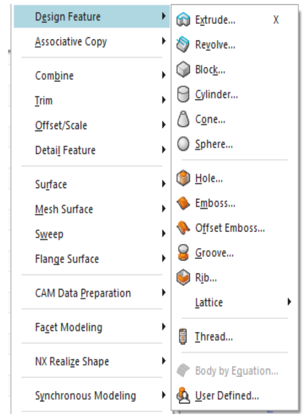


4.1.3 Swept Features

These NX features enable you to create bodies by extruding or revolving sections. Swept Features include:

- Extruded Body

- Revolved Body
- Sweep along Guide
- Tube
- Styled Sweep

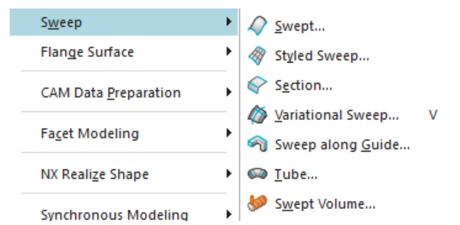


To select a swept feature, you can do the following:

➢ Click on **Insert** → **Design Feature** for **Extrude** and **Revolve** or click on **Extrude** in **Base** group in the ribbon bar

OR

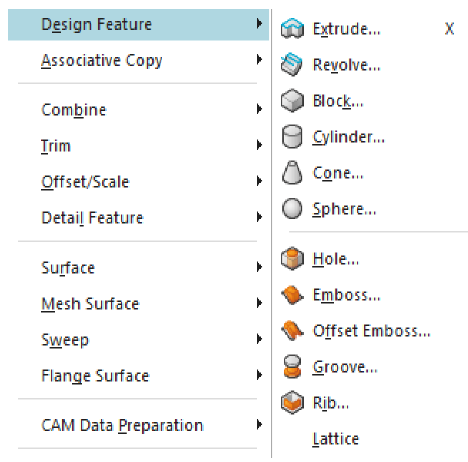
➢ Click on **Insert** → **Sweep** or click on **Surface** in **Base** group in the ribbon bar to find all the options available including **Sweep**



4.1.4 Remove Features

Remove Features let you create tool bodies and use them to remove solid shapes from 3D geometry.

➢ Click on **Insert** → **Design Feature**



Remove Features include:

- Hole
- Emboss
- Groove
- Rib
- Lattice

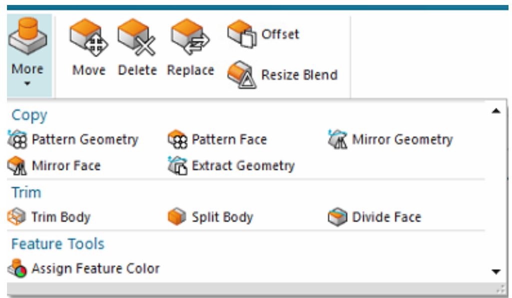
4.1.5 Extract Features

These features let you create geometries / bodies by extracting curves, faces and regions. These features are widely spaced under *Associative Copy* and *Offset/Scale* menus. Extract Features include:

- Extract
- Solid to Shell
- Thicken Sheet
- Bounded plane

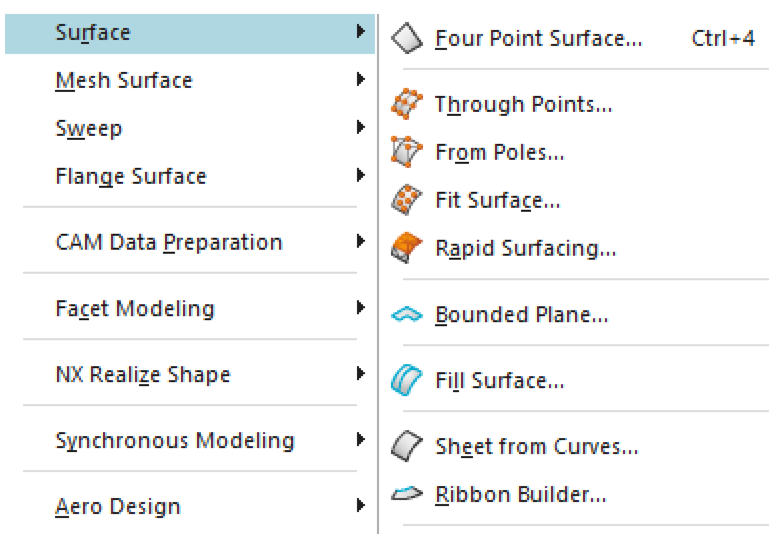
- Sheet from curves

Click on **Insert** → **Associative Copy** → **Extract** for **Extract** options or click on **More** in **Base** group in the ribbon bar of **Home** to find **Extract Geometry**



➢ Click on **Insert** → **Offset/Scale** for **Solid** to **Shell** and **Thicken Sheet** Assistant or click on **Shell** in **Base** group in the ribbon bar of **Home** or **Thicken** in **Base** group in the ribbon bar of **Surface** to find **Offset/Scale** options

➢ Click on **Insert** → **Surface** for **Bounded Plane** and **Sheet from curves**



4.1.6 User-Defined features

These custom build features allow you to create your own sets of formed features to automate commonly used design elements. You can use user-defined features to extend the range and power of the built-in form features.

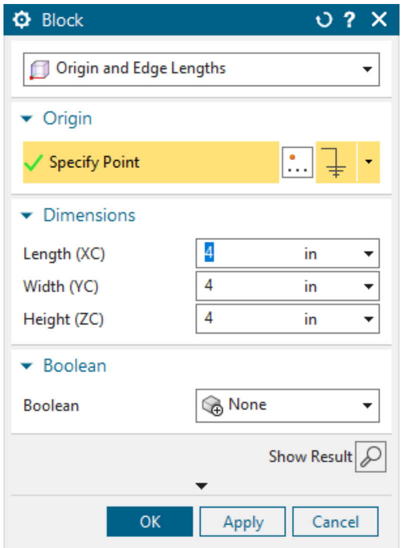
- Click on **Insert → Design Feature → User Defined**

4.2 PRIMITIVES

Primitive features are base features from which many other features can be created. The basic primitives are blocks, cylinders, cones and spheres. Primitives are non-associative which means they are not associated to the geometry used to create them.

Note that usually *Swept Features* are used to create *Primitives* instead of the commands mentioned here.

4.2.1 Model a Block



- Create a new file and name it as **Arborpress_plate.prt**
- Choose **Insert → Design Feature → Block** or click on the **Block** icon in the **Form Feature Toolbar**

The *Block* window appears. There are three main things to define a block, which are the *Type*, *Origin* and the *Dimensions* of the block. To access the *Types*, scroll the drop-down menu under *Type*. There are three ways to create a block primitive:

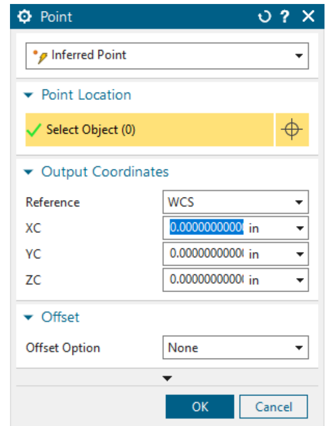
- Origin and Edge Lengths
- Height and Two Points
- Two Diagonal Points

- Make sure the **Origin and Edge Lengths** method is selected

Now, we will choose the origin using the *Point Constructor*:

- Click on the **Point Dialog** icon under the **Origin**

The *Point Constructor* dialog box will open. The XC, YC, ZC points should have a default value of 0.



- ➢ Click **OK**

The *Block* window will reappear again.

- ➢ Type the following dimensions in the window

  Length (XC) = **65** inches

  Width (YC) = **85** inches

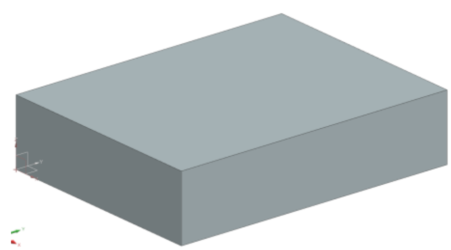
  Height (ZC) = **20** inches

- ➢ Click **OK**

If you do not see anything on the screen,

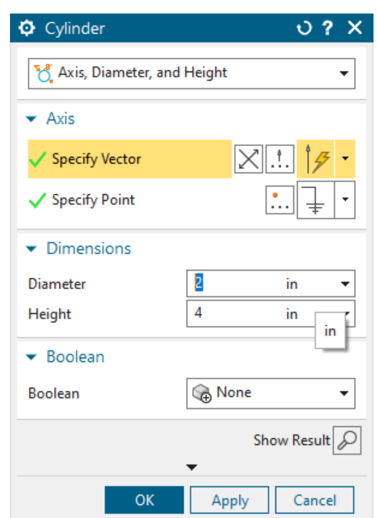
- ➢ Right-click (MB3) and select **FIT**. You can also press **<Ctrl> + F**
- ➢ Right-click on the screen and click on **Orient View** →**Trimetric**

You should be able to see the complete plate solid model. Save and close the part file.



4.2.2 Model a Shaft

Let's move on to model a shaft having two cylinders and one cone joined together using Primitives.



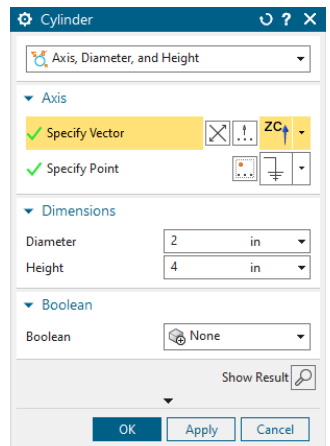
- ➢ Create a new file and save it as **Impeller_shaft.prt**

➢ Choose **Insert** → **Design Feature** → **Cylinder** or click on **More** in **Base** group in the ribbon bar of **Home** to find **Cylinder** in **Design Feature** section

Similar to the *Block*, there are three things that need to be defined to create a cylinder: *Type, Axis & Origin,* and *Dimensions.*

A *Cylinder* can be defined by two *types* which can be obtained by scrolling the drop-down menu under *Type*

- Axis, Diameter, and Height
- Arc and Height



➢ Select **Axis, Diameter, and Height**

➢ Click on the **Vector Constructor** icon next to **Specify Vector** and select the **ZC** Axis icon

➢ Click on the **Point Dialog** icon next to **Specify Point** to set the origin of the cylinder

➢ Set all the **XC**, **YC**, and **ZC** coordinates to be 0

You can see that the selected point is the origin of WCS

➢ In the next dialog box of the window, type in the following values

Diameter = **4** inches

Height = **18** inches

➢ Click **OK**

➢ Right-click on the screen, choose **Orient View** → **Isometric**

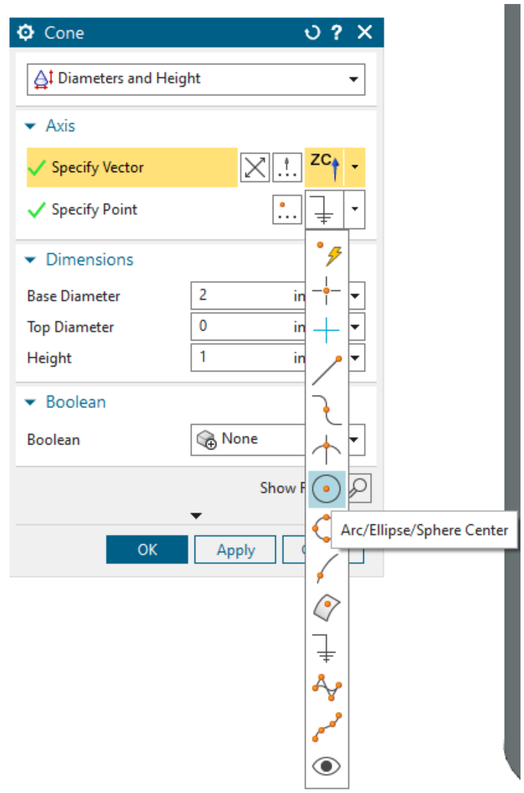
The cylinder will look as shown on the right. Now we will create a cone at one end of the cylinder.

➢ Choose **Insert** → **Design Feature** → **Cone** or click on **More** in **Feature** group in the ribbon bar of **Home** to find **Cone** in **Design Feature** section

Similar to *Block* and *Cylinder*, there are various ways to create a cone which can be seen by scrolling the drop-down menu in the *Type* box.

- Diameters and Height

- Diameters and Half Angle
- Base Diameter, Height, and Half Angle
- Top Diameter, Height, and Half Angle
- Two Coaxial Arcs



➢ Select **Diameters and Height**

➢ Click on the Vector Constructor icon next to **Specify Vector**

➢ Choose the **ZC-Axis** icon so the vector is pointing in the positive Z direction

➢ Click on the **Point Constructor** icon next to **Specify Point** to set the origin of the cylinder.

The *Point Constructor* window will appear next.

➢ Choose the **Arc/Ellipse/Sphere Center** icon on the dialog box and click on the top circular edge of the cylinder

OR

➢ For the **Point Dialog**, type in the following values:
XC = **0**; YC = **0**; ZC = **18**

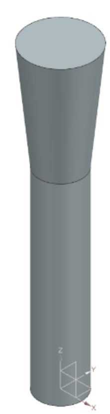
➢ Click **OK**

➢ In the **Cone Window**, type in the following values:

Base Diameter = **4** inches
Top Diameter = **6** inches
Height = **10** inches

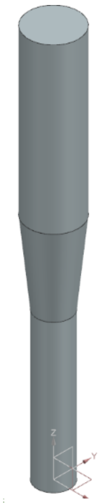
➢ On the Boolean Operation window, choose **Unite** and select the cylinder

➢ Click **OK**



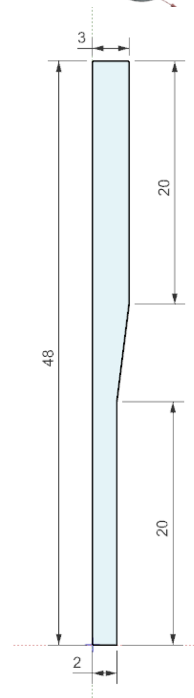
Now the cone will appear on top of the cylinder. The shaft is as shown on right.

Now we will create one more cylinder on top of the cone.



- Repeat the same procedure as before to create another **Cylinder**. The vector should be pointing in the positive *ZC-direction*. On the **Point Constructor** window, again click on the **Center** icon and construct it at the center point of the base of the cone. The cylinder should have a diameter of **6** inches and a height of **20** inches. Unite the cylinder with the old structure.

The complete shaft will look as shown in the right figure. Do not forget to save the model.

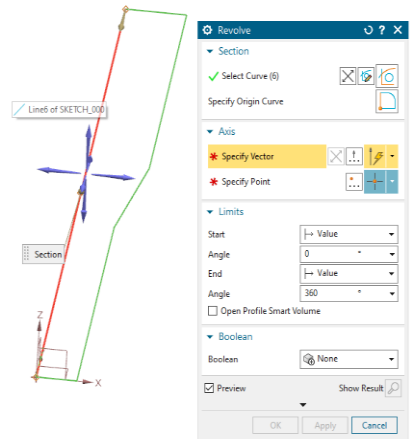


Another option of using the primitives features to model this shaft, it can also be quickly modeled by using **Sketch** and **Revolve** feature, this is more recommended approach to create geometries.

- Click **Sketch** and choose **XC-YC** plane
- Use Line feature to draw the draft as shown in the right (Unit: inch)
- Click **Finish**
- Click **Revolve** in **BASE** feature 

- Select all Curves
- Use Inferred Vector to choose the red line as the revolve axis as shown in the bottom-right figure
- Click **OK**

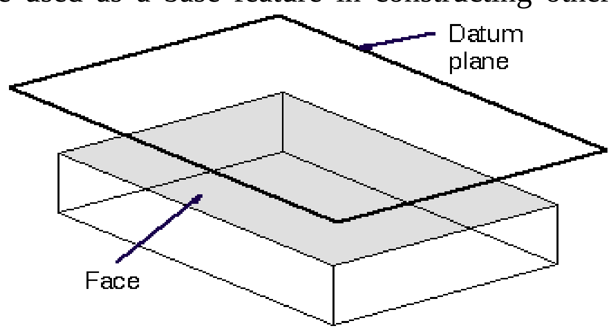
- ➢ The shaft is modeled as showing below
- ➢ Save the model



4.3 REFERENCE FEATURES

4.3.1 Datum Plane

Datum Planes are reference features that can be used as a base feature in constructing other features. They assist in creating features on cylinders, cones, spheres, and revolved solid bodies which do not have a planar surface and also aid in creating features at angles other than normal to the faces of the target solid. We will follow some simple steps to practice *Reference Features*. For starters, we will create a *Datum Plane* that is offset from a base face as shown in the figure above.



- ➢ "Open the model **Arborpress_plate.prt**"
- ➢ Choose **Insert** → **Datum** → **Datum Plane**

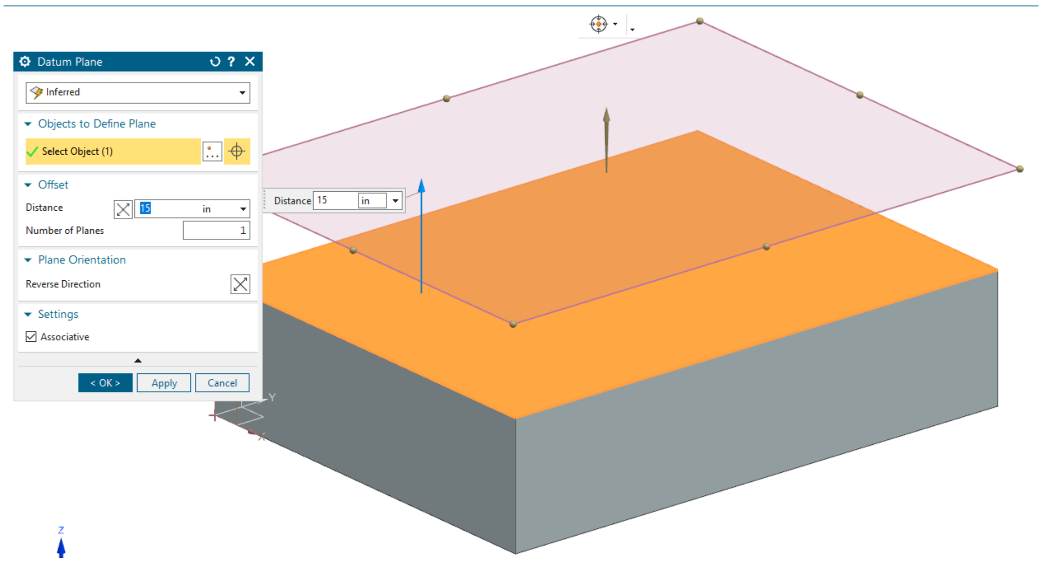
The *Datum Plane* dialog can also be opened by clicking the icon as shown in the figure below from the *Construction Toolbar*.

The *Datum Plane* window allows you to choose the method of selection. However, NX is smart enough to judge the method depending on the entity you select if you keep in *Inferred* option, which is also the *Default* option.

- Click on the top surface of the block so that it becomes highlighted

The vector displays the positive offset direction that the datum plane will be created in. If you had selected the bottom face, the vector would have pointed downward, away from the solid.

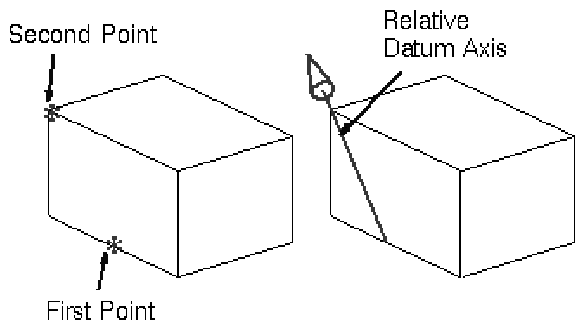
- Set the **Offset Distance** value as **15** inches in the dialog box and click **OK**



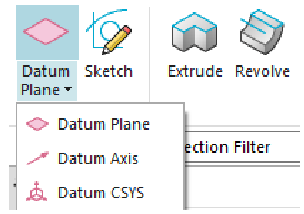
- If you don't see the complete model and plane, right-click and select ***FIT*** (or double click anywhere on the blank screen of design space in NX).

4.3.2 Datum Axis

In this part, you are going to create a *Datum Axis*. A *Datum Axis* is a reference feature that can be used to create *Datum Planes*, *Revolved Features*, *Extruded Bodies*, etc. It can be created either relative to another object or as a fixed axis (i.e., not referencing, and not constrained by other geometric objects).



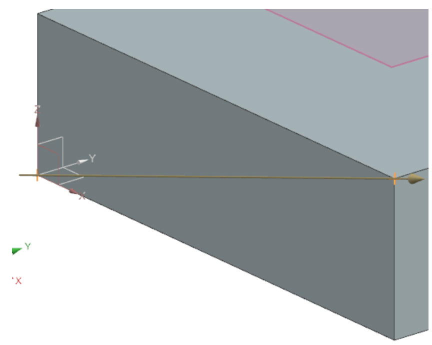
➢ Choose **Insert → Datum → Datum Axis**



The *Datum Axis* dialog can also be opened by clicking the icon as shown in the figure on the right from the *Construction* toolbar.

The next window allows you to choose the method of selecting the axis. Note that NX can judge which method to use depending on the entity you select.

There are various ways to make a *Datum Axis*. They include *Point and Direction*, *Two Points*, *Two Planes*, etc.



➢ Select the two points on the block as shown in the figure on the right

➢ Click **OK**

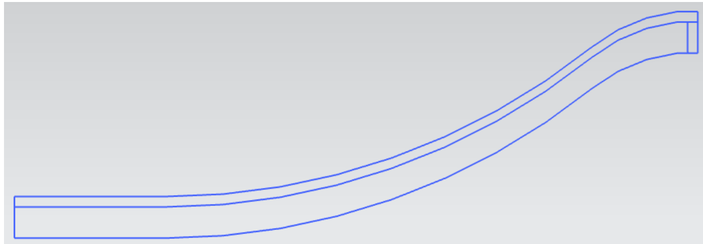
The *Datum Axis* will be a diagonal as shown.

4.4 DESIGN FEATURES

Two important *Design Features* (*Extrude* and *Revolve*) are discussed here using a practical example which is the continuation of the lower casing of the impeller which we started in the previous chapter.

➢ Open the **Impeller_lower_casing.prt** (created in Chapter 3.5.2)

In the previous section, we finished the two dimensional sketching of this part and it should look similar to the below figure.



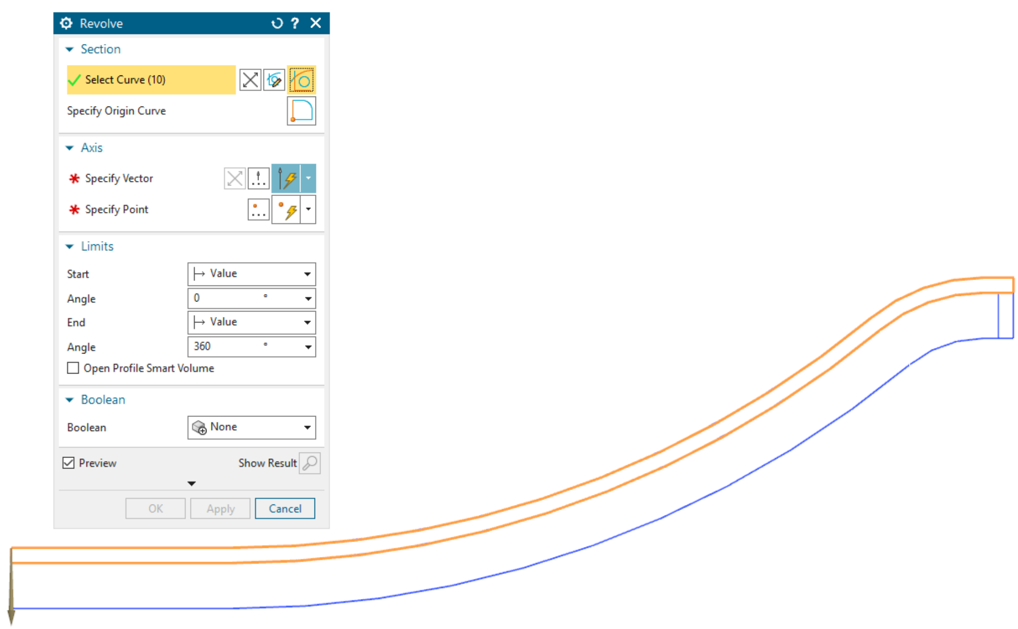
- ➤ Click on **Insert** → **Design Feature** → **Revolve**

OR

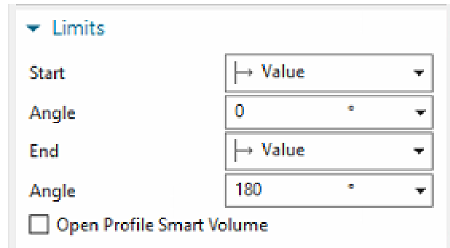
- ➤ Click on the **Revolve** button in the **Base Group**

Make sure that the *Selection Filter* is set to *Single Curve* as shown below on the *Selection Filter Toolbar*

- ➤ Click on each of the 10 curves as shown in the next figure
- ➤ In the **Axis** dialog box, in the **Specify Vector** option choose the **Positive XC-direction**

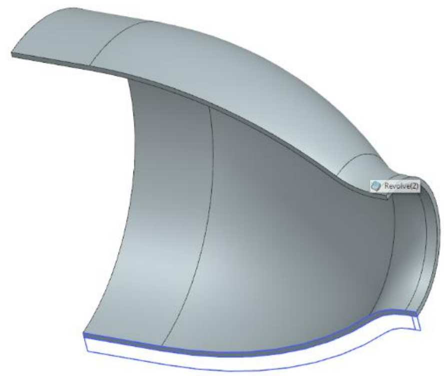


- In the **Specify Point** option, open **Point Dialog** enter the coordinates **(0, 0, 0)** so the curve revolves around XC-axis with respect to the origin



- Keep the **Start Angle** as **0** and enter **180** as the value for the **End Angle**
- Click **OK**

The solid is shown on the right. Now, we will create the edges.

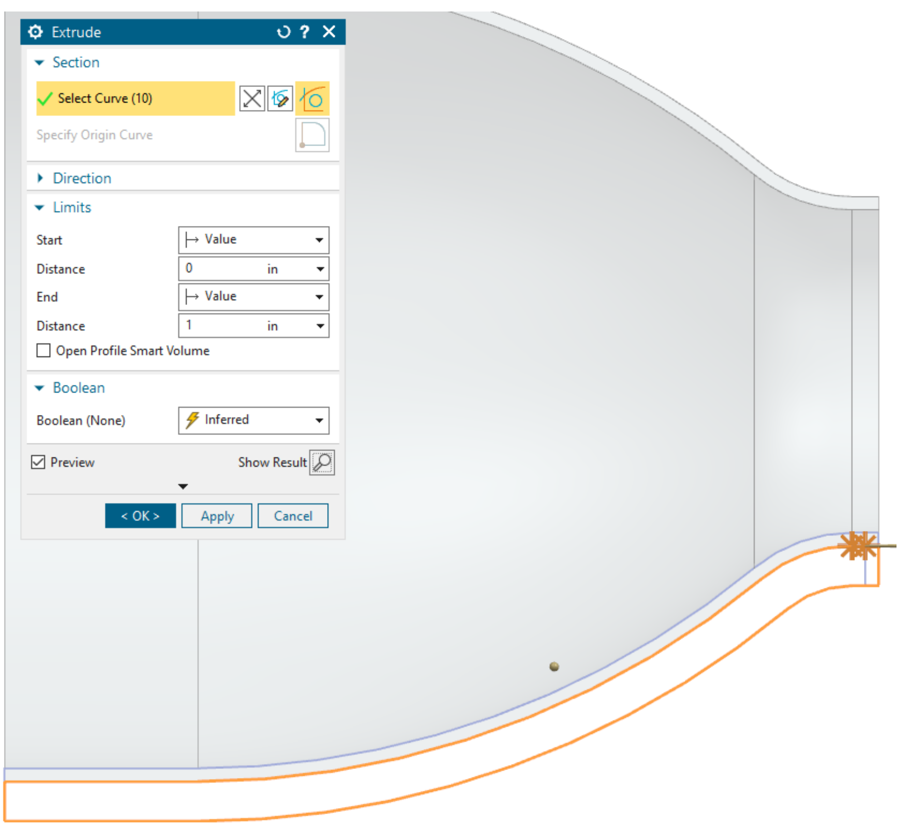


- Click on **Insert** → **Design Feature** → **Extrude**

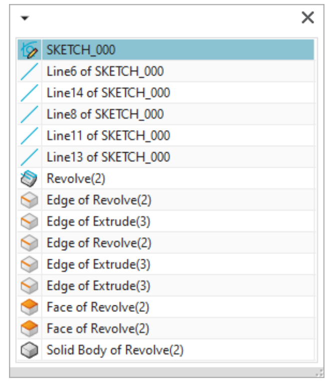
OR

- Click on the **Extrude** button in the **Base Group**

- Select the outer curve of the casing as shown in the figure below (again make sure that the **Selection Filter** is set to **Single Curve**).



Note: In case you are not able to select the proper lines, left-click and hold the mouse button and you will see a dialog box pop-up as shown which will provide you the options of which curve to select.

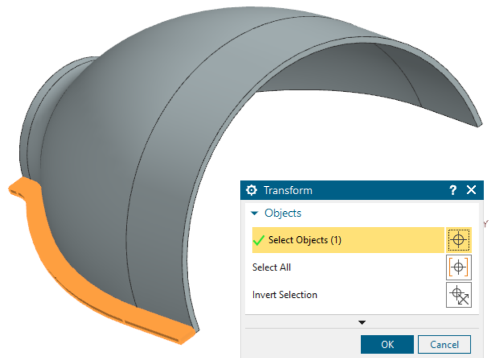


- ➢ Extrude this piece in the **negative Z-direction** by **0.5** inches

The final solid will be seen as follows.

We will now use the *Mirror* option to create an edge on the other side.

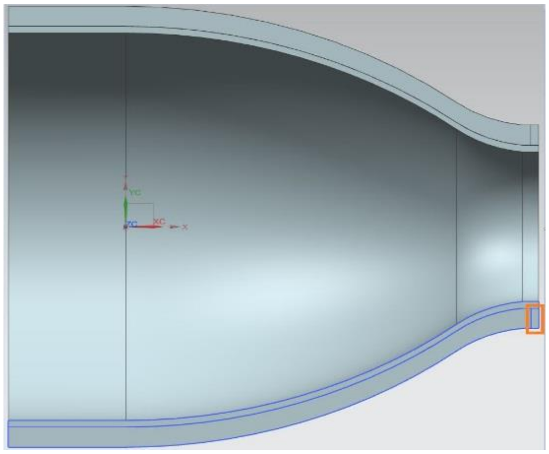
- ➢ Choose **Edit → Transform**
- ➢ Select the solid edge as shown. For this you will have to change the Filter in the dialog box to **Solid Body**



- ➢ Click **OK**
- ➢ Choose **Mirror Through a Plane**
- ➢ Select the **XC-ZC Plane** as shown below
- ➢ Click **OK**
- ➢ Select **Copy**
- ➢ Click **Cancel**

The edge will be mirrored to the other side as shown below.

We will now create a flange at the smaller opening of the casing as shown.

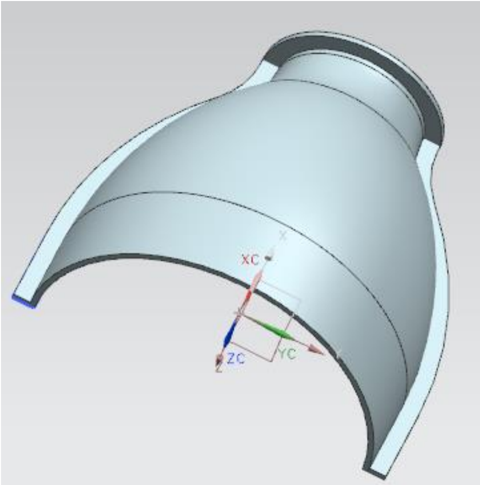


- ➢ Click on **Insert** → **Design Feature** → **Revolve**

Again, make sure that the *Selection Filter* is set to *Single Curve.* The default *Inferred Curve* option will select the entire sketch instead of individual curves.

- ➢ Revolve this rectangle in the positive **XC-direction** relative to the **Origin** just like for the casing. The **End Angle** should be **180**

This will form the edge as shown below.



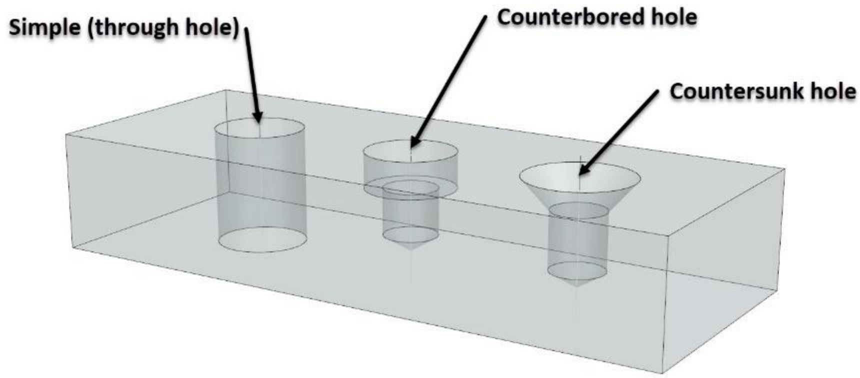
The lower casing is completed. Do not forget to save the model.

4.5 REMOVE FEATURES

Remove Features enable you to remove a portion of the existing 3d geometry or geometries to create an object with additional features that are part of the design, which are illustrated below.

4.5.1 General Hole

This option lets you create *Simple, Counterbored, Countersunk* and *Tapered* holes in solid bodies.



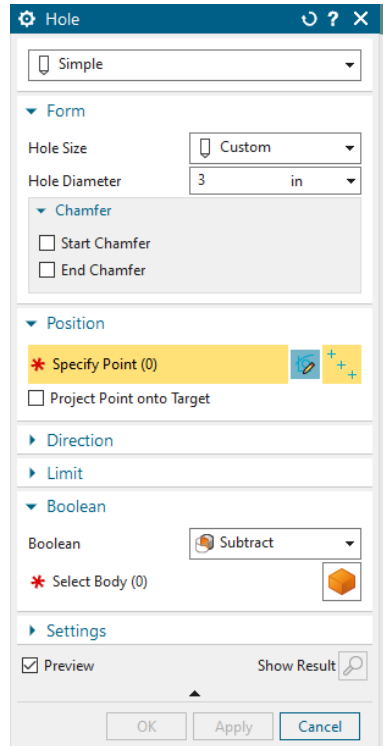
- Open the file **Arborpress_plate.prt**
- Choose **Insert** → **Design Features** → **Hole**

OR

- Click on the icon in the **Feature Toolbar** as shown

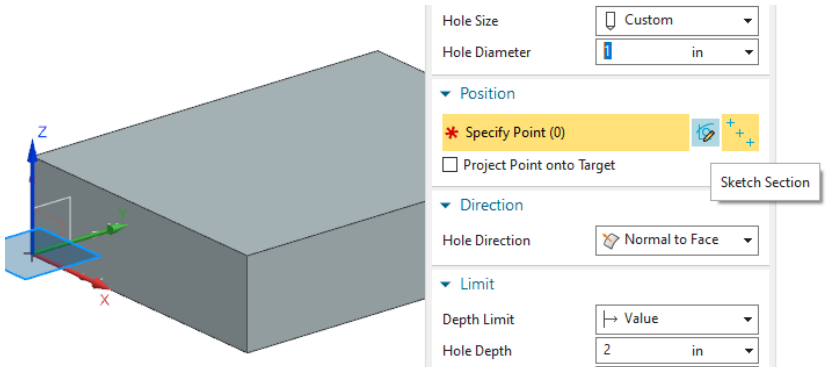
The ***Hole*** feature dialog will open. There are various selections that need to be done prior to making the holes. First you need to select the *Type* of the hole.

➢ Select the default **Simple**



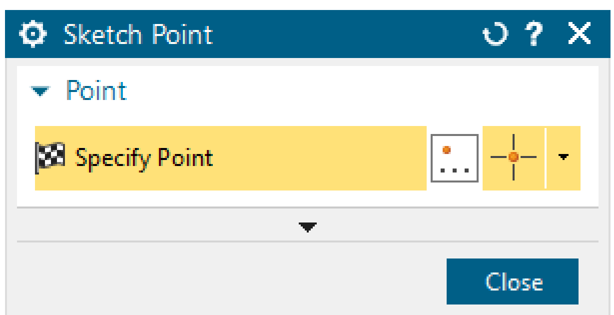
Next, you need to define the points at which you need to make the holes.

➢ Click on the **Sketch Section** icon in the **Position** dialog box and choose the top face of the plate as the **Sketch Plane**



➢ Click **OK**

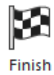
This will take you to the ***Sketch Plane***.



➢ Click on **Point Feature** 

 and the **Point Dialog** icon and specify all the points as given in the table below

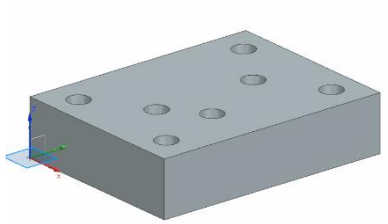
X	Y	Z
11.25	10.00	0.00
32.50	23.50	0.00
53.75	10.00	0.00
11.25	75.00	0.00
32.50	61.50	0.00
53.75	75.00	0.00

- Click **OK** after you enter the coordinates of each point
- Click **Close** once you have entered all the points
- Click on **Finish** flag in the top left corner of the window 

This will take you out of the *Sketch* mode and bring back to the original *Hole* window on the graphics screen.

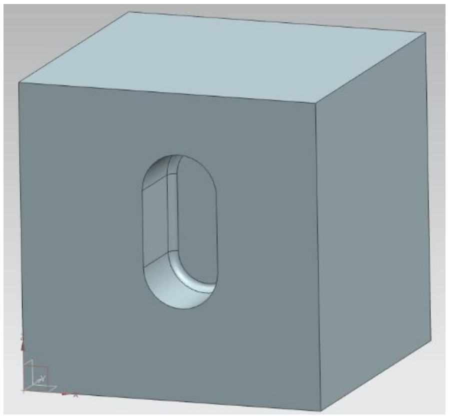
- In the **Form** dialog, choose the default option of **Simple Hole**
- Enter the following values in the **Dimensions** window

  Diameter = **8** inches
  Depth = **25** inches
  Tip Angle = **118** degrees
- Choose **Subtract** in the **Boolean** dialog box and click **OK**



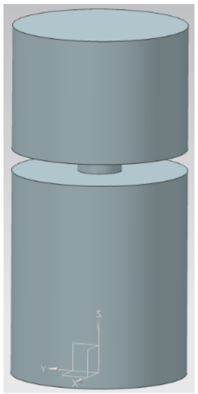
Make sure to save the model

4.5.2 Slot



This NX Slot option lets you create a passage through or into a solid body in the shape of a straight slot. An automatic subtract is performed on the current target solid. It can be rectangular, T-slot, U-Slot, Ball end or Dovetail. An example is shown on the right. Later releases of NX, Slot is command has been "hidden" or becomes a Legacy function, you can use command finder to locate this function within NX.

4.5.3 Groove



This NX Groove option lets you create a groove in a solid body, as if a form tool moved inward (from an external placement face) or outward (from an internal placement face) on a revolving part, as with a turning operation. An example is shown on the right.

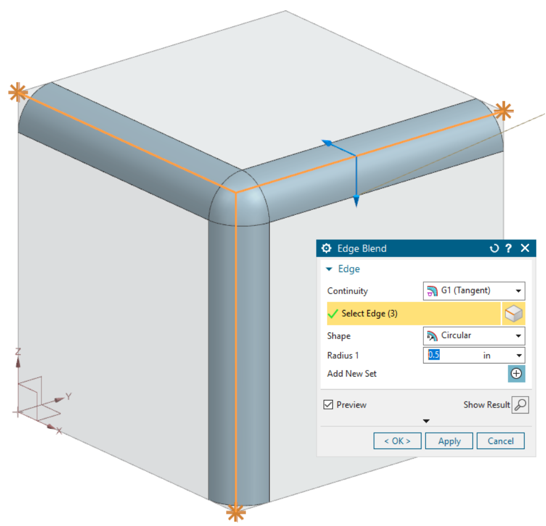
Note: *Pocket, Slot,* and *Groove* features are not commonly used in best practice. All the models created using these features can be modeled using *2D Sketches* and *Extrude/Revolve.*

4.6 FEATURE OPERATIONS

Feature Operations are performed on the basic *Form Features* to smooth corners, create tapers, make threads, do instancing and unite or subtract certain solids from other solids. Some of the *Feature Operations* are explained below.

4.6.1 Edge Blend

An *Edge Blend* is a radius blend that is tangent to the blended faces with controls of which type of tangencies implies. This feature modifies a solid body by rounding selected edges. This command can be found under ***Insert → Detail Feature → Edge Blend***. You can also click on its icon in the ***Base** Group.* You need to select the edges to be blended and define the *Radius* of the *Blend* as shown below.



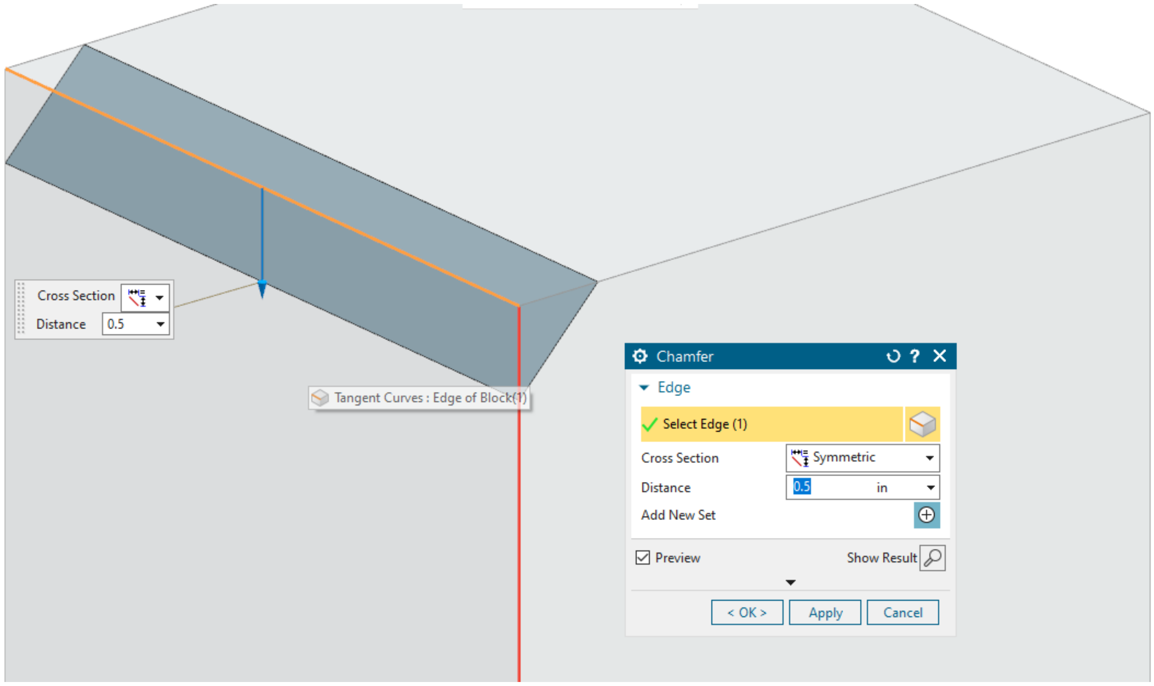
Similar to *Edge Blend* you can also do a *Face Blend* 

by selecting two adjacent faces.

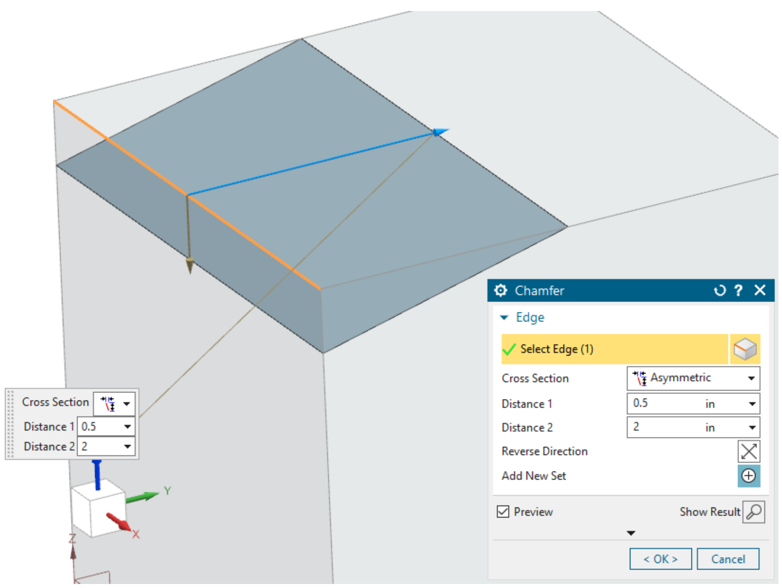
4.6.2 Chamfer

The [NX *Chamfer Function*] operates very similarly to the *Blend Function* by adding or subtracting material relative to whether the edge is an outside chamfer or an inside chamfer. This command **Chamfer** 

can be found in the **Base** group of **Home**. You need to select the edges to be chamfered and define the *Distance* of the *Chamfer* as shown below.



In the Cross Section, Asymmetric could model chamfer edge with different distances in each side

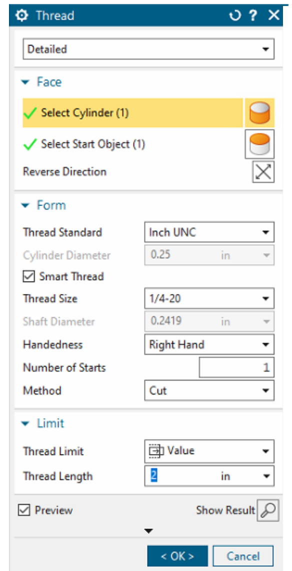


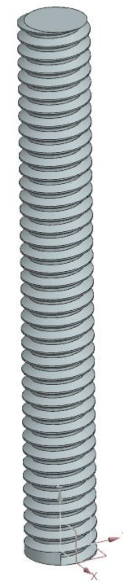
4.6.3 Thread

Threads can only be created on cylindrical faces. [The NX *Thread Function*]() lets you create *Symbolic* or *Detailed* threads (on solid bodies) that are right or left handed, external or internal, on cylindrical faces such as *Holes*, *Bosses*, or *Cylinders*. It also lets you select the method of creating the threads such as cut, rolled, milled or ground. You can create different types of threads such as metric, unified, acme and so on. To use this command, go to

Insert → Design Feature → Thread.

An example of a *Detailed Thread* is shown on the right.

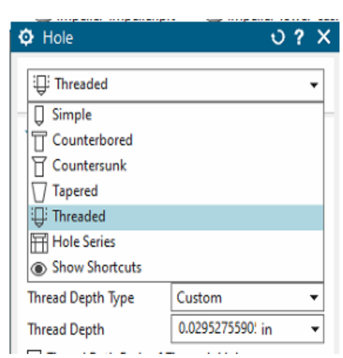




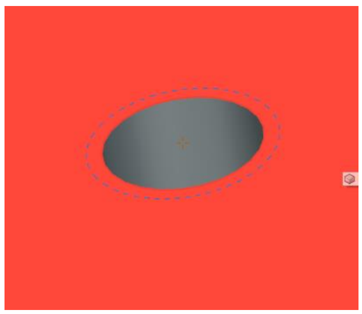
An example of a *Symbolic Thread* is shown below

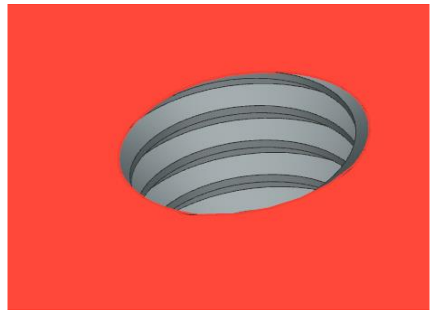
You can see the actual thread using *Detailed Thread.* However, we **highly** recommend you use the ***Symbolic Thread*** instead of ***Detailed Thread*** because modeling the actual threads in your model will increase the burden of graphics card and slow down your graphic process speed and performance. This will become more evident when you are opening or loading a part or assembly with detailed threads.

For *Threaded Holes*, it is recommended to use **Hole** command with *Threaded* mode instead of using *Thread* command (*Insert → Design Feature → Hole*), this is due to the fact for reducing the number of features in the design.



Note: use **hole** command with *Treaded* mode to create threaded hole will only show Symbolic Thread (left figure) instead of actual thread. If you need to show the real thread in your hole (right figure), you still need to use Thread command (*Insert → Design Feature → Hole).*

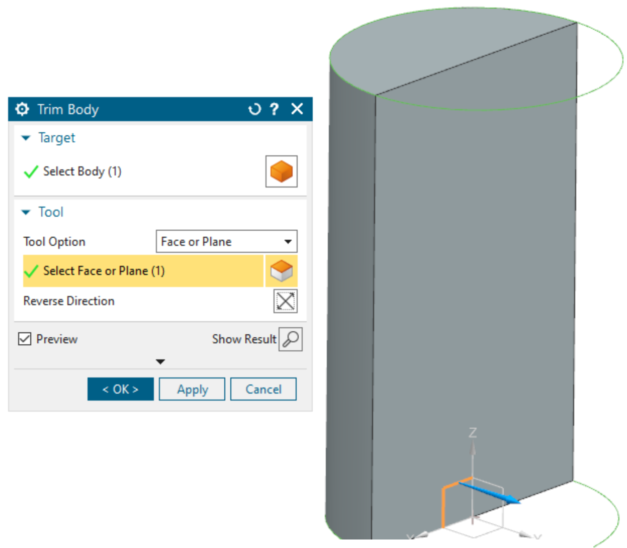




4.6.4 Trim Body

You can use the *Trim Body* function to trim a solid body with a sheet body or a *Datum Plane*, and at the same time retain parameters and associativity. To use this command, go to

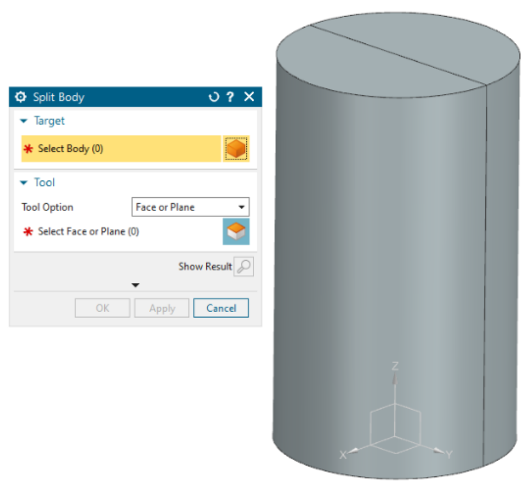
Insert → Trim → Trim Body or click on its icon in the *Feature Group.* An example is shown on the right.



4.6.5 Split Body

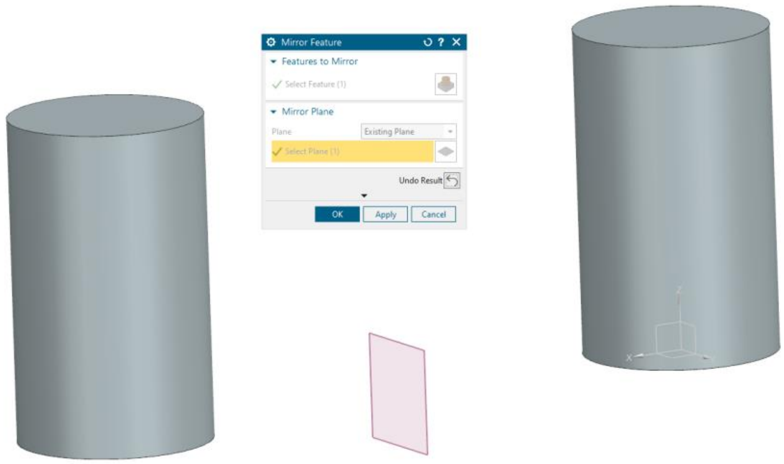
A solid body can be split into two bodies by a plane or a sheet body. To use this command, go to

Insert → Trim → Split Body or click on its icon in the *Feature Group.* An example is shown on the right.



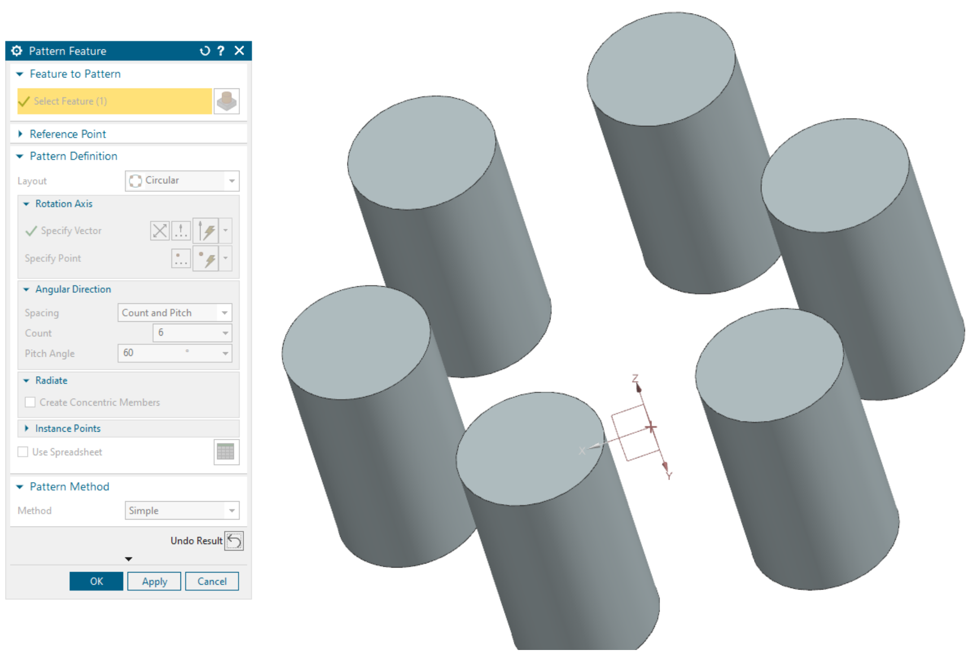
4.6.6 Mirror

Mirror is a type of *Associative Copy* in which a solid body is created by mirroring the body with respect to a plane. To use this command, go to *Insert → Associative Copy → Mirror Feature* or click on its icon in the *Mirror Feature* in the *Base group of Ribbon Bar.* An example is shown below.

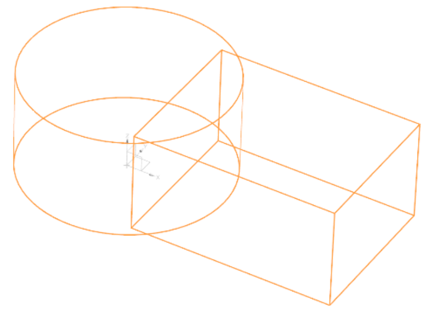


4.6.7 Pattern

A *Design Feature* or a *Detail Feature* can be made into dependent copies in the form of a [*Pattern Array*](). It can be *Linear, Circular, Polygon, Spiral*, etc. This particularly helpful feature saves plenty of time and modeling when you have similar features. For example, threads of a gear or holes on a mounting plate, etc. This command can be found under *Insert → Associative Copy → Pattern Feature.* You can also click on its icon in the *Feature Group.* An example is shown below.



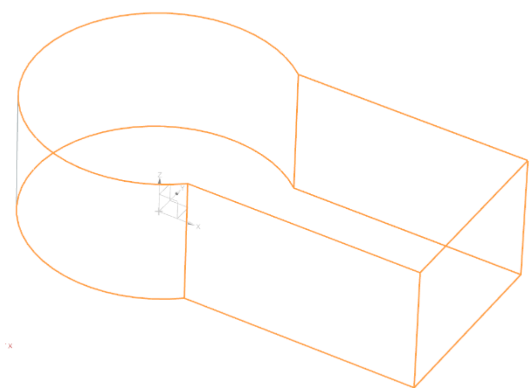
4.6.8 Boolean Operations

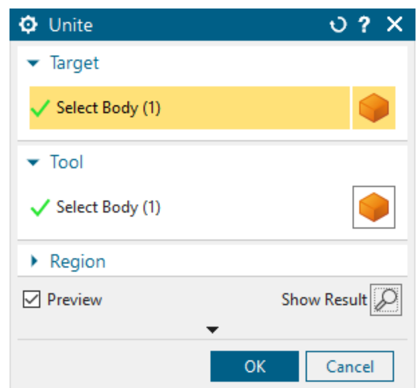


There are three types of [*Boolean Operations: Unite, Subtract,* and *Intersect*]. These options can be used when two or more solid bodies share the same model space in the part file. To use this command, go to *Insert* → *Combine* or click on their icons in the *Feature Group*. Consider two solids given: a block and a cylinder are next to each other as shown on the right.

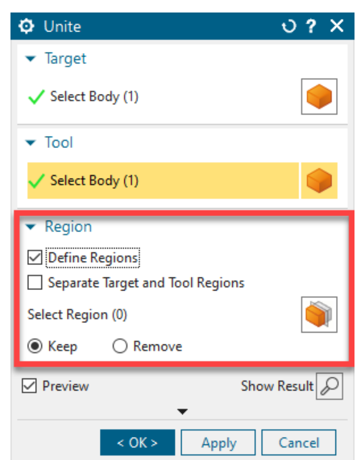
4.6.8.1 Unite

The unite command adds the *Tool body* with the *Target body*. For the above example, the output will be as follows if *Unite* option is used.



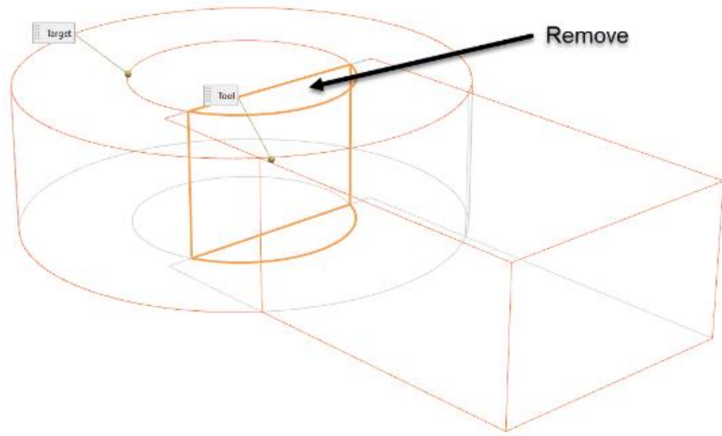


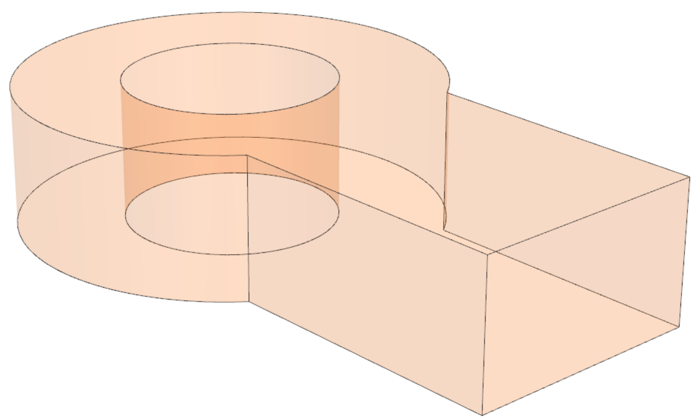
When Unit command of two similar geometries as shown below, where there is a hole through the center of the cylinder, you can remove the extra section of rectangle that interferes with the hole by selecting / expanding the Region Option during the Unit operation.



Select the Target Body → select Tool Body → select Define Regions → (option Keep or Remove) → select Remove.

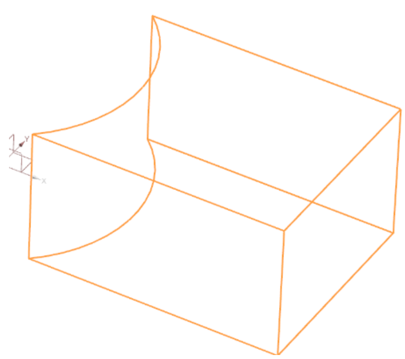
The interfered section of rectangle is removed as shown below.

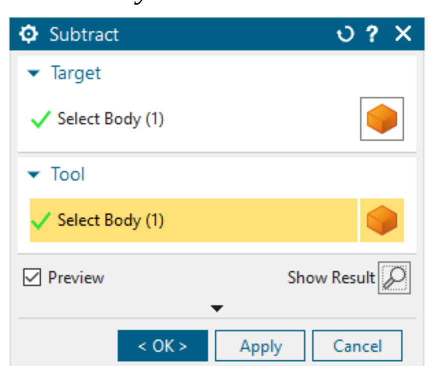




4.6.8.2 Subtract 

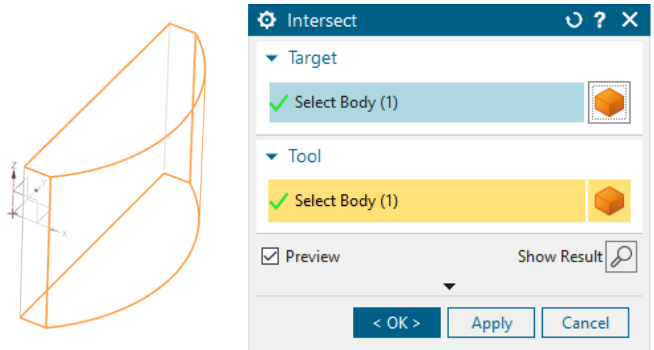
When using the subtract option, the *Tool Body* is subtracted from the *Target Body*. The following would be the output if the *Block* is used as the *Target* and the *Cylinder* as the *Tool*.





4.6.8.3 Intersect 

This command leaves the volume that is common to both the *Target Body* and the *Tool Body*. The output is shown below.

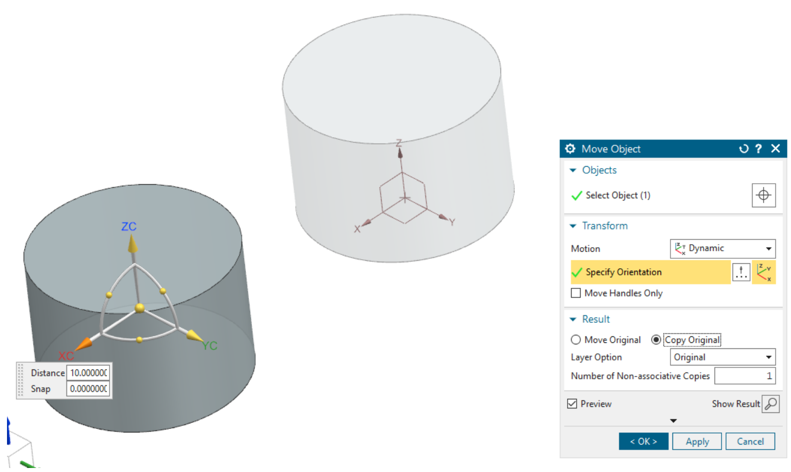


4.6.9 Move

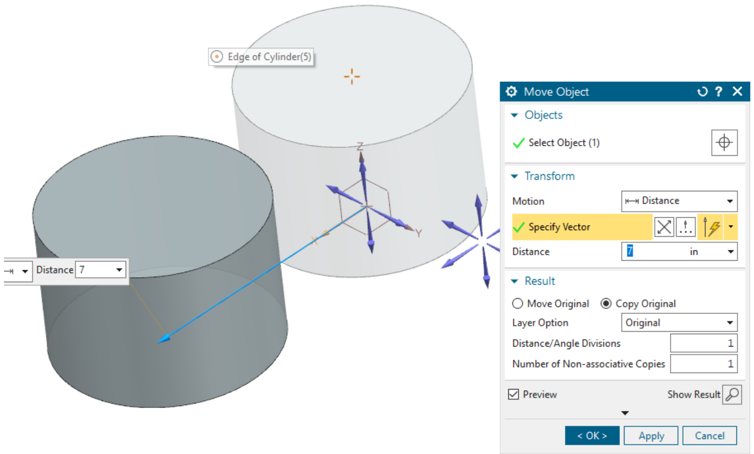
If you want to *Move* an object with respect to a fixed entity,

➢ Click on **Edit → Move Object**

You can select the type of motion from the *Motion* drop-down menu. The default option is *Dynamic***.** With this you can move the object in any direction. There are several other ways of moving the object.



If you choose *Distance* you can move the selected object in the X-Y-Z direction by the distance that you enter.



- Click on **Specify Vector** and select the direction.
- For example, type **7** in the **Distance** box. This will translate the cylinder a distance of 7 inches along X-Axis
- Click **OK**

As you can see, we have moved the cylinder along the X-direction. Similarly, we can also copy the cylinder by a specified distance or to a specified location by selecting the *Copy Original* option in the *Result*.

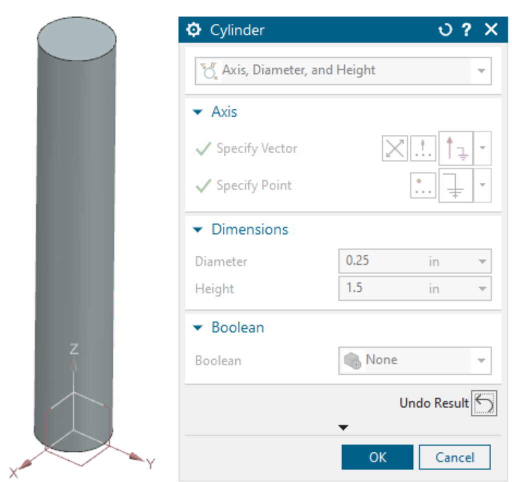
4.7 EXAMPLES

4.7.1 Hexagonal Screw

- Create a new file and save it as **Impeller_hexa-bolt.prt**
- Choose **Insert → Design Feature → Cylinder**
- The cylinder should be pointing in the **Positive ZC-Direction** with the center set at the **Origin** and with the following dimensions:

  Diameter = **0.25** inches

  Height = **1.5** inches



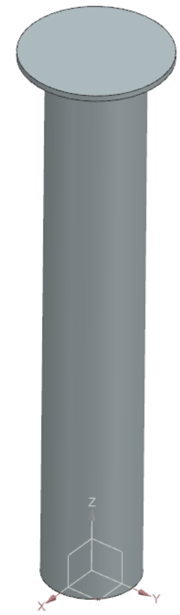
Now create a small step cylinder on top of the existing cylinder.

- Create a **Cylinder** with the following dimensions:

  Diameter = **0.387** inches

  Height = **0.0156** inches

- Click on the top face of the existing cylinder
- On the **Point Constructor** window, choose the **Arc/Ellipse/Sphere Center** icon from the drop-down **Type** menu
- Click **OK** to close the **Point Constructor** window
- Under the **Boolean** drop-down menu, choose **Unite**



The two cylinders should look like the figure shown on the right.

- Click **Sketch** and choose the top surface of thin cylinder and **OK**
- Choose **Insert → Curve → Polygon**
- Select the center of the top circle as the **Center Point**
- On the **Sides** window, type **6** for the **Number of Sides**

There are three ways to draw the polygon.

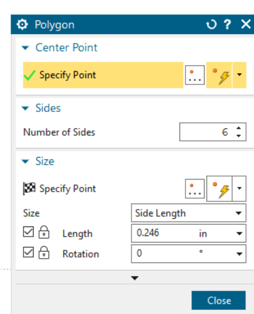
- *Inscribed Radius*
- *Circumscribed Radius*
- *Side Length*

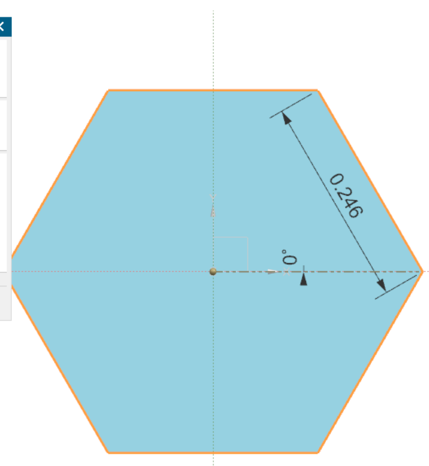
- Choose **Side Length** and enter the following dimensions:

  Length = **0.246** inches

  Rotation = **0.00** degree

- Click **OK**

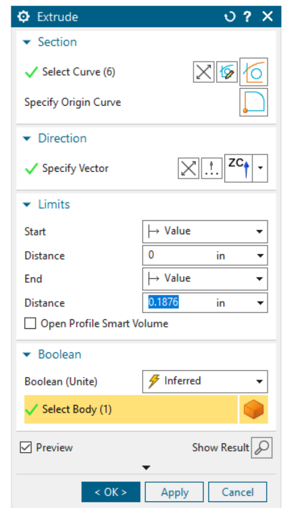


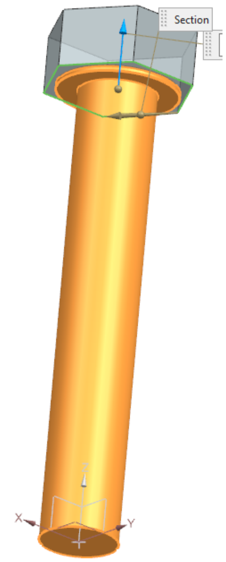


Now we will extrude this polygon.

- Choose **Insert → Design Feature → Extrude**

- ➢ Choose the Hexagon Sketch to be extruded
- ➢ Enter the **End Distance** as **0.1876** inches



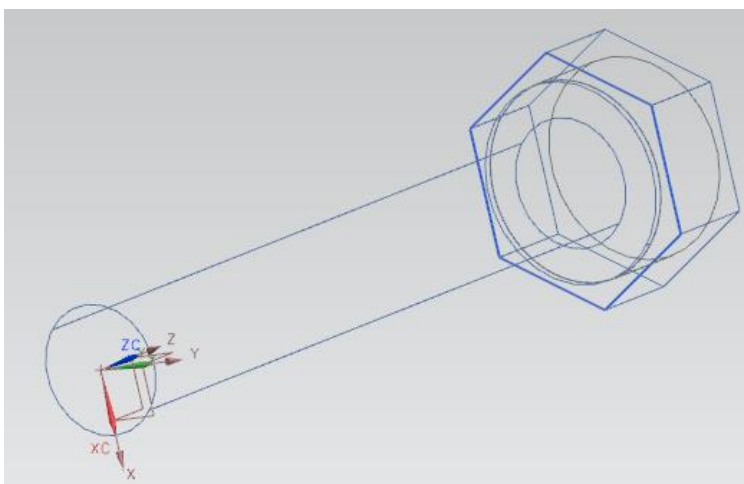


The model looks like the figure on the right after extrusion.

- ➢ On top of the cylinder that has a diameter of 0.387 inches, insert another cylinder with the following dimensions.

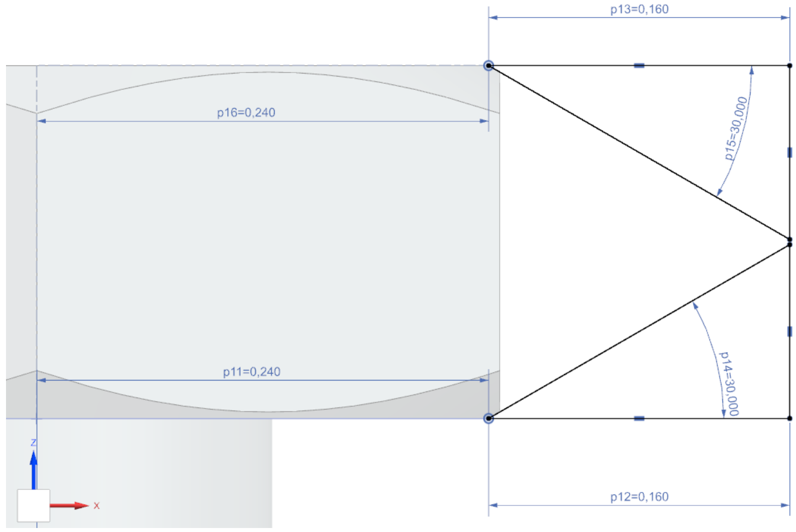
  Diameter = **0.387** inches
  Height = **0.1875** inches

You will only be able to see this cylinder when the model is in *Static Wireframe* since the cylinder is inside the hexagon head. The model will look like the following.

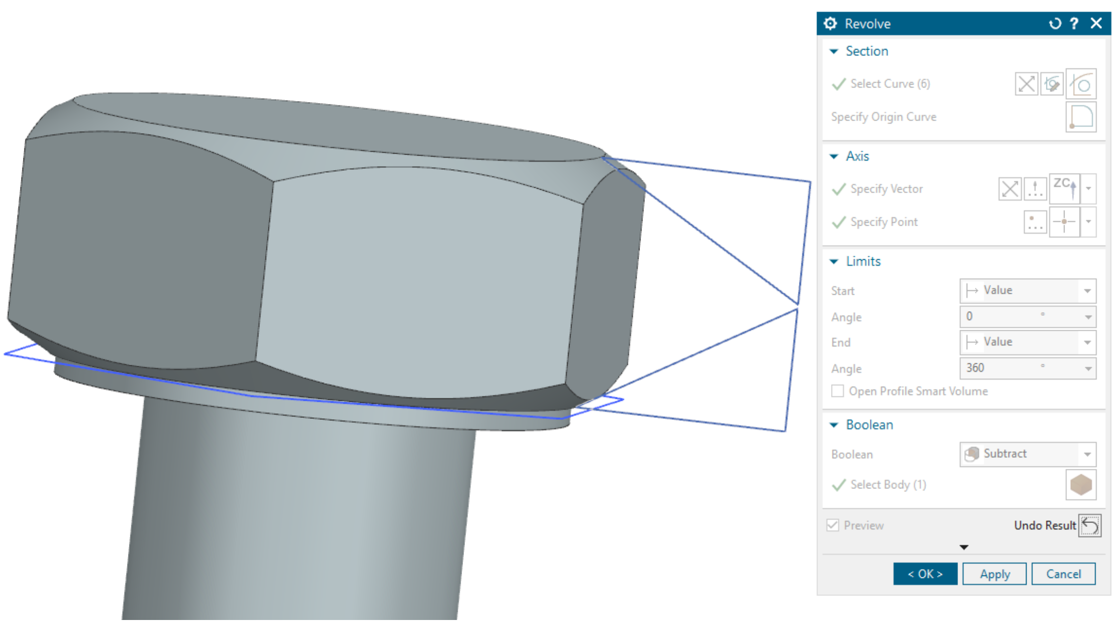


We will now use the feature operation *Revolve*.

- ➢ Click **Sketch** and make a drawing like figure below on X-Z Plane



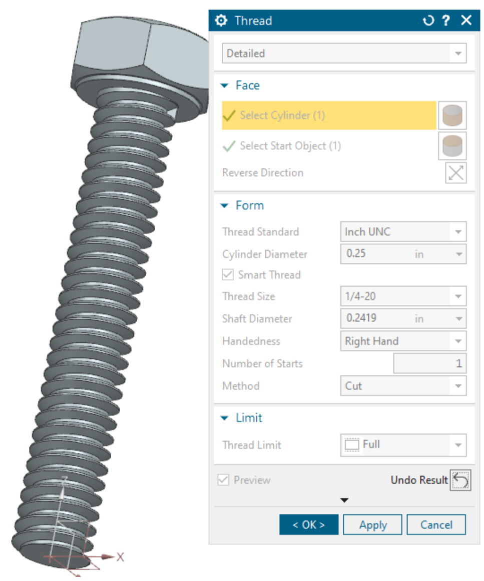
- ➢ **Finish**
- ➢ Choose **Insert** → **Design Feature** → **Revolve**
- ➢ Choose **6 curves** from the sketch
- ➢ Click **Specify Vector** and choose ZC axis
- ➢ Click **Specify Point** to **(0,0,0)** in **Point Dialog**
- ➢ Choose **Substract** in **Boolean operation**
- ➢ **OK**



This will give you the hexagonal bolt as shown.

Now we will add *Threading* to the hexagonal bolt.

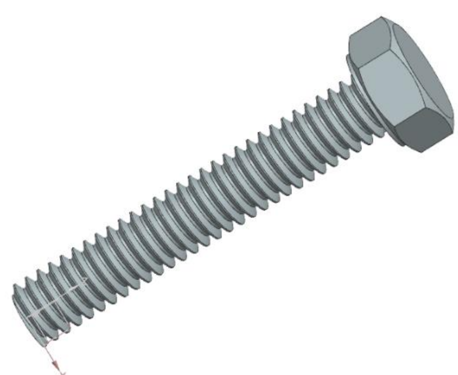
- ➢ Choose **Insert** → **Design Feature** → **Thread**
- ➢ Choose **Detailed**
- ➢ Keep the **Handedness** to be **Right Hand**
- ➢ Click on the bolt shaft (the long cylinder below the hexagon head)



Once the shaft is selected, all the values will be displayed in the *Thread* window. Keep all these default values.

- ➢ Click **OK**

The hexagon bolt should now look like the figure on the right. Save the model.

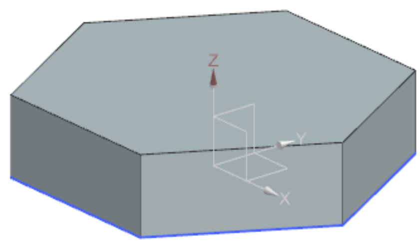


4.7.2 Hexagonal Nut

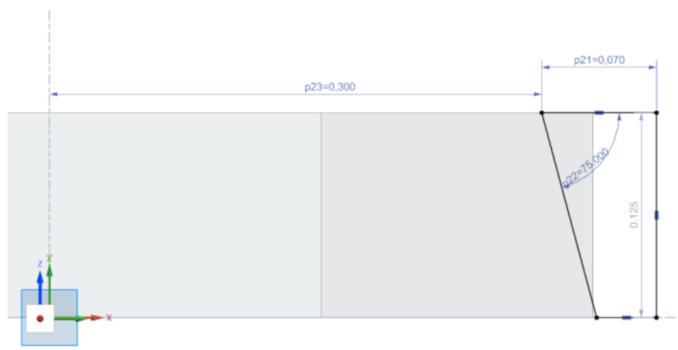
Additional reference material regarding how to create Hexagonal Nut.

- ➢ Create a new **Model** file and save it as **Impeller_hexa-nut.prt**
- ➢ Click **Sketch** and choose X-Y plane
- ➢ Choose **Insert** → **Curve** → **Polygon**
- ➢ Input **Number of Sides** to be **6**
- ➢ Create a hexagon with each side measuring **0.28685** inches and Rotation of 0 degree and constructed at the **Origin**
- ➢ **Finish**
- ➢ Choose **Insert** → **Design Feature** → **Extrude**
- ➢ Select the **Hexagon** to be extruded and enter the **End Distance** as **0.125** inches
- ➢

The figure of the model is shown.



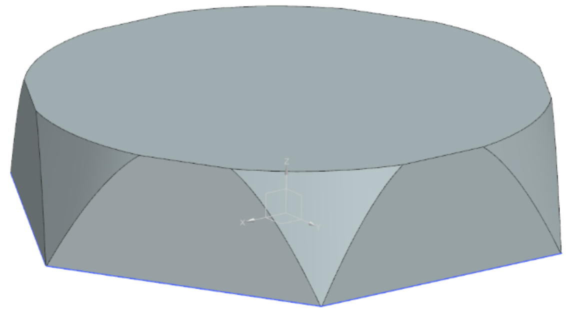
- ➢ Click **Sketch** and make a drawing like figure below on Y-Z Plane



- ➢ Choose **Insert** → **Design Feature** → **Revolve**
- ➢ Choose **3 curves** of triangle from the sketch
- ➢ Click **Specify Vector** and choose ZC axis
- ➢ Click **Specify Point** to **(0,0,0)** in **Point Dialog**

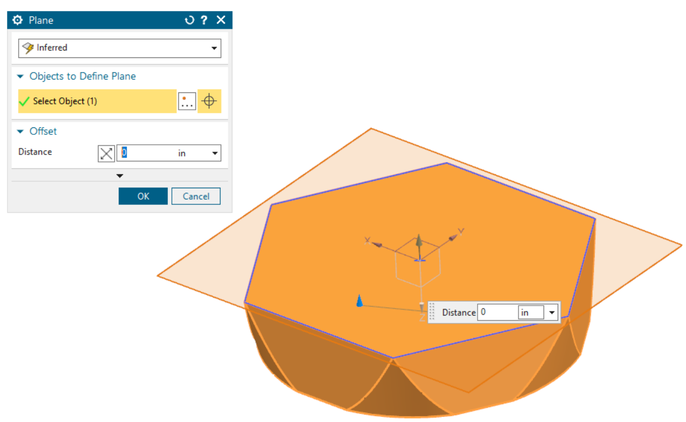
- ➢ Choose **Substract** in **Boolean operation**
- ➢ **OK**

The model will look like the following.



We will now use a *Mirror* command to create the other side of the *Nut.*

- ➢ Choose **Menu → Edit → Transform**
- ➢ Select the model and click **OK**
- ➢ Click **Mirror Through a Plane**
- ➢ Click on the flat side of the model as shown. Be careful not to select any edges



- ➢ Click on **OK**
- ➢ Click on **Copy**
- ➢ Click **Cancel**

You will get the following model.

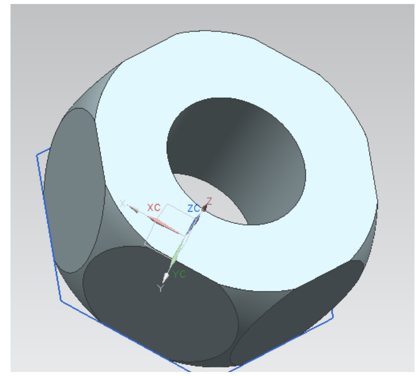
- ➢ Choose **Insert → Combine → Unite**
- ➢ Select the two halves and **Unite** them

- Insert a **Cylinder** with the vector pointing in the **ZC-Direction** and with the following dimensions:

  Diameter = **0.25** inches

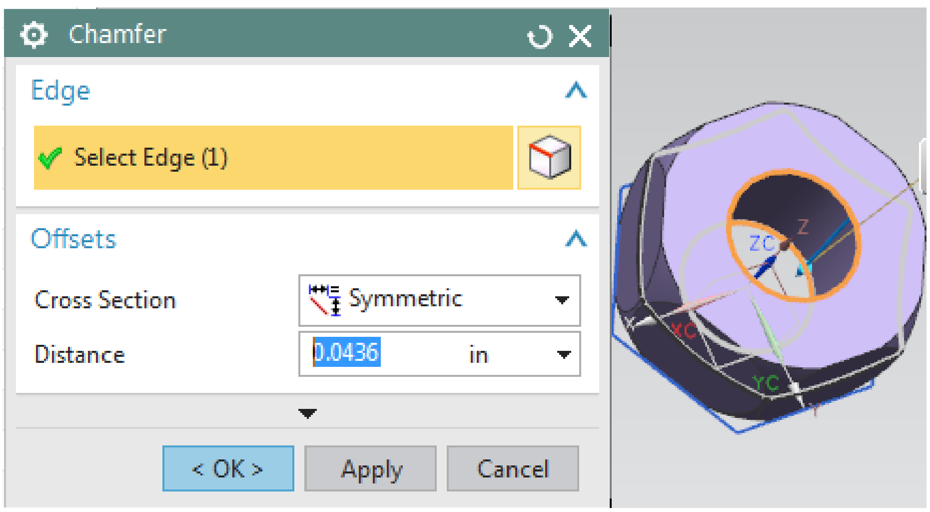
  Height = **1** inch

  Put the cylinder on the **Origin** and **Subtract** this cylinder from the hexagonal nut



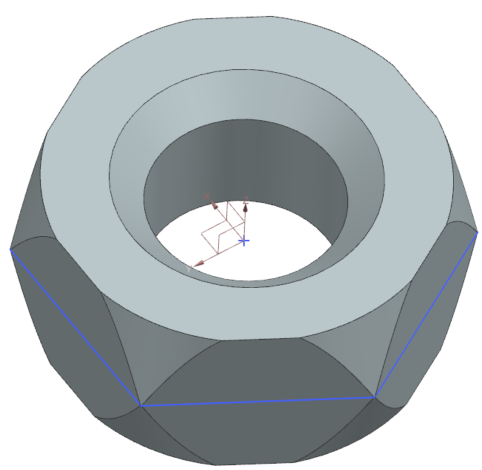
Now, we will chamfer the inside edges of the nut.

- Choose **Insert → Detail Feature → Chamfer**



- Select the two inner edges as shown and click **OK**
- Enter the **Distance** as **0.0436** inches and click **OK**

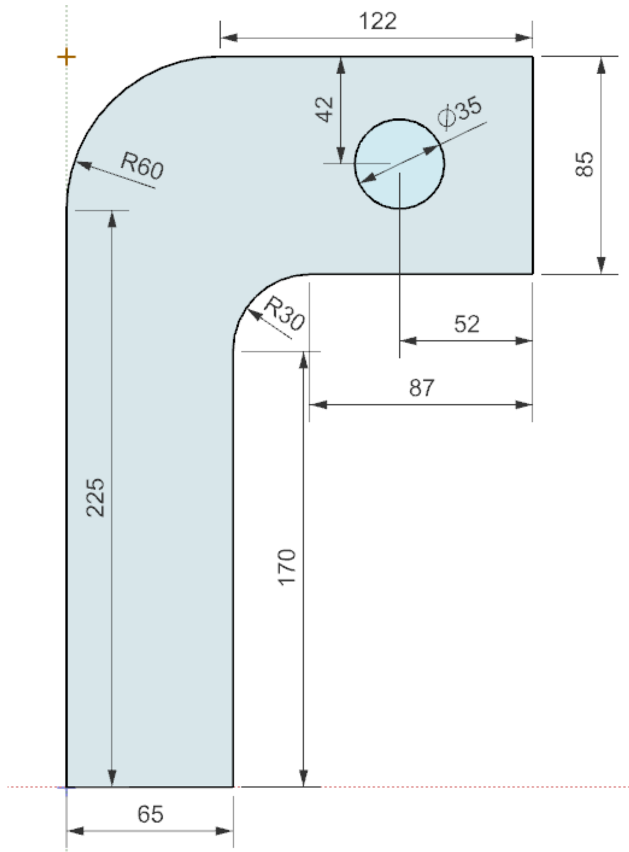
You will see the chamfer on the nut. Save the model.



4.7.3 L-Bar

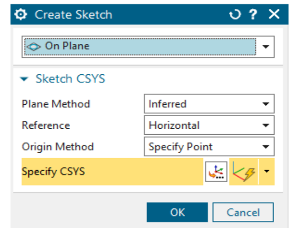
Here, we will use **Extrude *Feature*** *Operations* to efficiently [create this L-Bar](#).

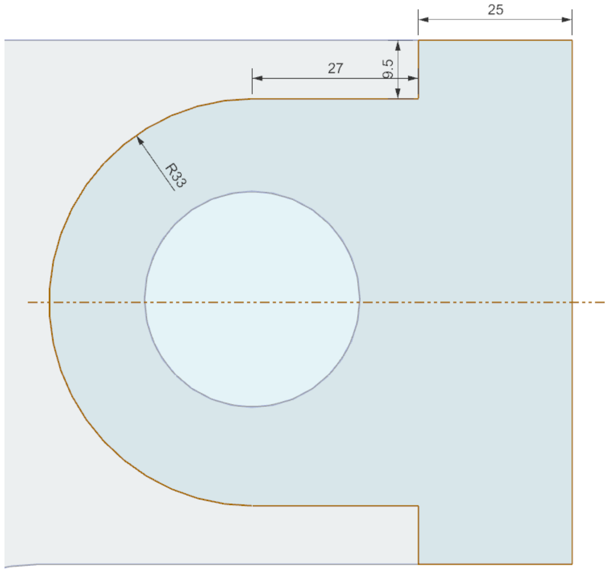
- ➢ Create a new file and save it as **Arborpress_L-bar**
- ➢ Click **Sketch** and make a drawing like figure below on X-Y Plane

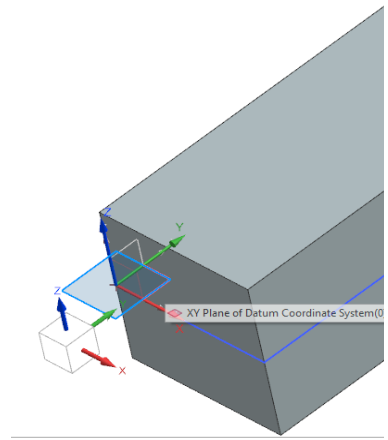


- ➢ **Finish**
- ➢ Choose **Insert** → **Design Feature** → **Extrude**
- ➢ Select Sketch curves above and change the width to **Symmetric Value** with the distance of **65** inches
- ➢ Click **OK**

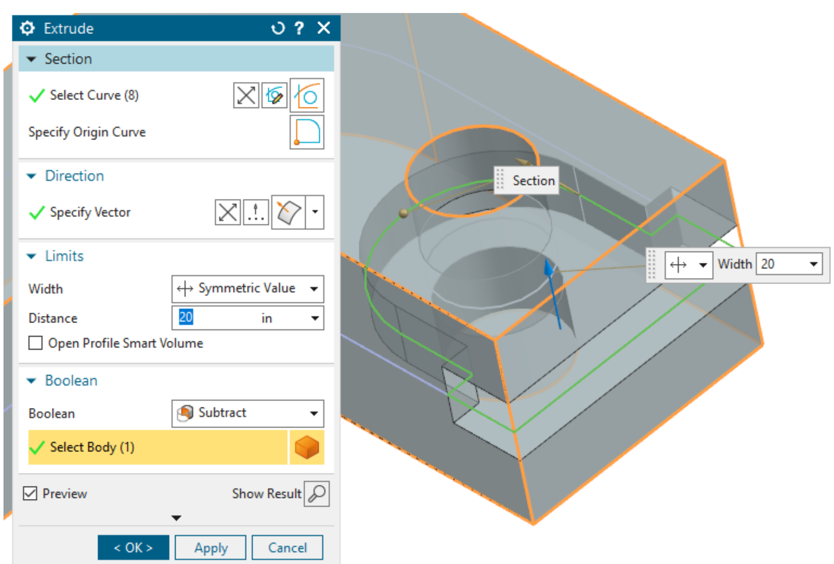
- ➢ Click **Sketch** and choose X-Y Plane like right figure
- ➢ Make a drawing like figure below







- ➢ **Finish**
- ➢ Choose **Insert** → **Design Feature** → **Extrude**
- ➢ Select Sketch curves above and change the width to **Symmetric Value** with the distance of **20** inches
- ➢ Change Boolean to **Subtract**

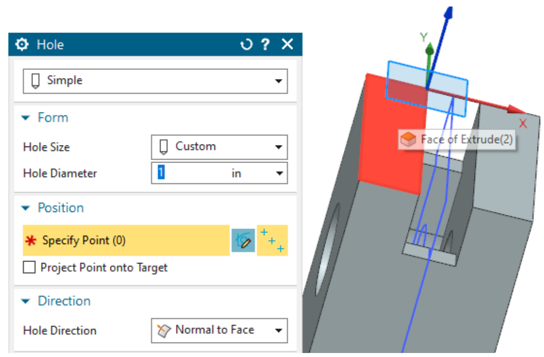


➢ **OK**

➢ Click **Hole** in the **Base** group of Ribbon bar and choose right plane of L-bar as reference plane

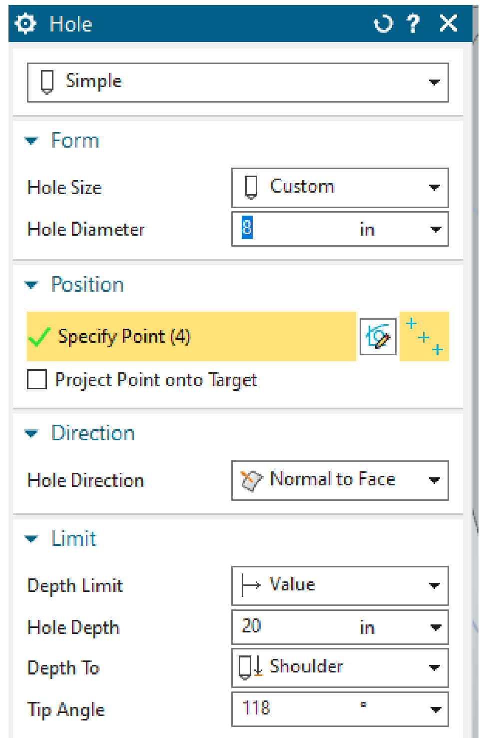
➢ Use Point Dialog to specify these points:

(182,275,21.25), (182,210,21.25)
(182,275,-21.25), (182,210,-21.25)

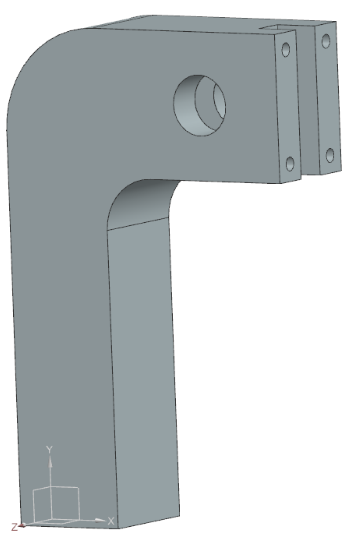


➢ Specify Hole Depth of 20 inches and Hole Diameter of 8 inches

➢ **OK**



You model should now look like the figure below. Save the model.



4.7.4 Rack

➢ Create a new part file and save it as **Arborpress_rack.prt**

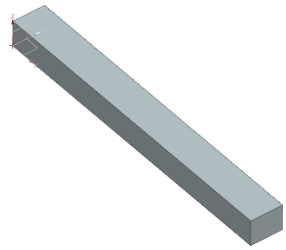
➢ Right-click, then choose **Orient View** → **Isometric**

➢ Choose **Insert** → **Design Feature** → **Block**

➢ Click Point Dialog in **Specify Point** and choose (0,0,0) as the reference point

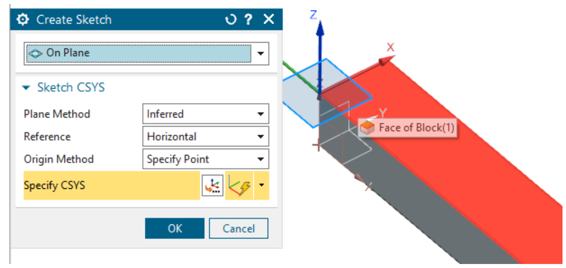
- Click **OK**
- Dimensions:

  XC: 240, YC: 25, ZC: 20

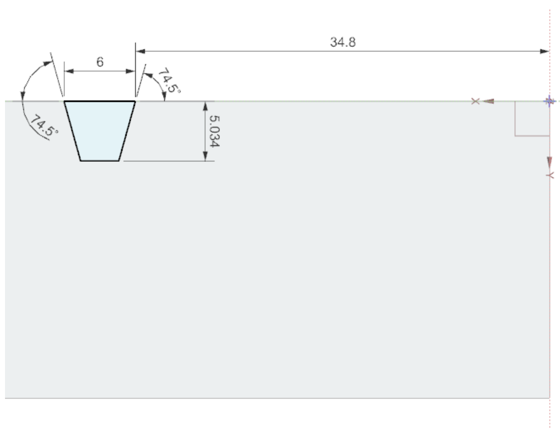


The extruded body will appear as shown on the right.

- Click **Sketch** and choose top surface of rectangular bar

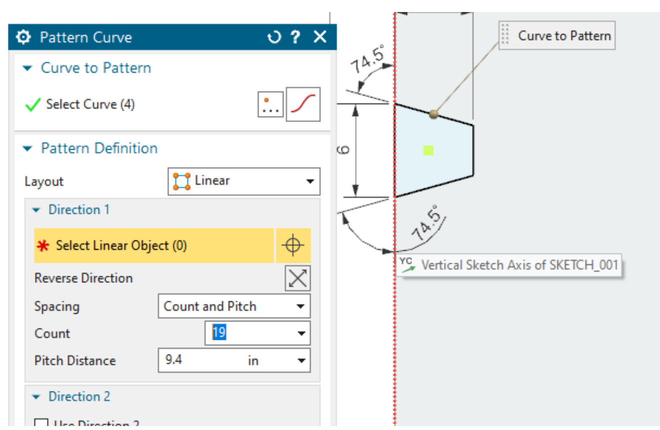


- Make a drawing like figure below

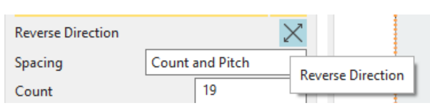


- Use **Pattern Feature**  to quickly duplicate these curves.

- Select 4 curves and select Vertical Sketch Axis in Select Linear Object like right figure shown

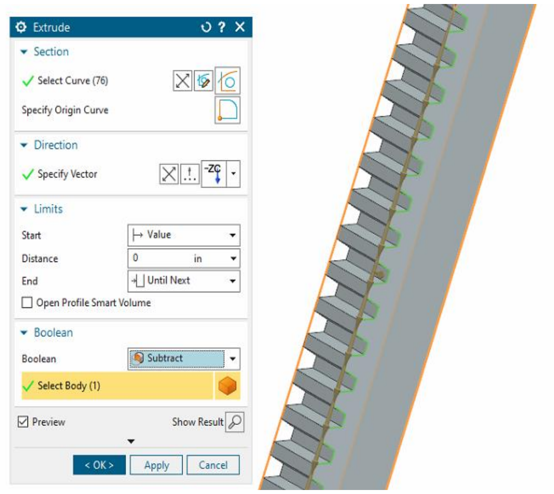


- Enter 19 in Count and 9.4 inches in Pitch Distance
- Click Reverse Direction sign

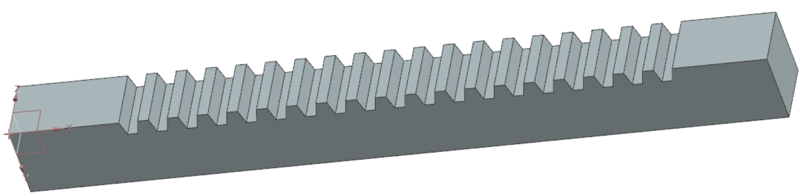


- **OK**
-

- ➢ Choose **Insert → Design Feature → Extrude**
- ➢ Select Sketch curves above
- ➢ Choose **-ZC** direction in the **Specify Vector**
- ➢ Change End to Until Next in Limits section
- ➢ Change Boolean to **Subtract** as shown in the right figure

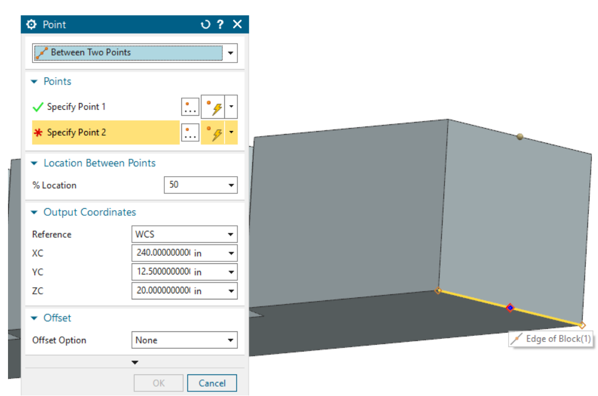


The model of the *Rack* will look as the one shown in the figure.



We will now create a *Hole* at the center of the rectangular cross section. To determine the center of the cross-section of the rectangular rack, we make use of the *Snap Points*

- ➢ Choose **Insert → Design Feature → Cylinder**
- ➢ Choose **–XC**-Direction in the **Specify Vector** dialog box
- ➢ Click on the **Point Dialog**
- ➢ In the **Points** dialog box select **Between Two Points** option and select the points as shown in the figure on the right (diagonally opposite points). The option selects the midpoint of the face for us



- ➢ Click **OK**

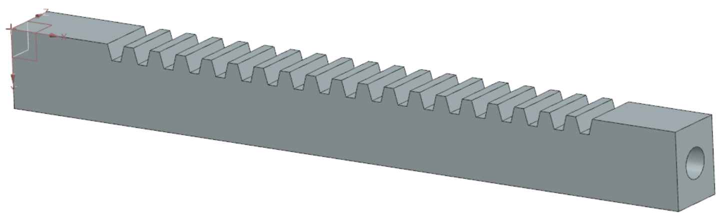
➢ Enter the following values in the **Dimension** dialog box

Diameter = **10** inches

Height = **20** inches

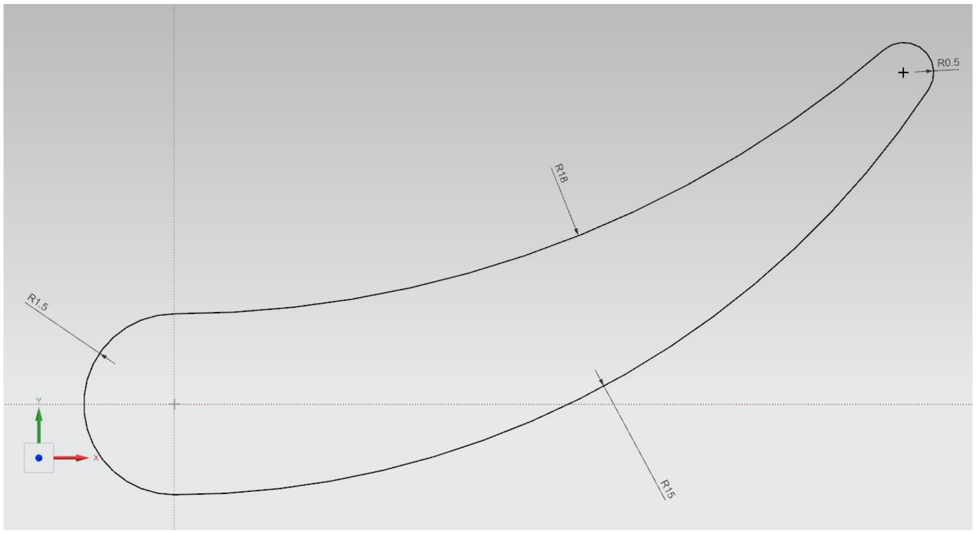
➢ Choose **Subtract** in the **Boolean** dialog box

The final model is shown below. Save and close the model.



4.7.5 Impeller

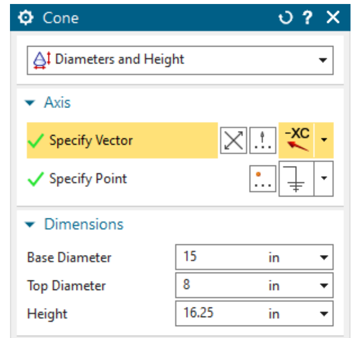
Open the **Impeller_impeller.prt** file you made in Chapter 3. It should like the figure below.



We will complete the Impeller but let us model a cone first.

➢ Choose **Insert → Design Feature → Cone**

- Select **Diameters and Height**



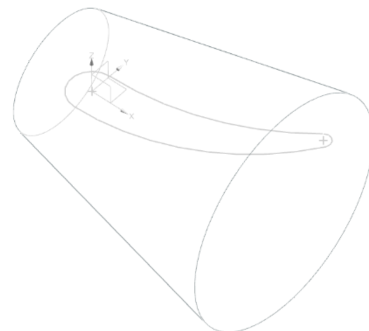
- Select the **–XC**-Direction in the **Specify Vector** dialog box
- In the **Point Dialog**, enter the coordinates **(14, 0, 0)**.
- Enter the following dimensions:

  Base Diameter = **15** inches

  Top Diameter = **8** inches

  Height = **16.25** inches

The cone will be seen as shown below if you choose *Static Wireframe View*.

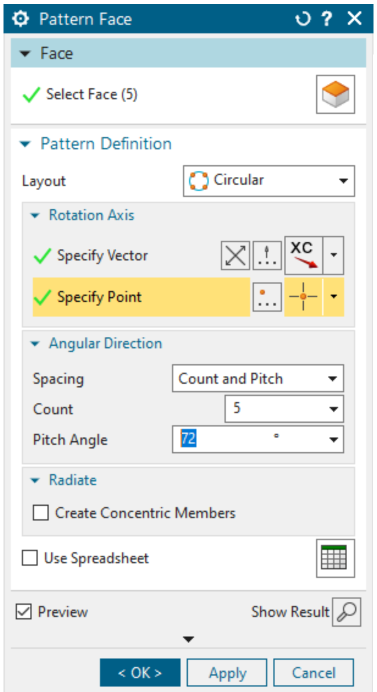


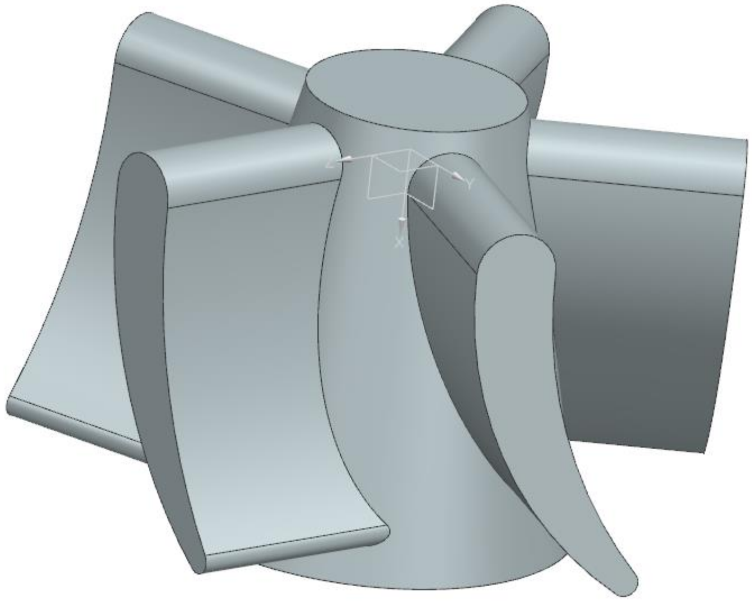
- **Extrude** the **Airfoil** curve in the **Z**-direction by **12** inches
- **Unite** the two solids in the **Boolean** operation dialog box

The model will be as follows.

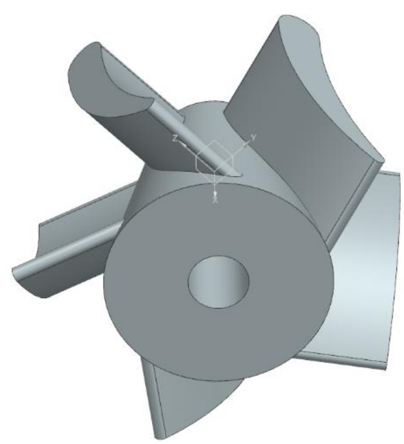
Now let us create five instances of this blade to make the impeller blades.

- Click on **Insert → Associative Copy → Pattern Feature**
- Select the **Airfoil** you just created
- Select **Circular** layout
- Select the **XC**-Direction for the **Specify Vector** and the **Origin** for the **Specify Point**
- For **Count**, type in **5** and for **Pitch Angle**, enter **72**
- Click **OK**

- 



Now, let us create two holes in the cone for the shaft and the locking pin. Note that these holes can also be created by *Hole* menu option.



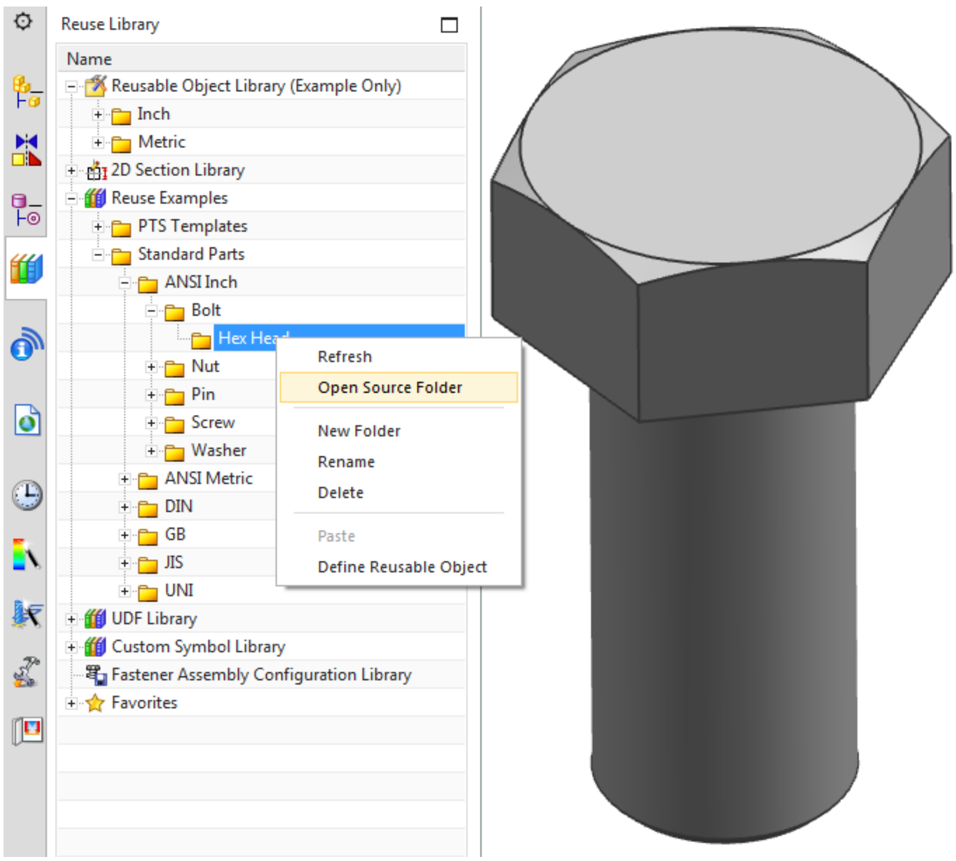
- **Subtract** a cylinder with a **Diameter** of **4** inches and a **Height** of **16** inches from the side of the cone with the larger diameter
- **Subtract** another cylinder with a **Diameter** of **0.275** inches and a **Height** of **0.25** inches from the side of the cone with the smaller diameter

The final model will look like the figure on the right. Save and close your work.

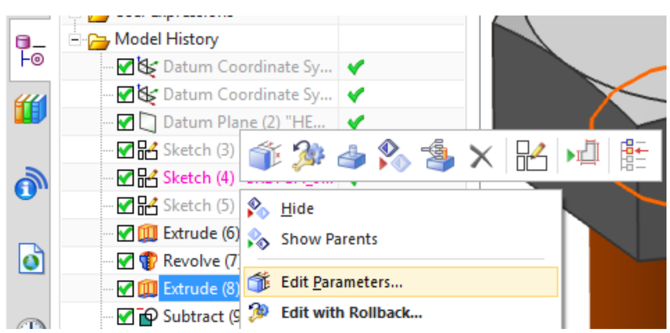
4.8 STANDARD PARTS LIBRARY

A better and faster approach for modeling standard parts like bolts, nuts, pins, screws, and washers is using the *Standard Parts Library*. For example, to model a hexagonal bolt,

- Choose **Reuse Library** → **Reuse Examples** → **Standard Parts** → **ANSI Inch** → **Bolt**
- Right-click on **Hex Head**
- Click on **Open Source Folder**
- Open **Hex Bolt, AI.prt**



You can now go to *Part Navigator* to see all the steps taken toward modeling this part and modify any feature. For example, to modify the length of the bolt, right-click on *Extrude (8) "BODY_EXTRUDE"* and choose *Edit Parameters.*



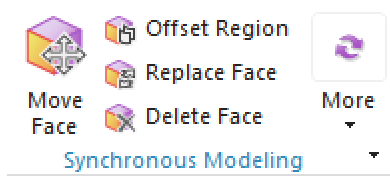
4.9 SYNCHRONOUS TECHNOLOGY

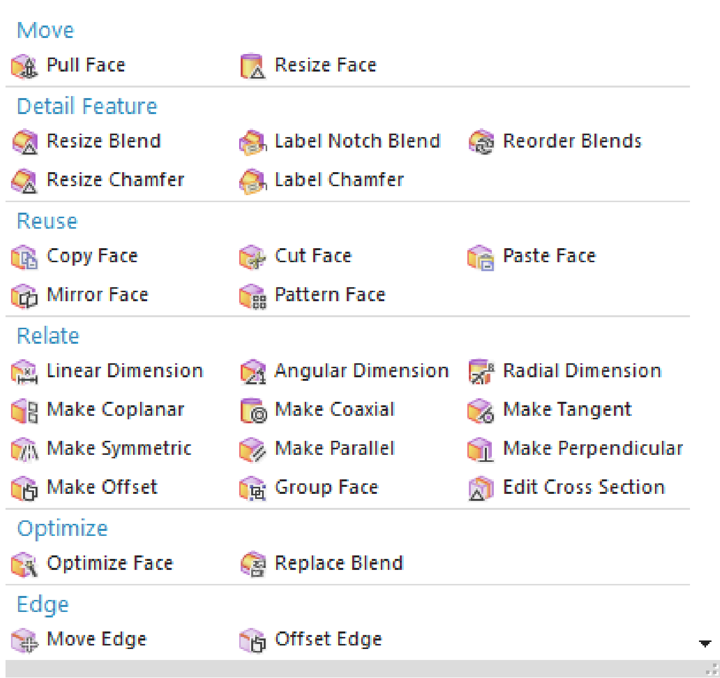
One of the important and unique features which NX offers apart from *Design Features* and *Freeform Modeling* is *Synchronous Technology*. With the options available in *Synchronous Modeling* group in the ribbon bar in the *Modeling Application* tab, the user can modify complex 3D models without the model history tree and without knowing the feature relationships and dependencies. The "push-and-pull" options can be used to modify the 3D model using faces, edges and cross-sections. NX supports the *Synchronous Modeling* to work with 3D models from CATIA, Pro/ENGINEER®, SolidWorks®, and Autodesk Inventor®, apart from the standard formats including IGES, ISO/STEP and JT.

For the purpose of illustrating the options available in *Synchronous Modeling*, let us consider the impeller part modeled in the previous section and export it as standard STEP format and save it.

- Open a new file in NX
- Choose **File** → **Import** → **Impeller_impeller.stp**

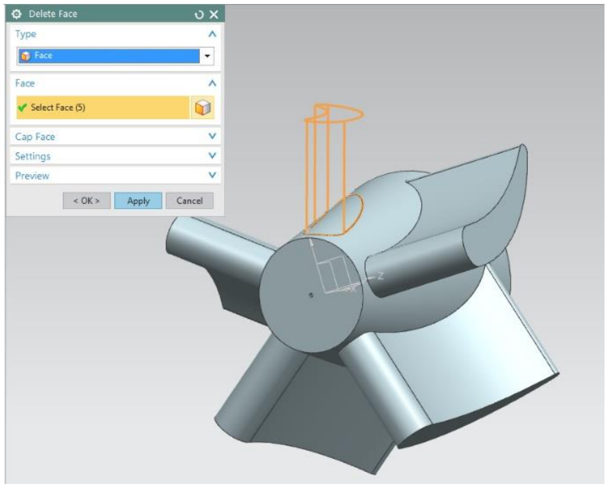
Observe here that the .stp file would not have any model history. We will explore some of the options available in the *Synchronous Modeling* group in the ribbon bar. Click *More* to view a comprehensive list of options available in synchronous modeling.

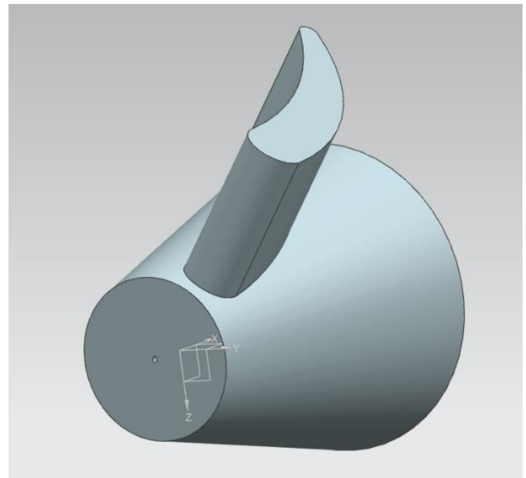




➢ Click **Delete Face** and select the faces of the blade to delete the blade

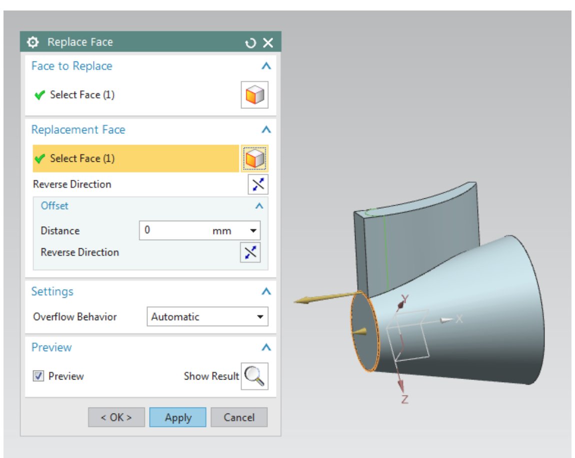
➢ Repeat the process and delete all except one blade. The part should look as shown below.

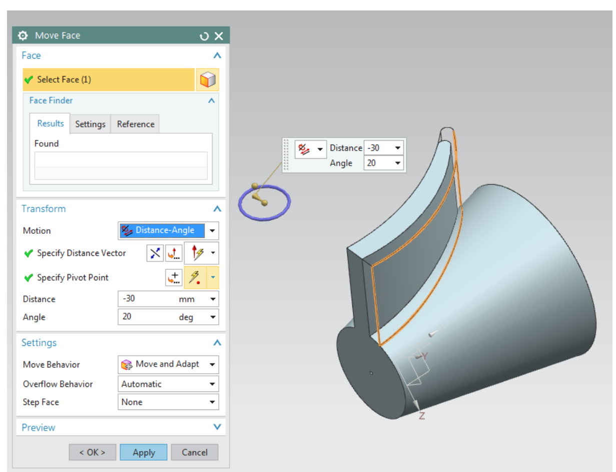




➢ Click **Replace Face** and select the end face of the blade with large blend radius as **Face to Replace** and select the flat surface of the cone with smaller diameter as the **Replacement Face** to delete the blade.

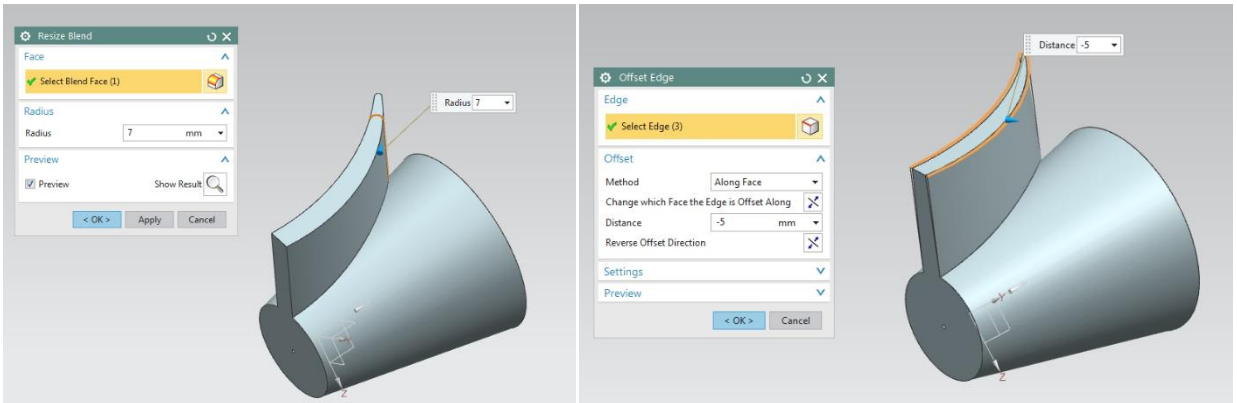
The part should look as shown below.



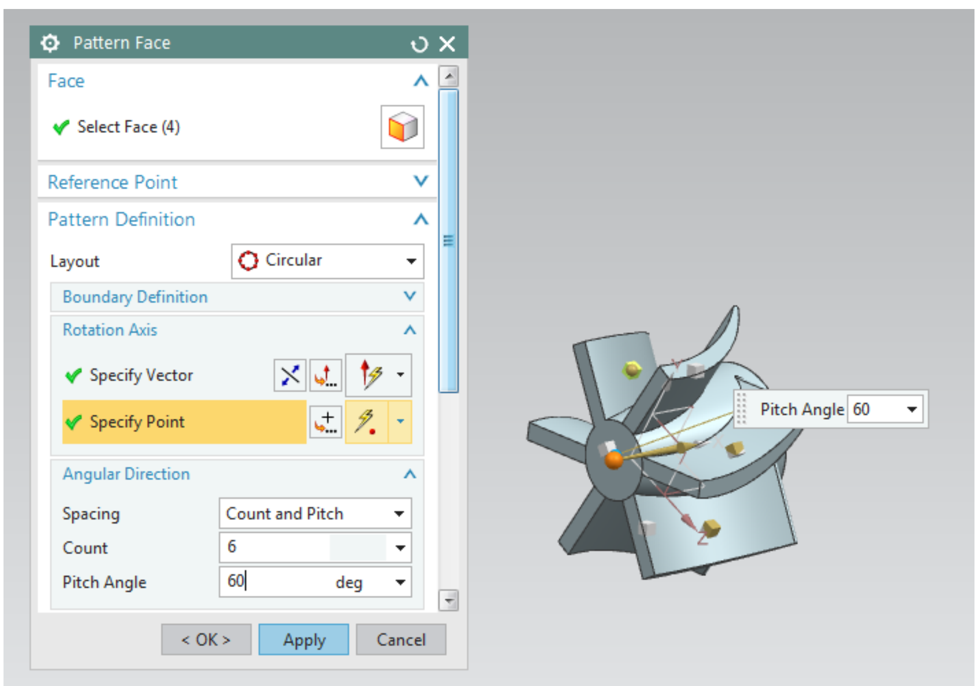


➢ Click **Move Face** and select one side of the blade and enter distance **-30** and angle **20** in the transform section

➢ Click **Resize Blend** and select the blended surface of the blade and enter radius as **7** mm to sharpen the end



- ➢ Click **Offset Edge** and select the top edge of the blade and choose the method along face and enter **-5** mm in the distance to offset the top surface of the blade
- ➢ Click **Pattern Face** and select four surfaces of the blade and choose **Circular Layout** and specify the conical axis as vector, center of the flat surface of the cone as point, count as **6** and pitch angle as **60** radius to pattern six blades.

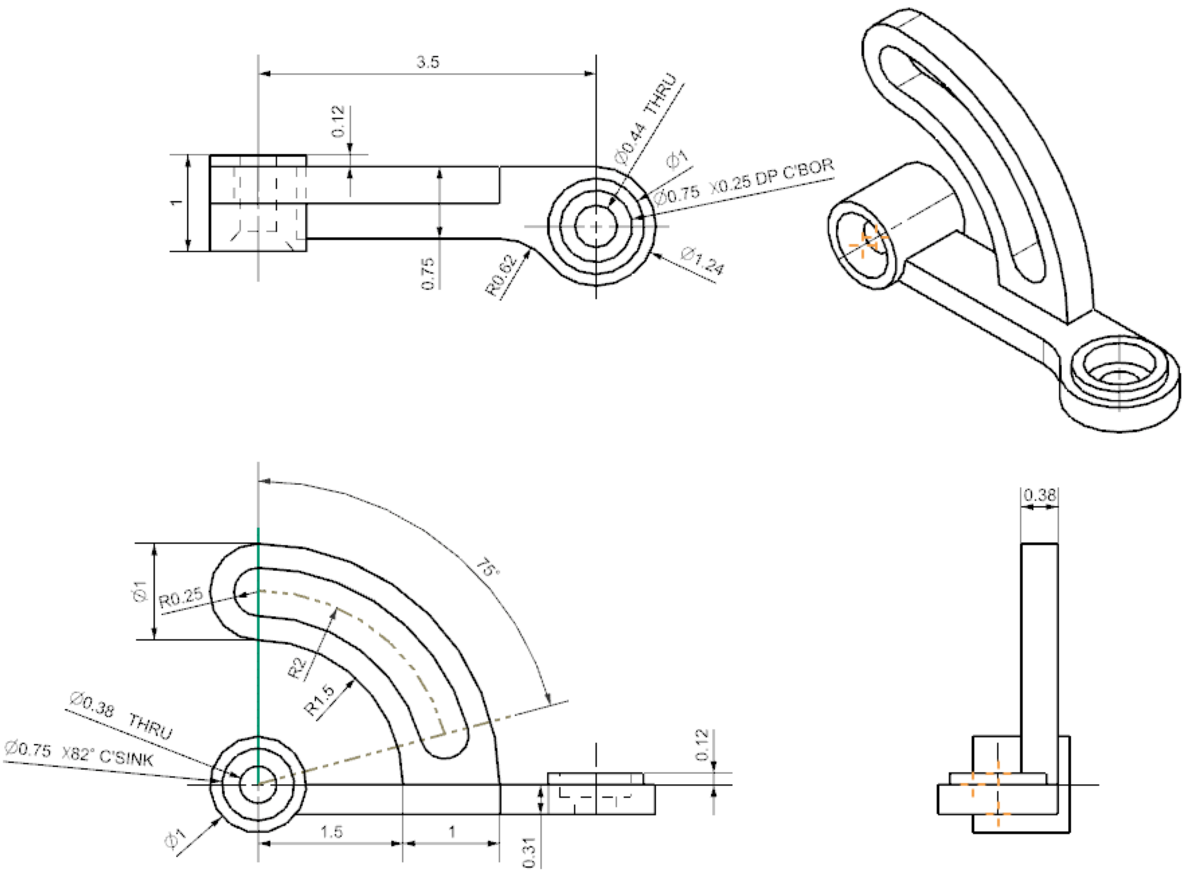


Therefore, it can be observed that a standard .stp file has been modified by increasing the number of blades and changing the blade profile. Similarly, the user can either modify any supported 3D model depending on the design need or create a new 3D model with synchronous modeling "push and pull" tools.

4.10 EXERCISES

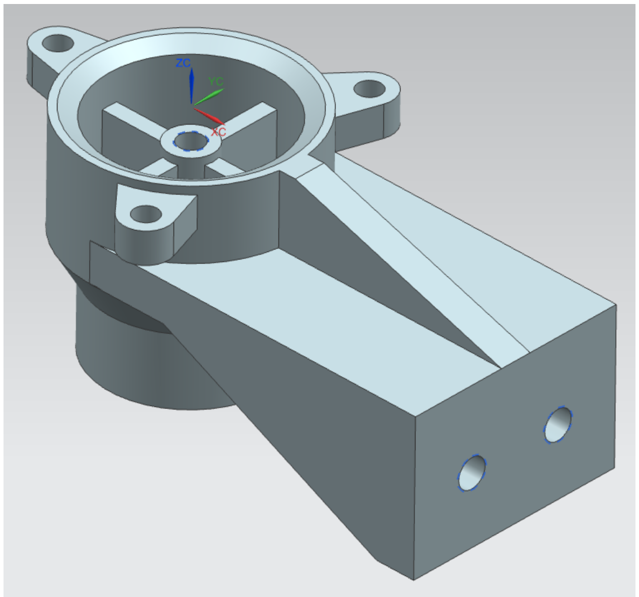
4.10.1 Rocker Arm

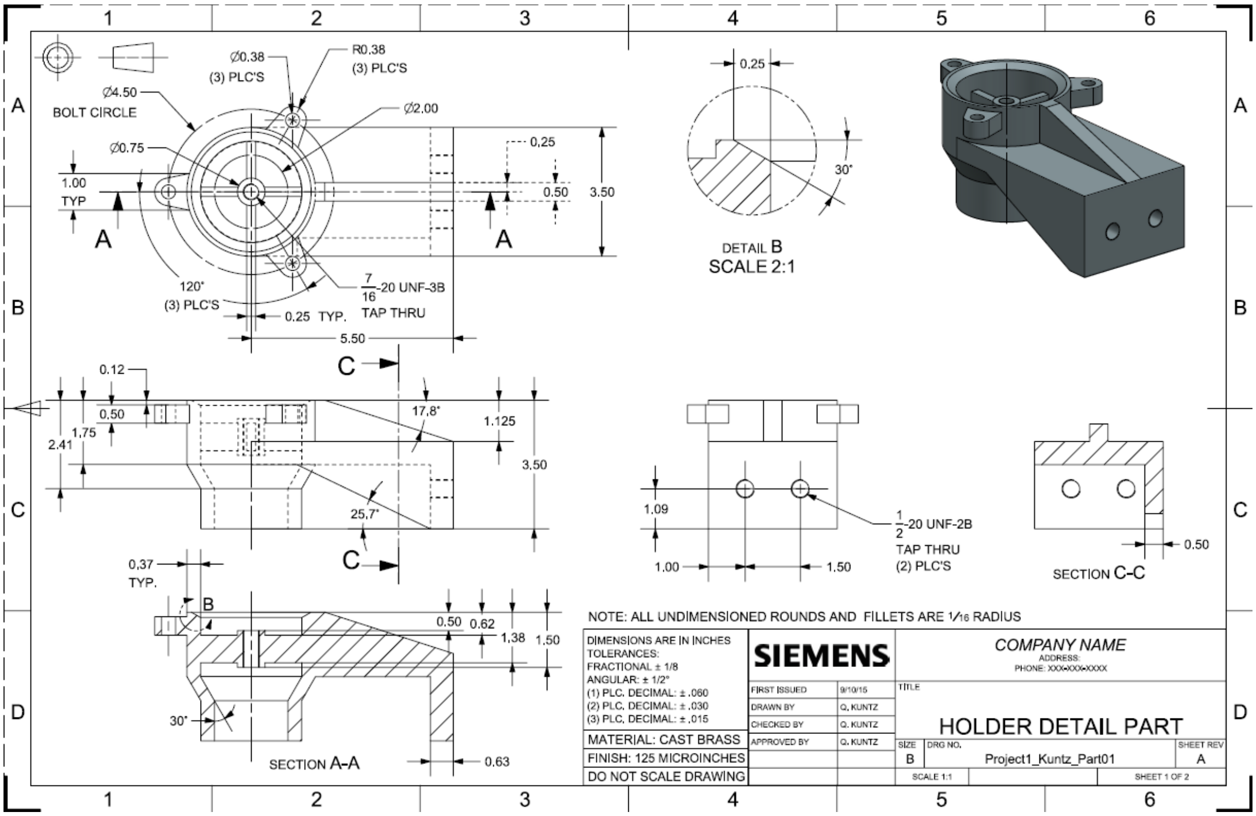
A rocker arm is an oscillating lever conveying radial movement from the cam lobe to linear movement at the poppet valve. Use the following drawings to model a rocker arm (dimensions are in inches).



4.10.2 Holder

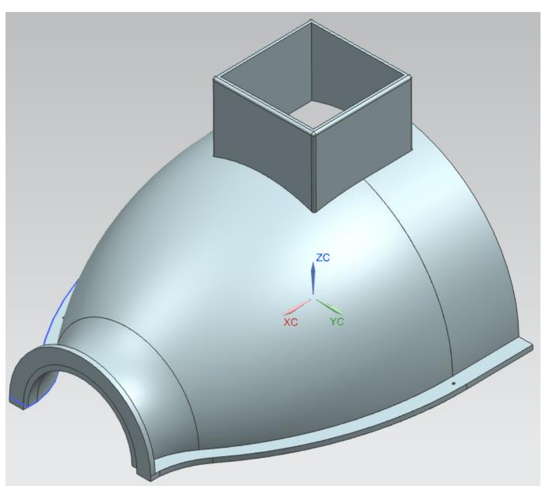
You have started to model a holder in chapter 3 by modeling a 2D sketching. Open your saved file and continue modeling this holder. Use the detailed drawing to replicate the part.





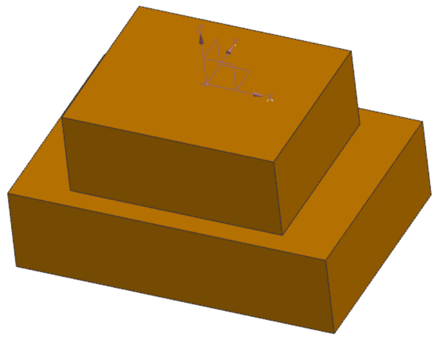
4.10.3 Impeller Upper Casing

Model the upper casing of the *Impeller* as shown below. The dimensions of the upper casing are the same as for the lower casing, which is described in Section 4.4 in detail. The dimensions for the manhole should be such that impeller blades can be seen and a hand can fit inside to clean the impeller.

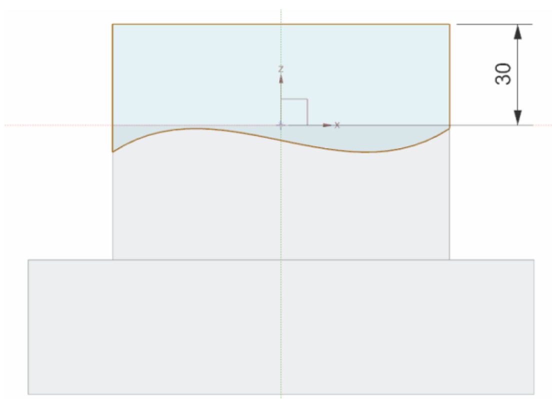


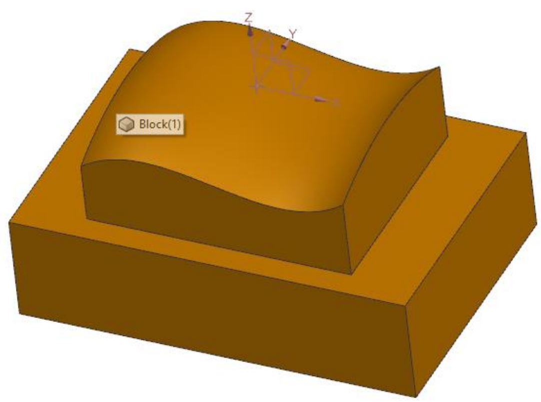
4.10.4 Die-Cavity

Model the following Die-Cavity part to be used in *Chapter 8 Manufacturing Module*. Create a new file *Die_cavity.prt* with units in *mm*. Create a rectangular *Block* with dimensions of *150, 100, 40* along *X, Y* and *Z*, respectively, with the point coordinates of *(-75, -50, -80)* for the *XC, YC and ZC* in Point Dialog. Create another block with the dimensions of *100, 80* and *40* along *X, Y* and *Z* and locate this block by entering the Point Dialog values of (-50, -40, -40). Unite this block with the first block.



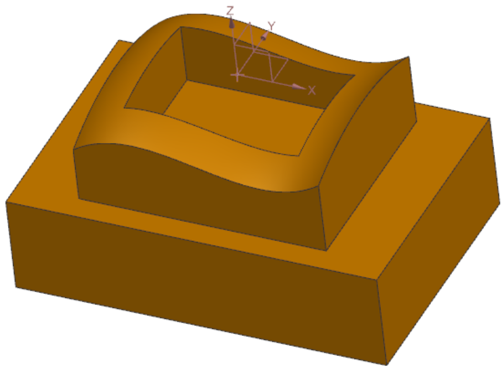
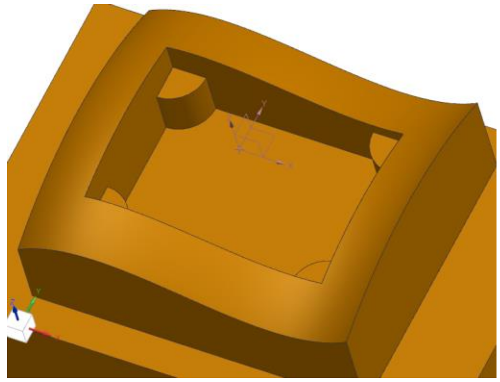
Create the sketch of a spline curve on X-Z plane as shown in the bottom-left figure. The spline curve is created with four points: (-50, 0, -8), (-25, 0, -1), (25, 0, -8), and (50, 0, -1).



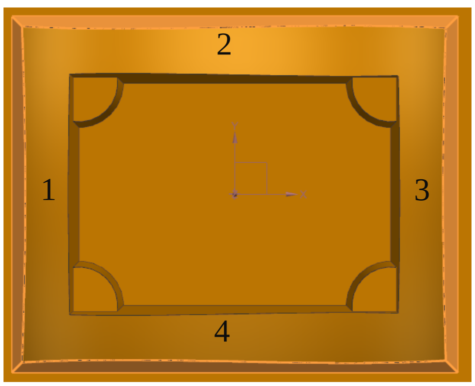


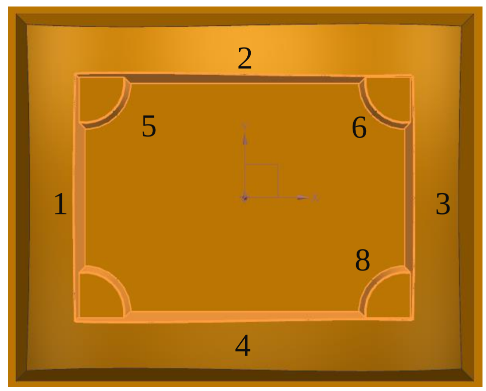
Revolve the sketch with the Specify Vector of XC and the Specify Point of (0, 0, -100), and use Boolean subtract to cut with the start angle of -45° and the end angle of 45° . The revolved model is shown on top-right figure.

Subtract a block of 70, 50, and 30 to create a huge cavity at the center with Point Dialog value of (-35, -25, -30). Create 4 cylinders at the inner corners of the cavity with 20 mm diameter and 15 mm height and Unite them with the model.

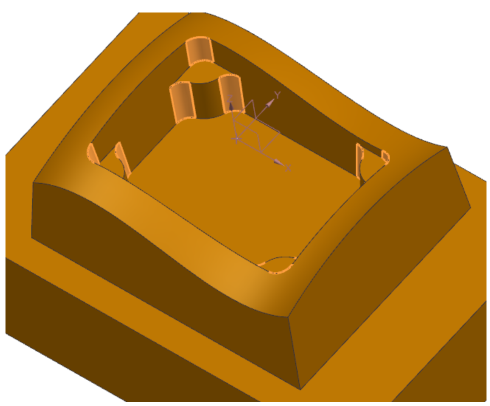
Add **Draft** feature on the outer 4 surfaces (left figure below) and inner 8 surfaces (right figure below) of the model with the angle of 5 degrees. The drafted model is shown below.



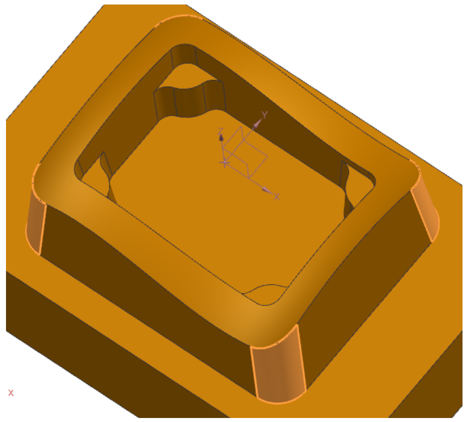


Add **Edge Blend** feature at the corners as shown below.

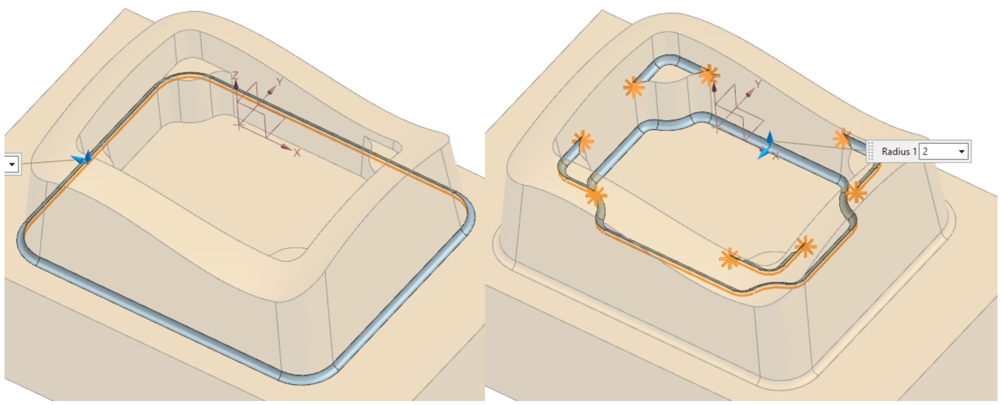
(1) 5 mm radii for the inner corners.



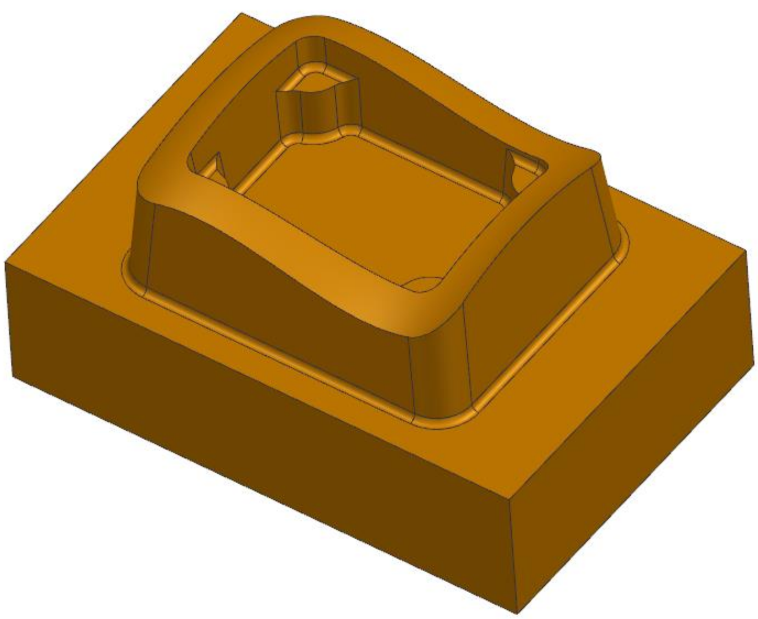
(2) 10 mm radii for the outer corners



(3) 2 mm radii for the outer (bottom left figure) and inner (bottom right figure) edges



The final *Die_cavity.prt* model is shown below.

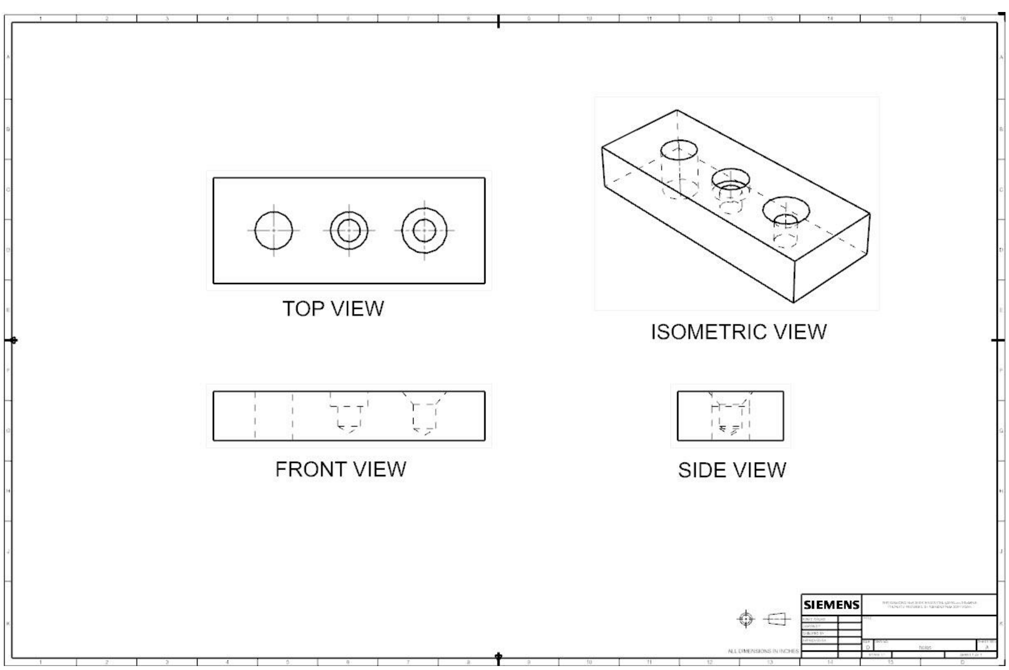


CHAPTER 5 – DRAFTING

The NX *Drafting* application lets you create 2D drawings, views, geometries, dimensions, and drafting annotations necessary for manufacturing as well as understanding of an industrial design. The goal of this chapter is to give the designer/draftsman enough knowledge of drafting tools to create a basic drawing of their design. The drafting application supports the drafting of engineering models in accordance with ANSI standards. After explaining the basics of the drafting application, we will go through a step-by-step approach for drafting some of the models created earlier.

5.1 OVERVIEW

The *Drafting* Application is designed to allow you produce and maintain industry standard engineering drawings directly from the 3D model or assembly part. Drawings created in the *Drafting* application are fully associative to the model and any changes made to the model are automatically (after a view update) reflected in the drawing. The *Drafting* application also offers a set of 2D drawing tools for 2D centric design and layout requirements. You can produce standalone 2D drawings. The *Drafting* Application is based on creating views from a solid model as illustrated below. *Drafting* makes it easy to create drawings with orthographic views, section views, imported view, auxiliary views, dimensions and other annotations.



Some of the useful features of the *Drafting Application* are:

1) After you choose the first view, the other orthographic views can be added and aligned with the click of a few buttons. (NX uses 3rd Angle Projection for view placement, this option can be changed to 1st angle if users need that option).

2) Each view is associated directly with the solid. Thus, when the solid is changed, the drawing will be updated directly along with the views and dimensions.

3) Drafting annotations (dimensions, labels, and symbols with leaders) are placed directly on the drawing and updated automatically when the solid is changed.

When you are modeling a part with multiple operations (e.g., drafting, assembly, CAM, etc.), master model concept can help you conducting these operations without adding loads on your computer system. It separates a CAD model into multiple files and reduce the file size. However, these files still linked to each other. Thus, you can speed up your modeling efficiency by applying the master model concept.

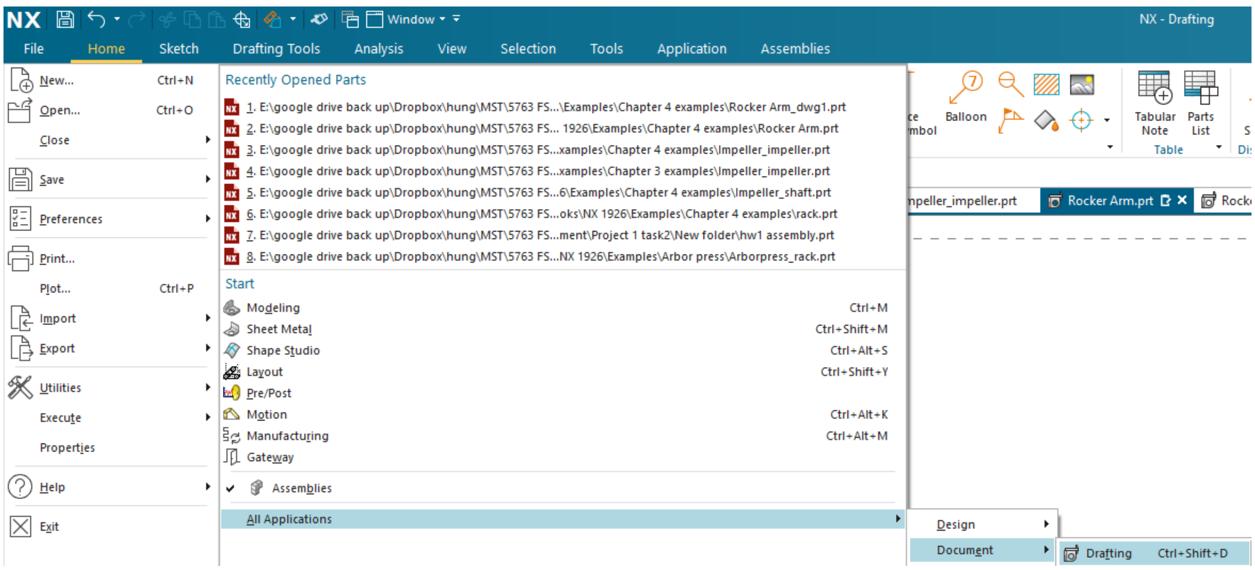
We will see how views are created and annotations are used and modified in the step-by-step examples.

5.2 CREATING A DRAWING

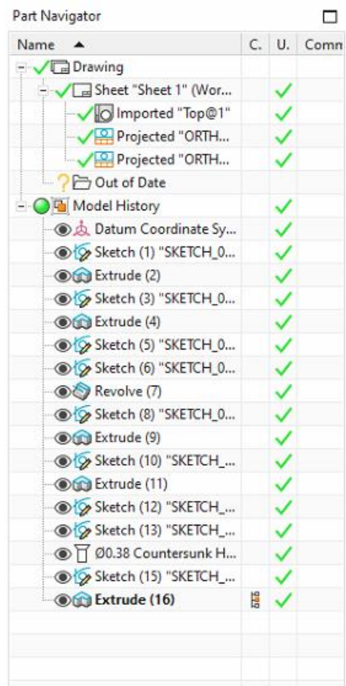
When creating a drawing for your model, two ways you can choose:

(1) Drawing stored within your NX part:

You can create a drawing in the same NX file by clicking **Drafting** in **File → All Applications → Document → Drafting**

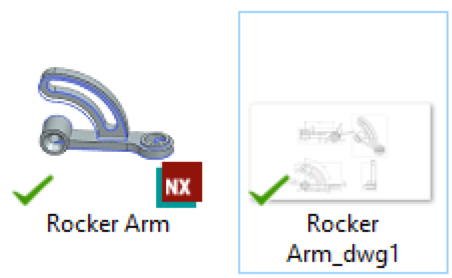


You will see your drawing under Part Navigator.

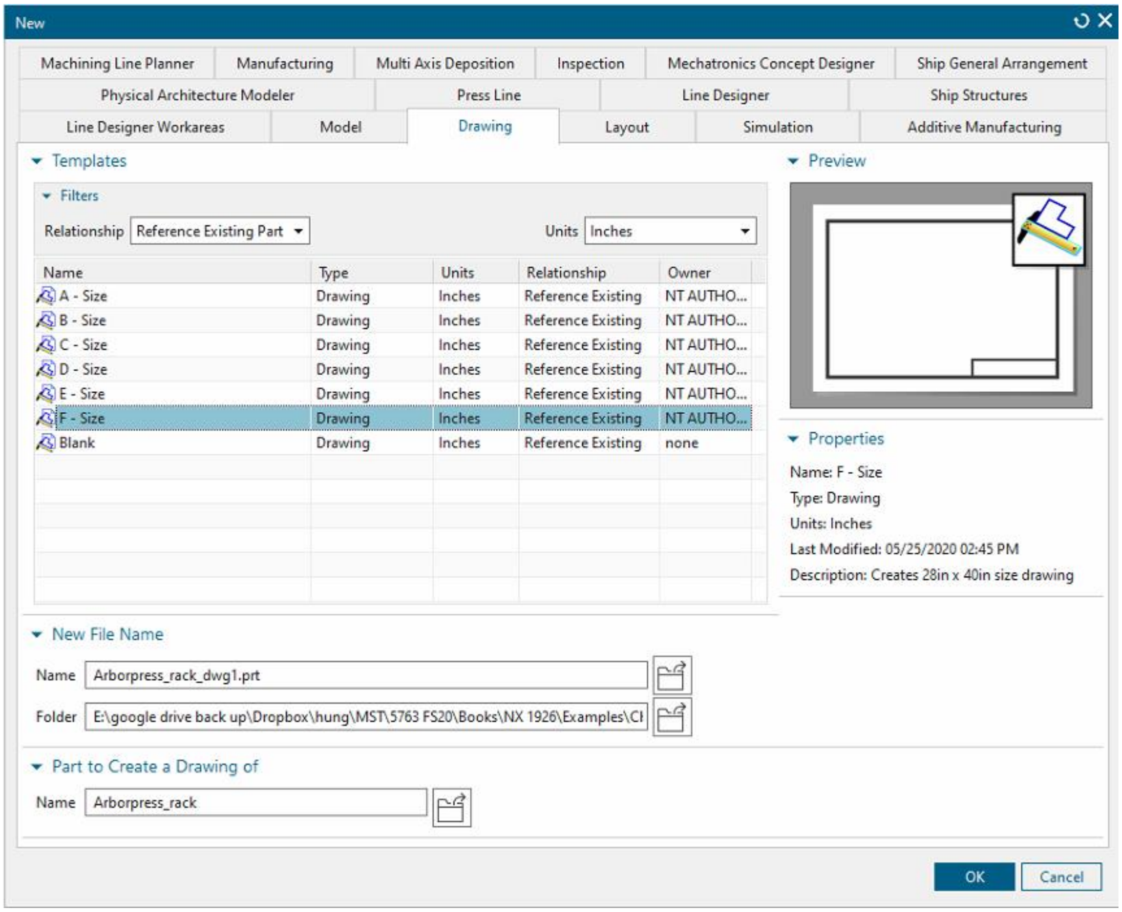


This way to create a drawing within the NX part is not recommended because it will increase the file size and the loading to the NX software and system when creating a complex part. So we will focus on explaining how to use alternative way to create drawings without slowing your system

(2) **Drawing stored in a separate file:** This method is based on the Master Model Concept to create a drawing in different file as shown in the right figure. Because it is recommended to use master model approach to generate your drawing to minimize your file size and system loading. Here we will briefly teach how to use Master Model Method to create the drawing for your NX model.



- ➢ Open the file **Arborpress_rack.prt**
- ➢ From the NX Interface, choose **File** → **NEW** → select **Drawing Tab**



- Select **a drawing size**

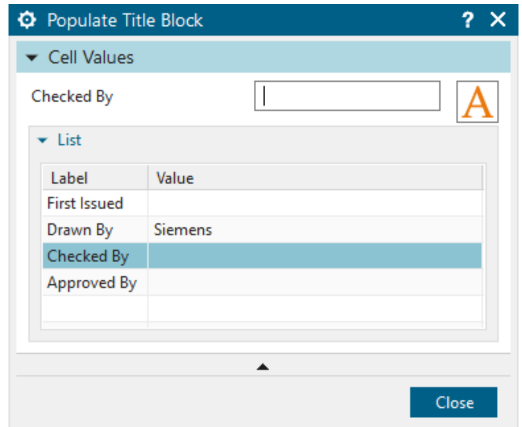
  Size allows you to choose the size of the *Sheet*. There are standard *Templates* that you can create for frequent use depending upon the company standards. There are several *Standard* sized *Sheets* available for you. You can also define a *Custom* sized sheet in case your drawings do not fit into a standard sized sheet. You can use *Preview* to show the overall design of the *Template.*

- **Unit**

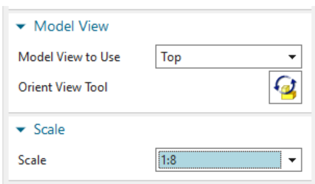
  Units follow the default units of the parent 3-D model. However, if you need the size of drawing is different from your NX model, you can change units here.

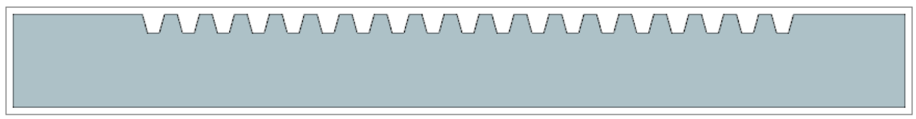
- click **OK**

- **Populate Title Block** will show up first to let you edit the information of your drawing.



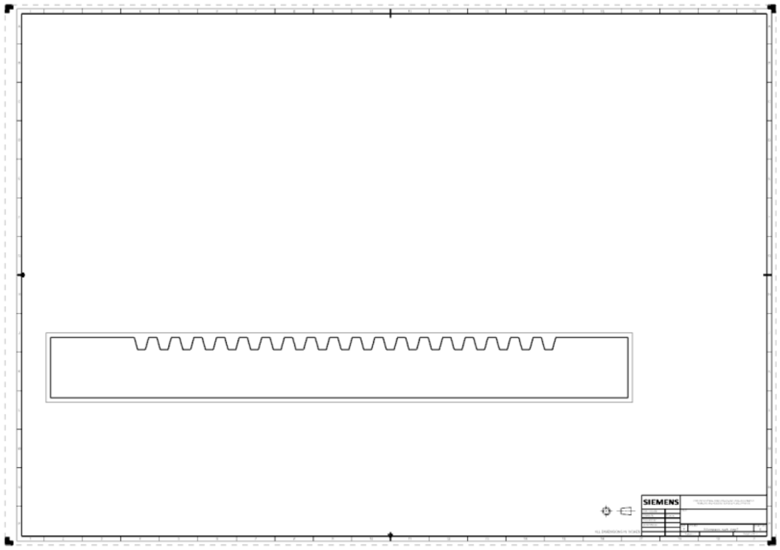
- **Base View** could let you import your drawing. If your part is too big or too small, you can change the **scale** to fit your drawing size. You can specify the model view you want to firstly import to your drawing. The setting for **Arborpress_rack** is showing below



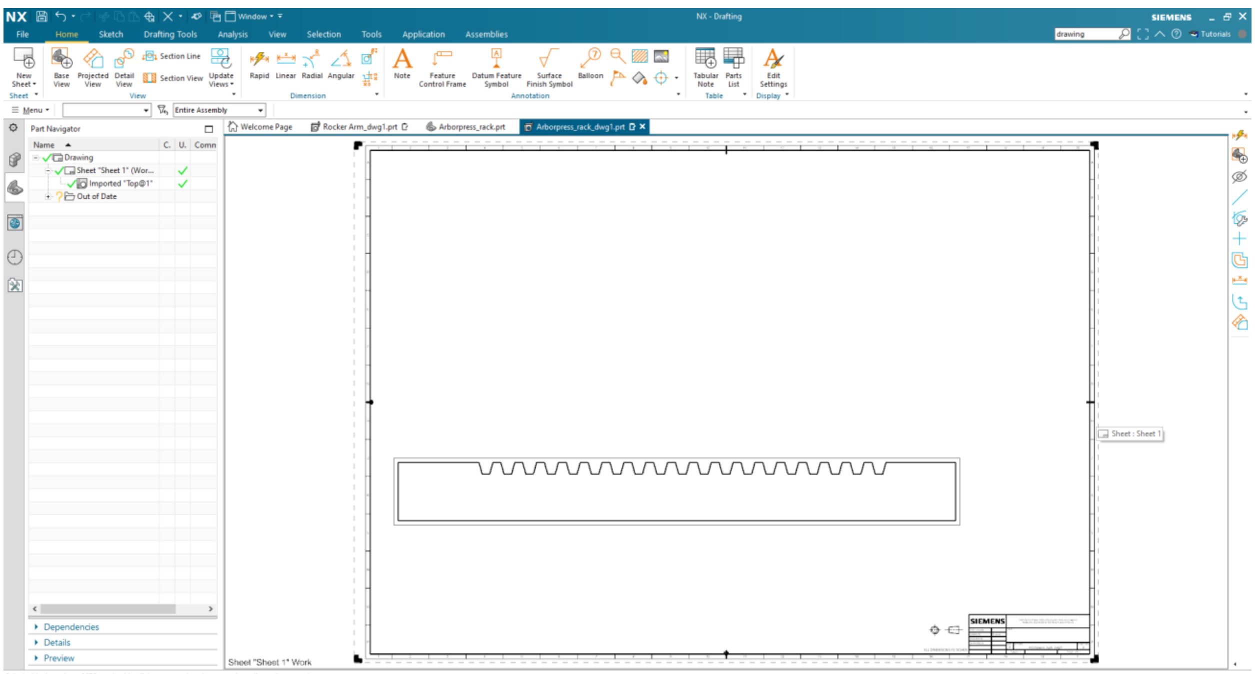


- **OK**

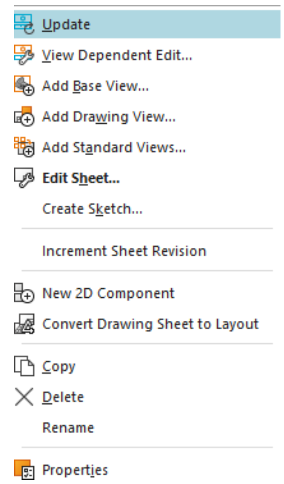
- Import **top view** to your drawing like figure below



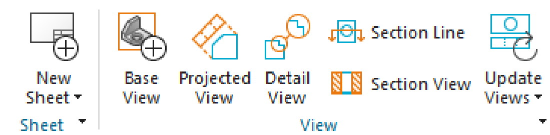
Let us first look at the *Drafting Application Interface*.



Although the **part navigator** only shows the drawing and does not see your part model history in this drawing file, your model file is linked to this drawing. That is to say, when you revised your model file, this drawing will be revised based on your new model by right click and choose **Update**. This is the Master Model Concept to minimize the content and size of a single file.

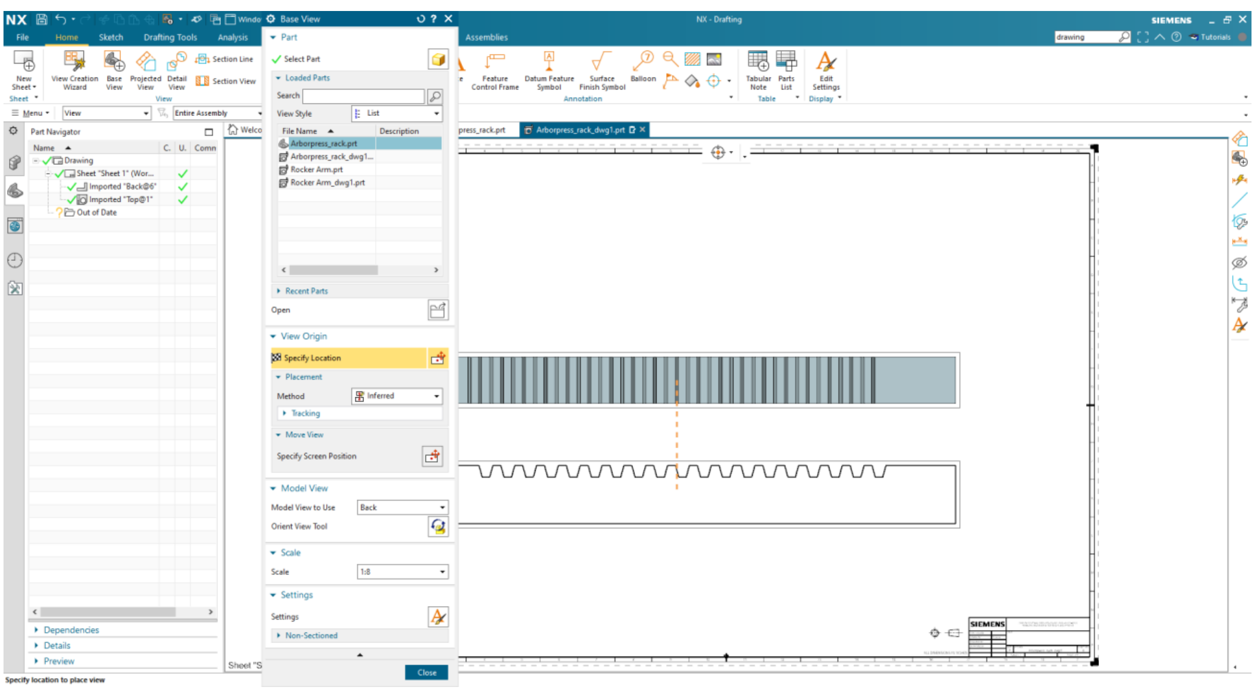


- Choose **Insert** → **View** → **Base** or click on **Base View** in the **View Group**



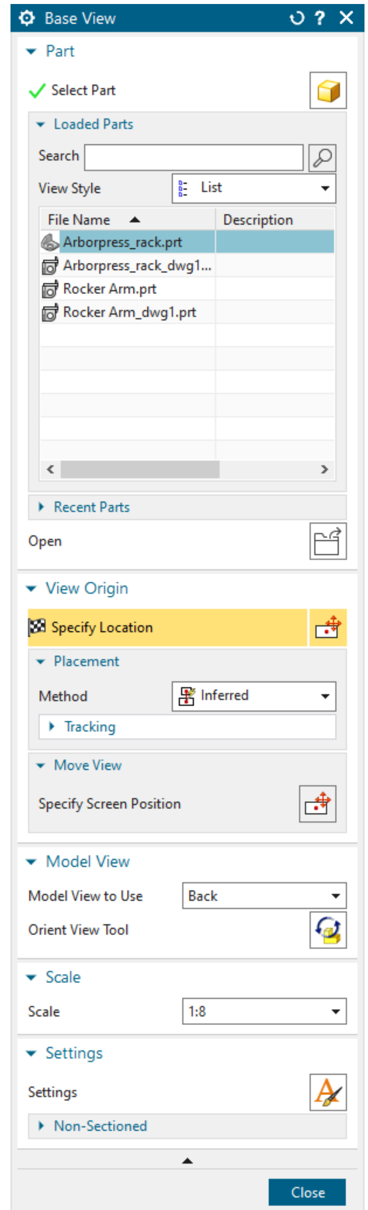
The *Base View* dialog box with the options of the **View** and the **Scale** will show up along with a floating drawing of the object.

- Choose the **View** to be **Back**
- Choose the **Scale** to **1:8**



You can find the *Back View* projection on the screen. You can move the mouse cursor on the screen and click on the place where you want the view.

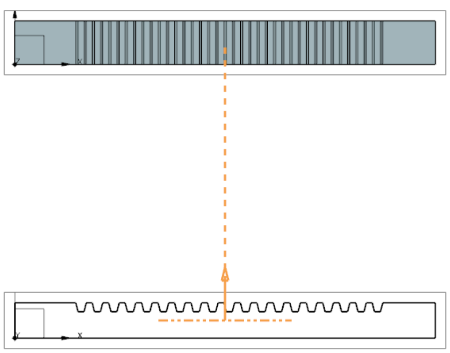
Once you set the *Back View* another dialog box will pop-up asking you to set the other views at any location on the screen within the *Sheet Boundary*.



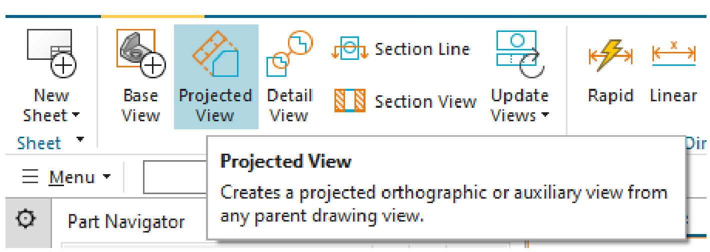
You can find different views by moving the cursor around the first view. If you want to add any orthographic views after closing this file or changing to other command modes

- Choose **Insert** → **View** → **Projected View** or choose **Projected View**  icon from the **View** group

Now, let us create all the other orthographic projected views and click on the screen at the desired position.



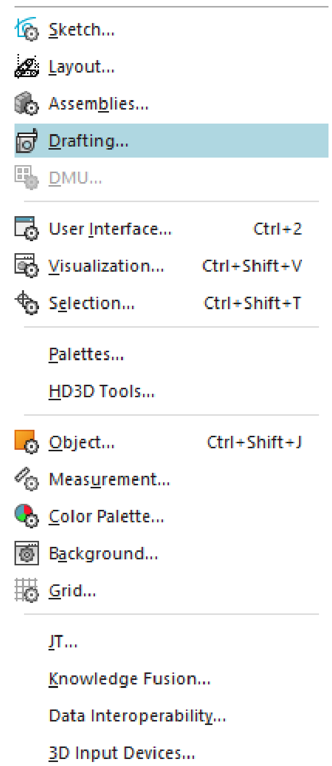
- In case you have closed the **Projected View** dialog box you can reopen it by clicking on the **Projected View** icon in the **View Group**



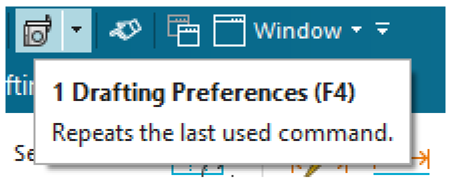
- Move the cursor and click to get the other views

- ➢ Click **Close** on the **Projected View** dialog box or press **<Esc>** key on the keyboard to close the window

Before creating the dimensions, let us remove the borders in each view as it adds to the confusion with the entity lines.

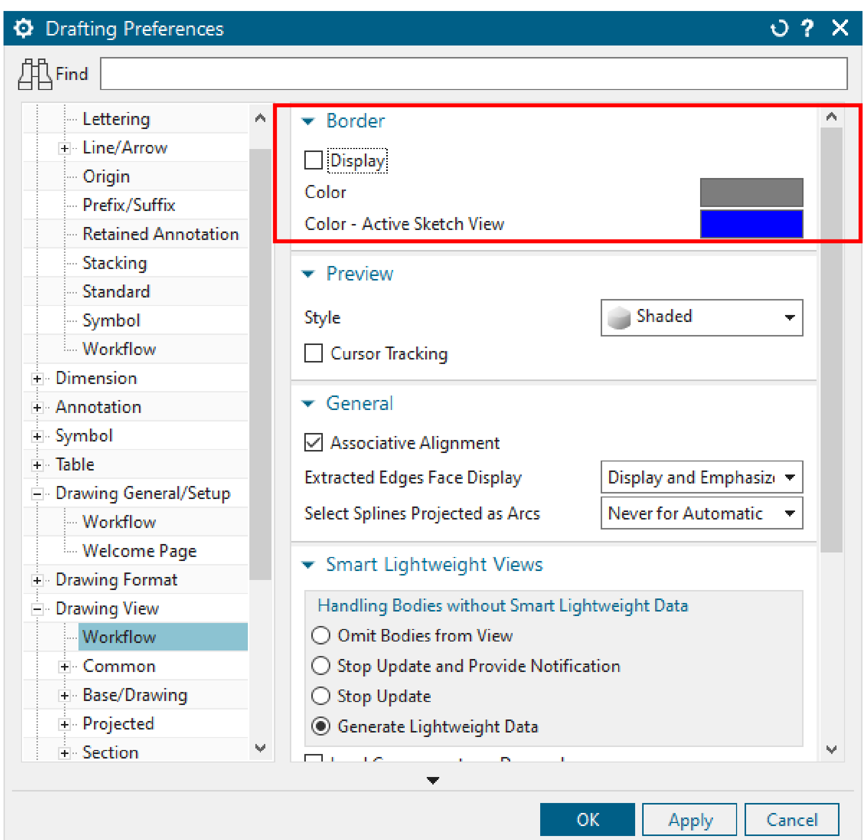


- ➢ Choose **Menu** → **Preferences** → **Drafting** or click on icon in the **Quick Access** toolbar to find the **Drafting Preferences**

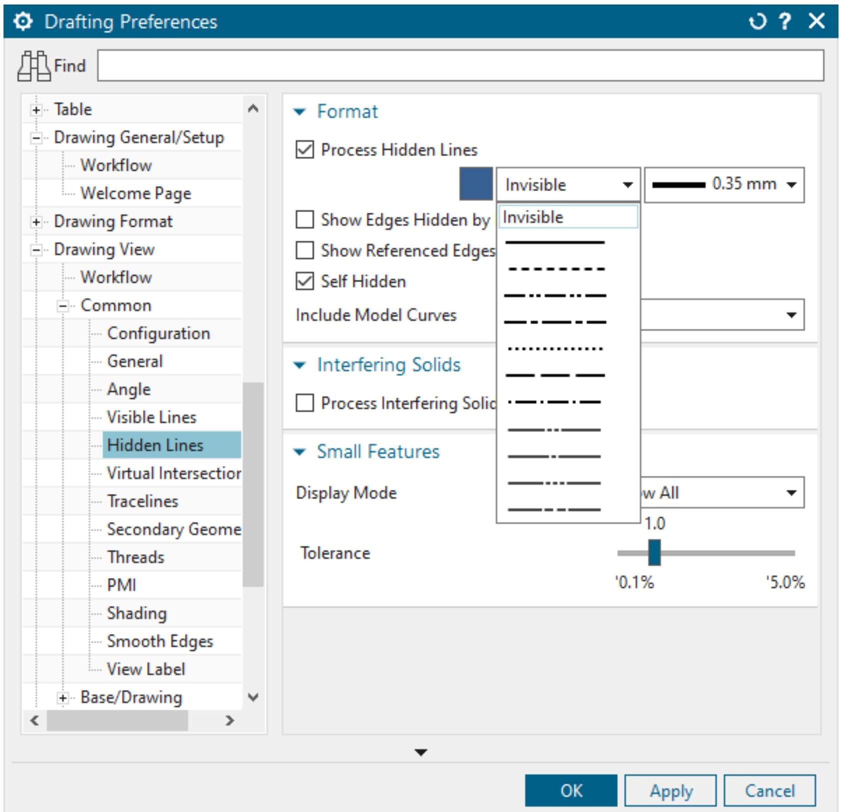


The *Drafting Preferences* window will pop up.

- ➢ Click on the **Drawing VIEW** tab button
- ➢ Uncheck the **Tick** mark on the **Display** in **Borders** as shown in the figure below and click **OK**

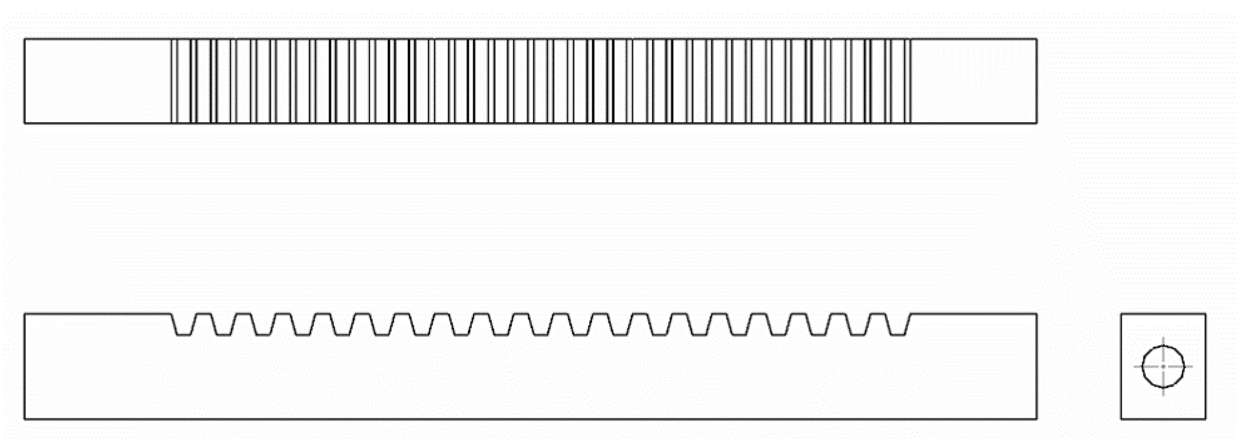


There are many other options like number of decimal places, hidden lines, angles, and threads that you can find here. For example, you can find options for hidden lines in *Drafting* Preferences → *Drawing View* → *Common* → *Hidden Lines* and change the format of *hidden lines* based on your preference.



5.3 DRAWING DIMENSIONING

Now let's move on to create the dimensions for these views.

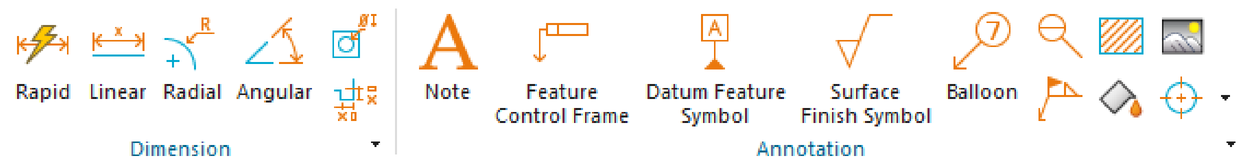


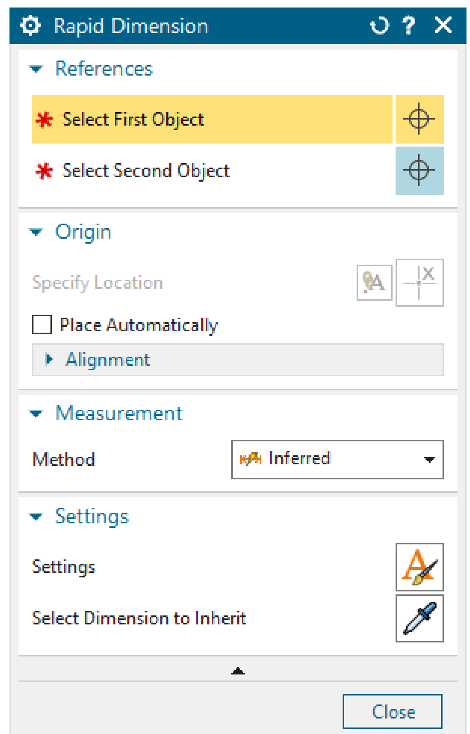
The dimensions can be inserted by either of the two ways as described below:

- Choose **Menu → Insert → Dimension**

OR

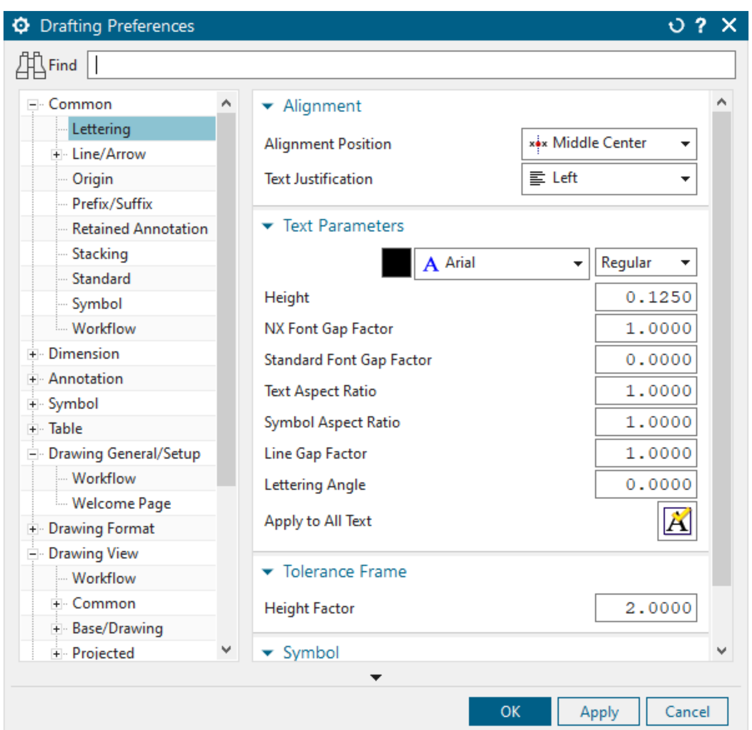
- Click on the **Dimension Toolbar** as shown in the following figure





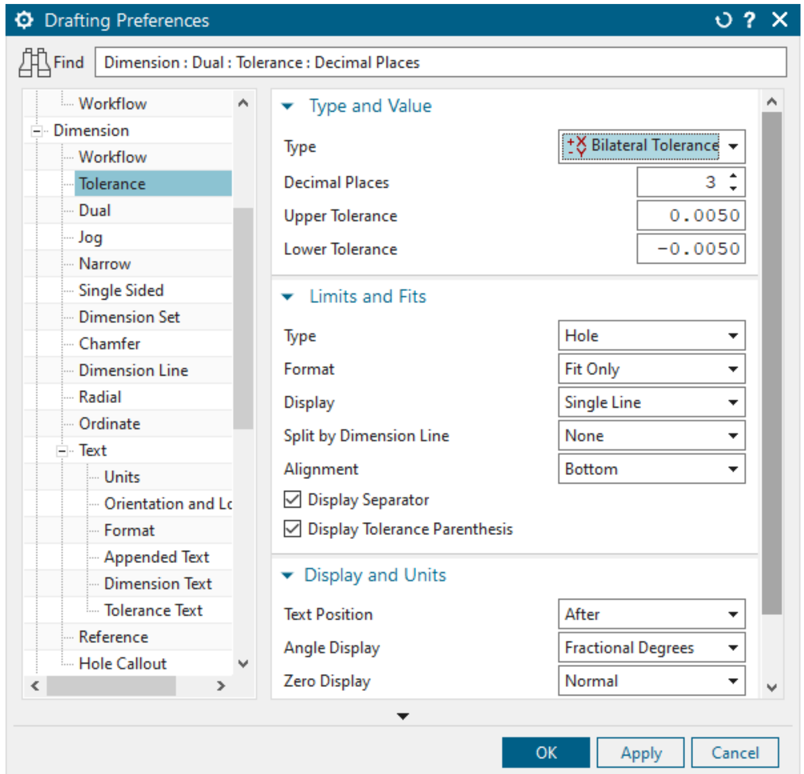
- Click on **Points** and **Edges**, move the mouse and click on the appropriate location to draw dimensions
- Choose **Menu → Preferences → Drafting**

Here you will be able to modify the settings for dimensioning. A dialog appears as shown below.

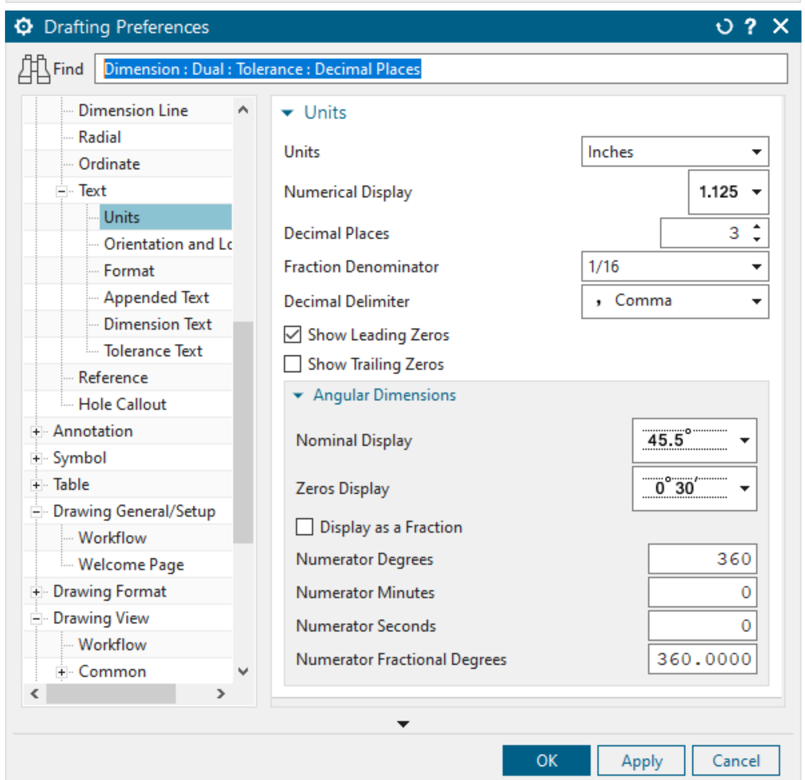


The first list is for *Lettering***.** This allows the user to justify and select the frame size. In the *Line/Arrow* section, you can vary the thickness of the arrow line, arrow head, angle format etc.

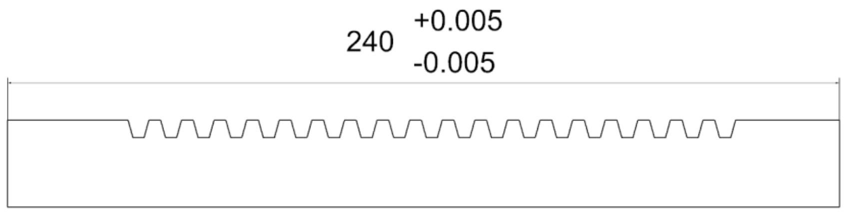
The most important section is the *Tolerance* list. Here you can vary the tolerance to the designed value.



The type of display, precision required for the digits and other similar options can be modified here. The next icon is the *Text* option, which you can use to edit the units, text style, font and other text related aspects.



- On the first view **(Front View**) that you created, click on the top left corner of the rack and then on the top right corner

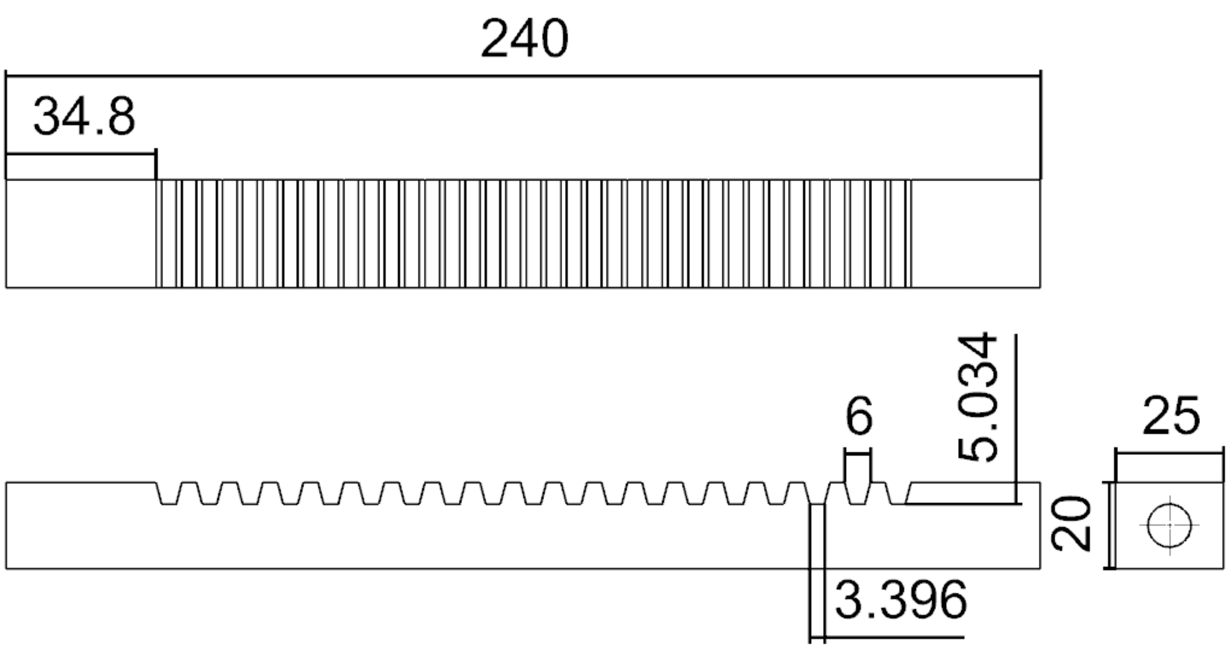


The dimension that represents the distance between these points will appear. You can put the location of the dimension by moving the mouse on the screen. Whenever you place your views in the *Sheet* take into consideration that you will be placing the dimensions around it.

- To set the dimension onto the drawing sheet, place the dimension well above the view as shown and click the left mouse button

Even after creating the dimension, you can edit the properties of the dimensions.

- Right-click on the dimension you just created and choose **Settings** or **Edit Display**
- You can modify font, color, style and other finer details here
- Give dimensions to all other views as shown in the following figure

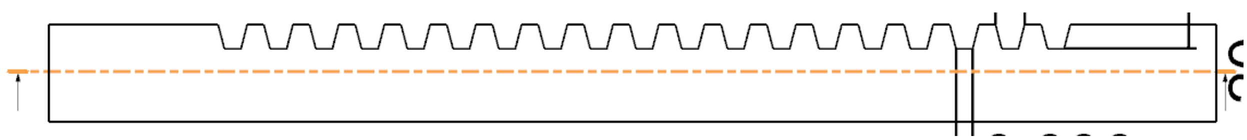


5.4 SECTIONAL VIEW

Let us create a *Sectional View* for the same part to show the depth and profile of the hole.

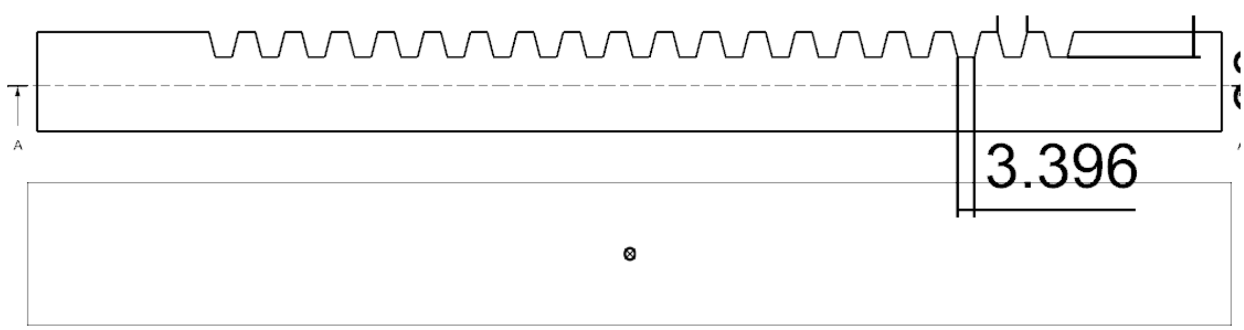
- Choose **Insert** → **View** → **Section** or click the **View Section** icon 

from the **View** group in the ribbon bar
- Click on the bottom of the **Base View** as shown in the figure. This will show a **Phantom Line** with two **Arrow** marks for the direction of the Section plane (orange dashed line with arrows pointing upwards).

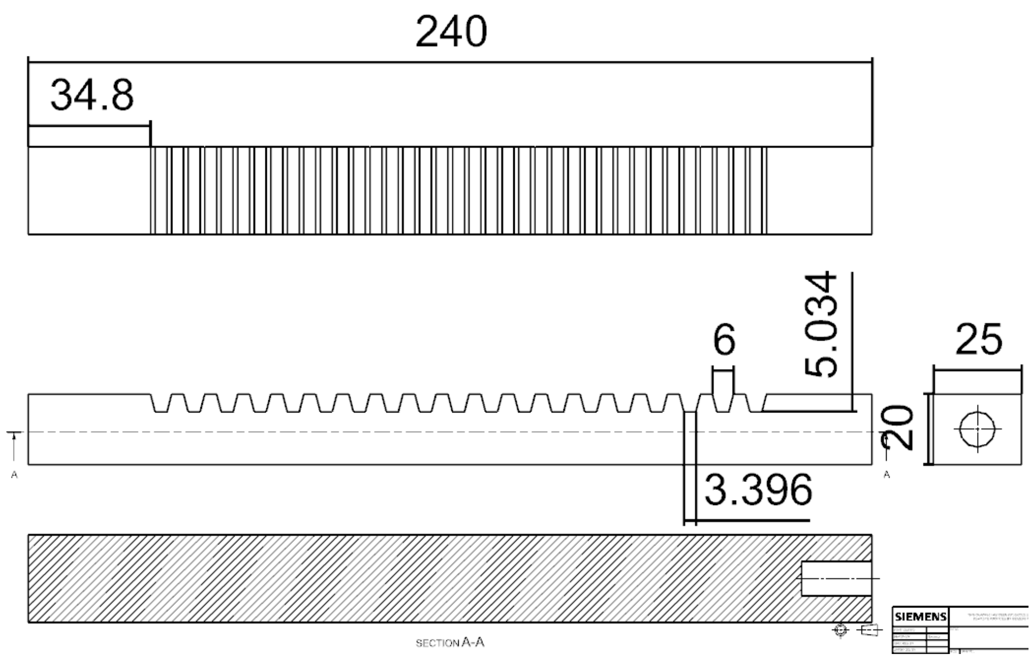


- Click on the middle of the **View** as shown. This will fix the position of the sectional line (Section Plane)

Now move the cursor around the view to get the direction of the **Plane of Section**. Keep the arrow pointing vertically upwards and drag the sectional view to the bottom of the **Base View**.



Adjust the positions of dimensions if they are interfering. The final drawing sheet should look like the one shown in the following figure.



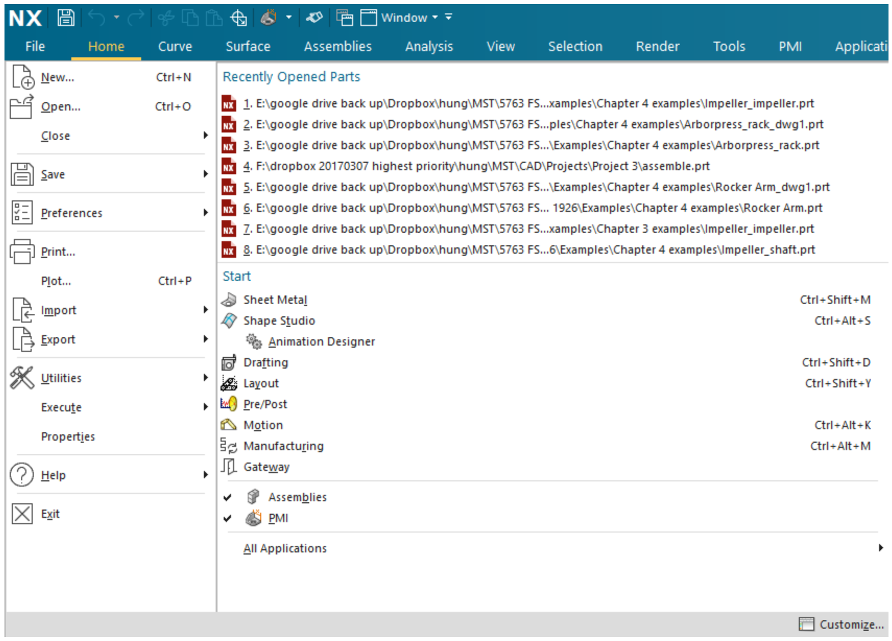
Save and close your model.

5.5 PRODUCT AND MANUFACTURING INFORMATION

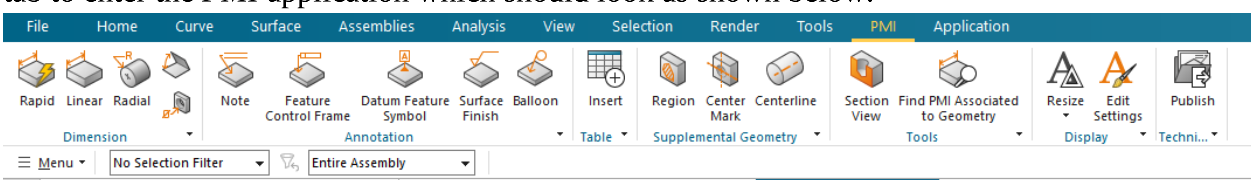
Product and Manufacturing Information (PMI) is one of the important applications in NX which provides annotation tools used to document products in a 3D environment. PMI application includes a comprehensive 3D annotation environment that enables design teams to share details such as Geometric Dimensioning and Tolerancing (GD&T), surface finish, welding information, material specifications, comments, government security information or proprietary information, etc. directly to the 3D model. PMI complies with industry standards for 3D product definition and therefore product teams working on collaborative projects would use 3D models as a legitimate method for fully documenting product and manufacturing information.

In the below example, we will open a NX part file, create PMI dimension entities and comments on the 3D model in the PMI application and learn how to inherit the dimensions and comments to the *Drafting* application. This is only for the purpose of illustration.

- Open the file **Impeller_impeller.prt**
- From the NX interface, choose **File** → **PMI** (turn on the check mark)



This should create an additional tab PMI in between *Tools* and *Application* tabs. Select the *PMI* tab to enter the *PMI* application which should look as shown below.



The ribbon bar in this mode would have the *Dimension*, *Annotation*, *Custom Symbols*, *Supplemental Geometry*, *Specialized* and *Security Marking* groups. Each group has several options which could help describe the modeled 3D part. For example, dimensioning options in *Dimension* group, *Surface Finish* and *Notes* in *Annotation* group.

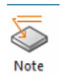
- Click **Rapid** icon 

- Select the end surfaces of the impeller as first and second objects to insert the linear dimension or click the **Linear** icon 

 to perform the same task
- Click the **Radial** icon 

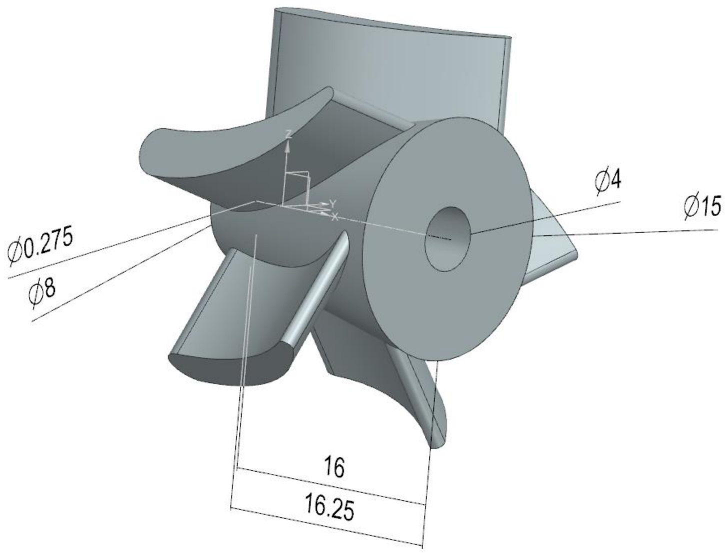
 in the **Dimension** group to insert the dimensions of the holes and curved surfaces on the impeller
- Click the **Centerline** icon 

 in the **Supplemental Geometry** group and select the inner surface of the impeller to insert the centerline for the part

➢ Click the **Note** icon  in **Annotation** group to provide any comments or **Surface Finish** icon, select the object, location of text and leader line to insert the specific surface finish details, if required

The *Trimetric* view of the impeller after PMI dimensioning would look as shown below.

Five impeller blades

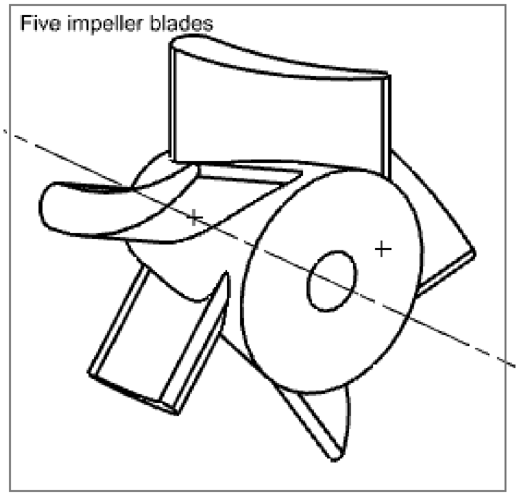


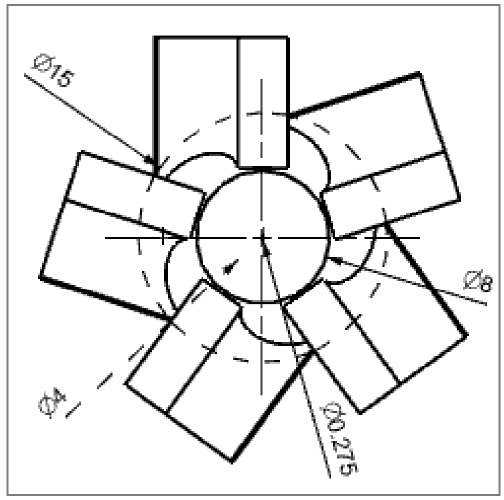
➢ Save the file, select **Application** tab and click on **Drafting** icon in the ribbon bar

➢ Follow the similar procedures explained in the previous section to create the **Drawing** sheet for the 3D part

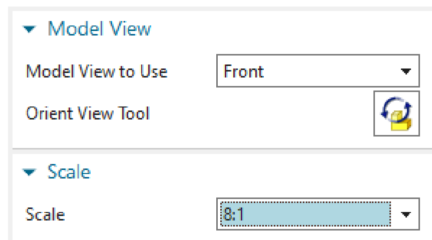
During the creation of the sheet, in the *View Creation Wizard*, select the *Inherit PMI* option, and select the *Aligned to Drawing (Entire Part)* and check the *Inherit PMI onto Drawing* option. This would inherit the dimensions of the 3D model and show on the drawing sheet including the comments as shown below. The user has to select the appropriate views to reflect the dimensions on the drawing sheet.

Siemens NX also enable legacy drawings with views and dimensions to be [converted back into 3D PMI entities](), users can select from the options of Drawing, Sheet, View, and Annotations, then converting drafting entities into PMI 3D entities.



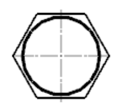


5.6 EXAMPLE

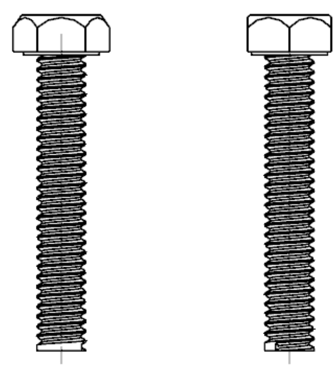


- Open the model **Impeller_hexa-bolt.prt**
- Choose **File** → **NEW** → **Drawing**
- On the **Sheet** window, select sheet **E -Size (34 X 44)** and change the **Scale** value to **8.0 : 1.0**
- Click **OK**
- Choose **Insert** → **View** → **Base View** or click the **Base View** icon

- Add the **Front** view by repeating the same procedure explained in the last example
- Add the **Orthographic Views** including the **Right** view and **Top** view



- Choose **Preferences** → **Drafting**
- Uncheck the box next to **Display Borders** under **Drawing View** Tab



The screen will have the following three views.

To visualize the hidden lines,

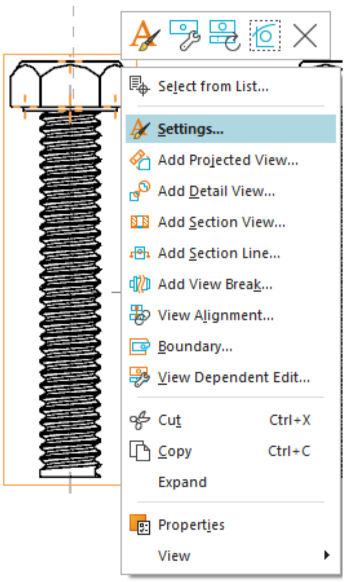
- Choose **Preferences** → **Drafting** → **Drawing View**

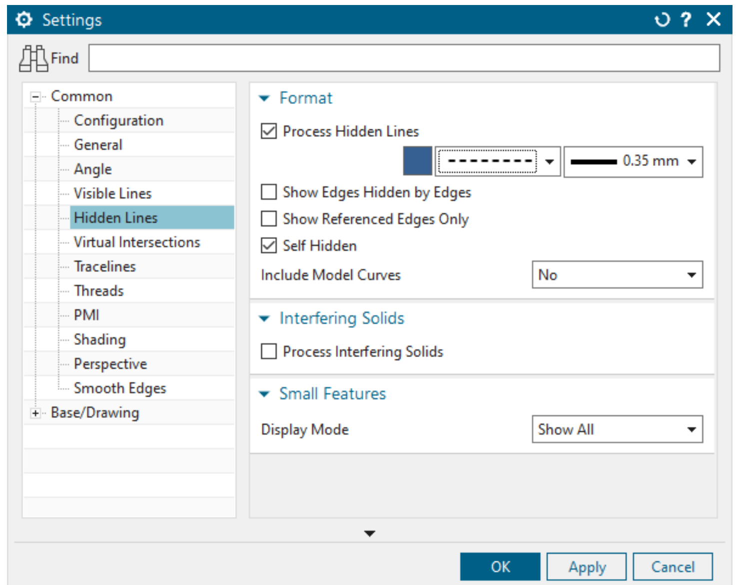
OR

- Select the views, right-click and choose **Settings** as shown below

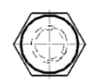
A window will pop up with various options pertaining to the views.

- Click on the **Hidden Lines** tab
- Change **Process Hidden Lines** to **Dashed Lines** as shown below and click **OK**





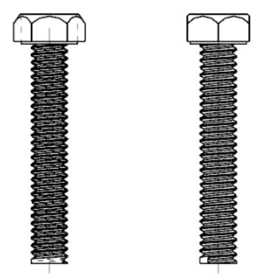
You can see the hidden lines as shown in the picture on the right.

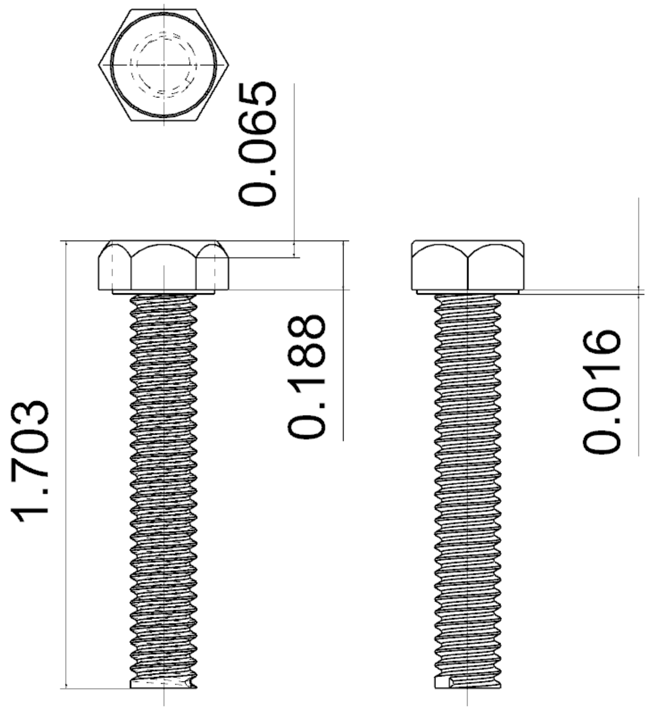


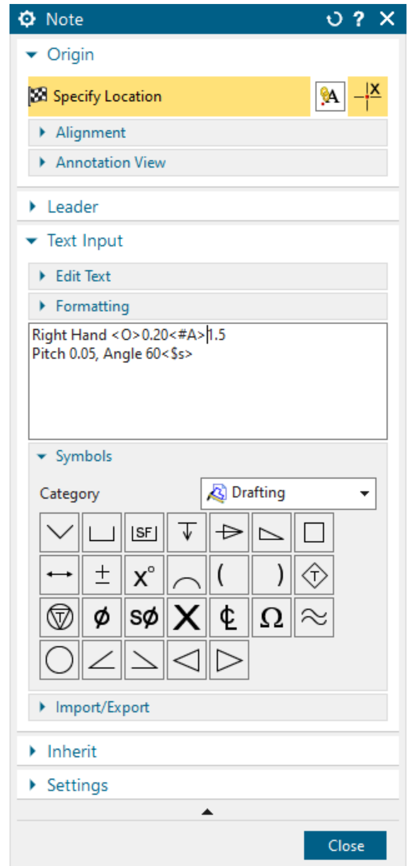
Now we will proceed to dimensioning.

- Choose **Insert** → **Dimensions** → **Linear** or click the **Linear Dimension**  icon in the **Dimension** group

- Give vertical dimensions to all the distances shown below





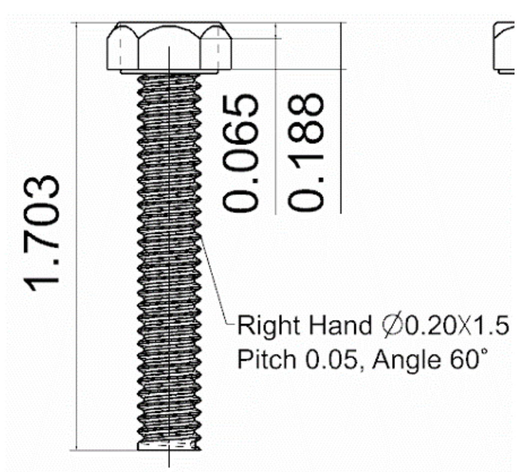


For the threading, we will use a leader line.

- ➢ Click on the **Note** icon  shown in the Toolbar
- ➢ In the **Note** window that opens, enter the following text. You can find Ø and the degree symbol on the **Symbols** tab

Right Hand Ø 0.20 x 1.5
Pitch 0.05, Angle 60⁰

- ➢ Click on the threaded shaft in the side view, hold the mouse and drag the **Leader** line next to the view. Let go of the mouse and click again to place the text.



- Close the **Annotation Editor**

Since the height of the *Lettering* is small, we will enlarge the character size as well as the arrow size.

- Right-click on the **Leader** and select **Settings**
- Click on the **Lettering** tab
- In the **Text Parameter** section, increase **Height** to make the leader legible
- Click on the **Line/Arrow** tab
- In the **Format** section, increase the **Length** of the **Leader**

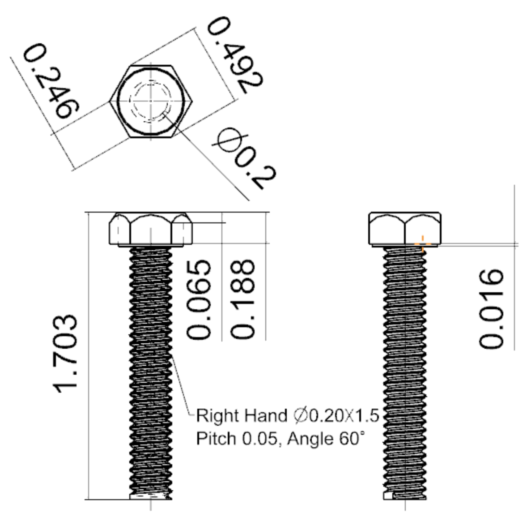
Now we will add additional dimensions and views.

- Choose **Insert → Dimensions → Radial** or click the **Radial Dimension** icon 

in the **Dimension** group
- Click the circle of the bolt in the top view to give the diameter dimension
- Click **Insert → View → Base View** of click the **Base View** icon 

- Select the **Isometric** view and place the view somewhere on the screen

The final drawing is shown below. Remember to save.



5.7 EXERCISE

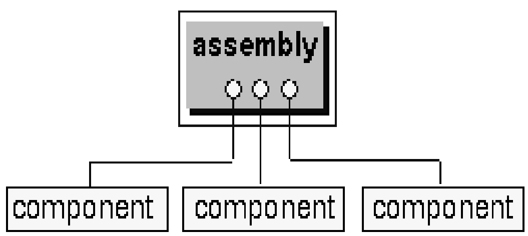
Perform *Drafting* and annotate dimensions for the parts that you modeled in chapter 4.

CHAPTER 6 – ASSEMBLY MODELING

Every day, we see many examples of components that are assembled together into one model such as bicycles, cars, and computers. All of these products were created by designing and manufacturing individual parts and then fitting them together. The designers who create them have to carefully plan each part so that they all fit together perfectly in order to perform the desired function.

In this chapter, you will learn two kinds of approaches used in *Assembly* modeling. We will practice assembly modeling using the impeller assembly as an example. Some parts of this assembly have already been modeled in earlier chapters.

NX *Assembly* is a part file that contains the individual parts. They are added to the part file in such a way that the parts are virtually in the assembly and linked to the original part. This eliminates the need for creating separate memory space for the individual parts in the computer. All the parts are selectable and can be used in the design process for information and mating to ensure a perfect fit as intended by the designers. The following figure shows how components are added to make an assembly.



6.1 TERMINOLOGY

Assembly

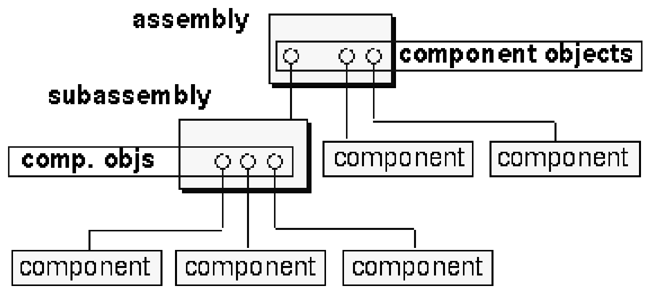
An assembly is a collection of pointers to piece parts and/or subassemblies. An assembly is a part file, which contains component objects.

Component Object

A component object is a non-geometric pointer to the part file that contains the component geometry. Component object stores information such as the *Layer*, *Color*, *Reference set*, *position data for component relative to assembly* and *path of the component part on file system.*

Component Part

A component part is a part file pointed to by a component object within an assembly. The actual geometry is stored in the component part and is **referenced**, **not copied** by the assembly.



Component Occurrences

An *occurrence* of a component is a pointer to geometry in the component file. Use component occurrences to create one or more references to a component without creating additional geometry.

Reference Set

A *reference set* is a named collection of objects in a component part or subassembly that you can use to simplify the representation of the component part in higher level assemblies.

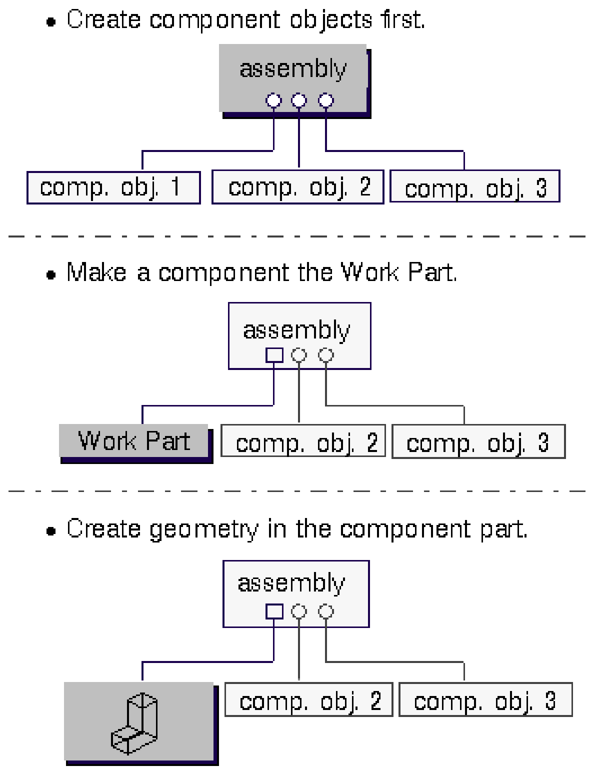
6.2 ASSEMBLING APPROACHES

There are two basic ways of creating any assembly model.

- Top-Down Approach
- Bottom-Up Approach

6.2.1 Top-Down Approach

In this approach, the assembly part file is created first and components are created in that file. Then individual parts are modeled. This type of modeling is useful in a new design. The **WAVE Geometry Linker** command could be used to model either associative linked objects or non-associative copies. When you modify the source geometry, the related linked geometry and most of its attributes will be updated based on your modification regarding the source geometry.

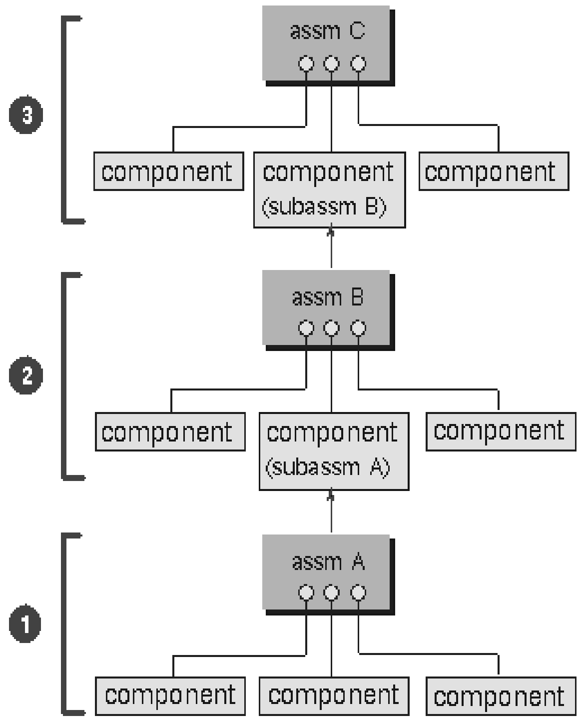


6.2.2 Bottom-Up Approach

The component parts are created first in the traditional way and then added to the assembly part file. This technique is particularly useful, when part files already exist from the previous designs, and can be reused.

6.2.3 Mixing and Matching

You can combine these two approaches, when necessary, to add flexibility to your assembly design needs.



6.3 ASSEMBLY AND CONSTRAINT NAVIGATORS

The *Assembly Navigator* and *Constraint Navigator* are located on top of the *Part Navigator* in the *Resource Bar* on the left of the screen. These navigators show you various things that form the assembly, including part hierarchy, the part name, information regarding the part such as whether the part is read only, count of objects, and the constraint status.

6.4 ASSEMBLY CONSTRAINTS

After the *Component Objects* are added to the assembly part file, each *Component Object* is mated with the existing objects. By assigning the mating conditions on components of an assembly, you establish positional relationships, or constraints, among those components. These relationships are termed *Mating Constraints*. A mating condition is made up of one or more mating constraints. There are different mating constraints as explained below:

**Touch/Align**: Planar objects selected to align will be coplanar but the normal to the planes will point in the same direction. Centerlines of cylindrical objects will be in line with each other.

Concentric: Constrains circular or elliptical edges of two components so the centers are coincident and the planes of the edges are coplanar.

Distance: This establishes a +/- distance (offset) value between two objects

Parallel: Objects selected will be parallel to each other.

Perpendicular: Objects selected will be perpendicular to each other.

Bond: Creates a weld and welds components together to move as a single object.

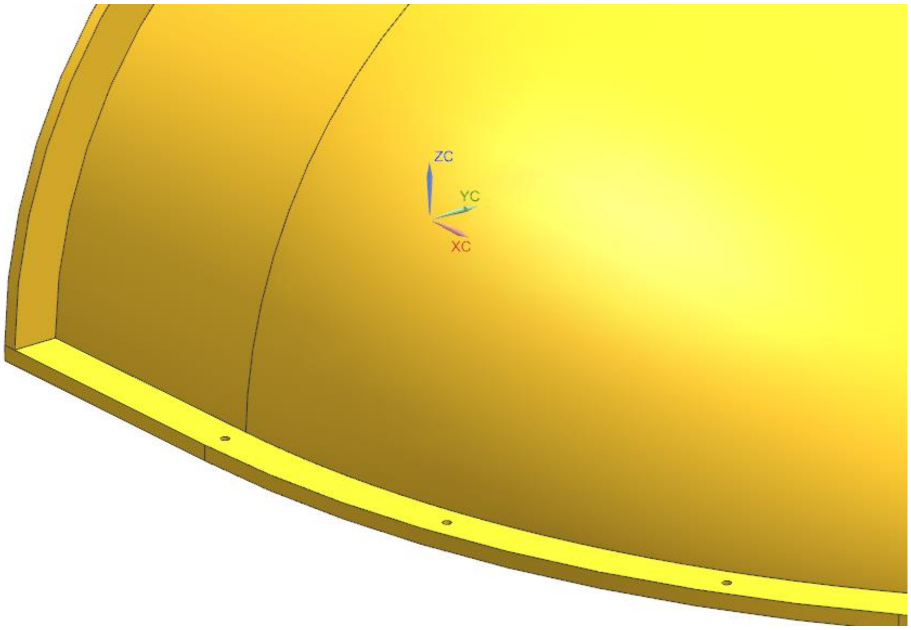
Center: Objects will be centered between other objects, i.e. locating a cylinder along a slot and centering the cylinder in the slot.

Angle: This fixes a constant angle between the two object entities chosen on the components to be assembled.

6.5 EXAMPLE

We will assemble the impeller component objects. You have modeled all the components in previous chapters. Now we have to insert them into the assembly environment and apply constraints to locate them relative to each other. Once the assembling is completed, we can create an exploded view and prepare the drafting.

Before starting the assembly modeling, make three through-holes on each side of the *Impeller-lower-casing* and *Impeller-upper-casing* (a total number of 6 holes for each casing) for the *Hexa-bolt*. Diameter of the holes should be 0.25 and their location should be similar to the figure below. Make sure to create the holes in the same places for both lower and upper casing, so that when they are assembled, they match.

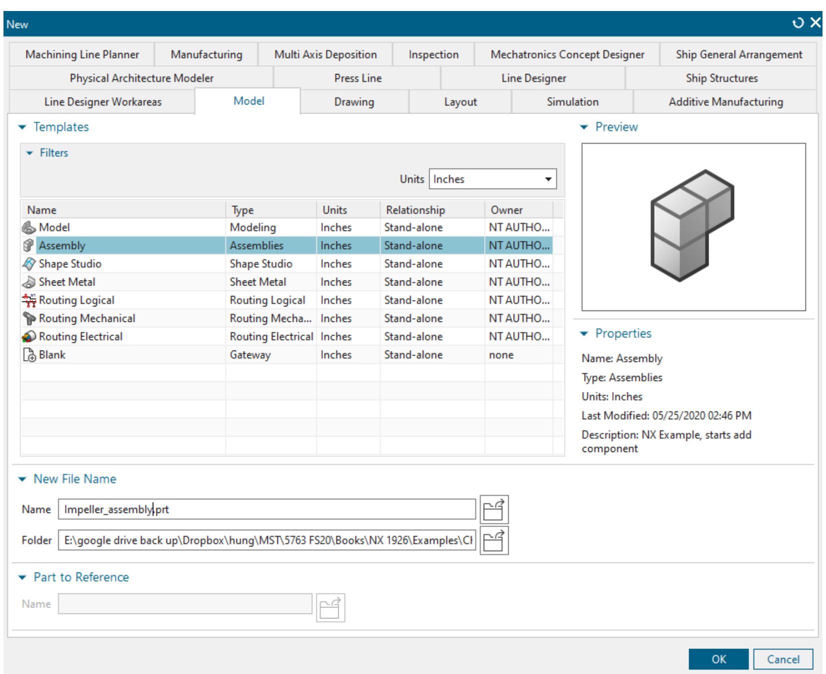


6.5.1 Starting an Assembly

- Create a new file
- Choose **Assembly** under the **Model** tab

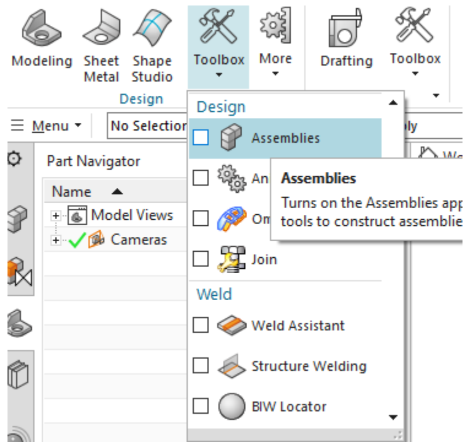
Set the **Units** → **Inches**

- Name it as **Impeller_assembly.prt**



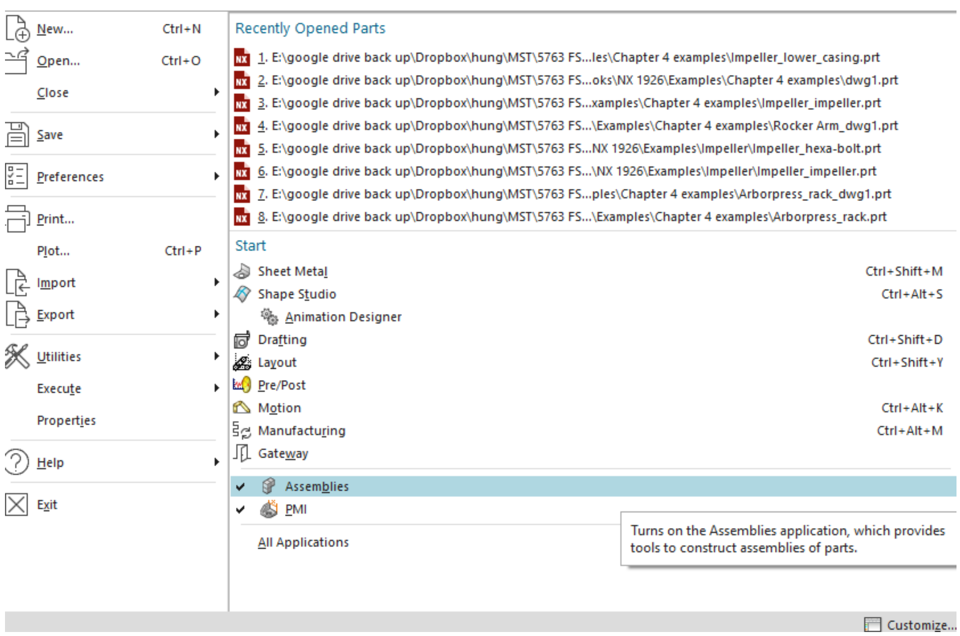
OR, if you are in the *Modeling Application* and want to start assembling,

- ➢ Turn on **Assemblies** option in **Application** tab and a new **Assemblies** tab shows up

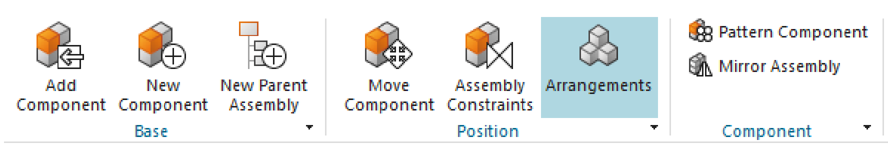


OR

Click **File** → **Assemblies** as shown below



- ➢ The **Assemblies** menu bar will now display tools for assembly



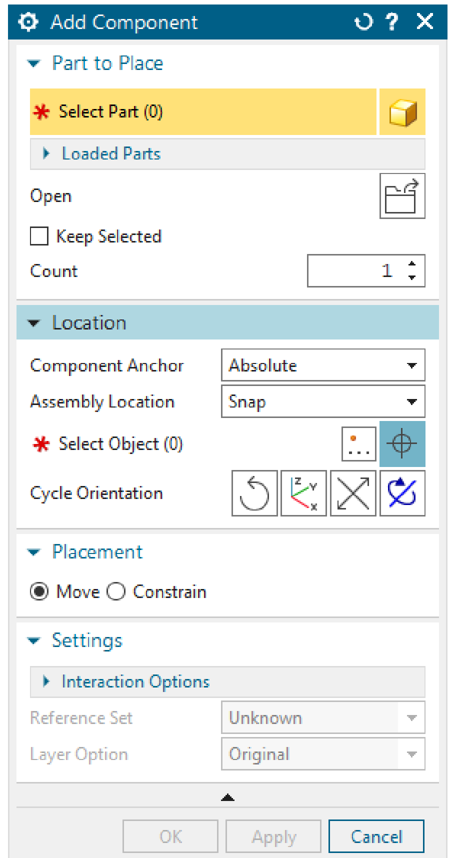
In the **Base** option,

- *Add Component* option adds new component objects whose part files are already created.
- *New Component* lets you create new component geometries inside the assembly file when you are using *Top-Down* approach of assembly.

The *Assembly Constraints* allows you to create assembly constraints and *Move Component* allows you to reposition the components wherever you want them in the assembly.

6.5.2 Adding Components and Constraints

➢ Choose **Add**



The dialogue box shown on the right side will pop up. You can select the part files from those existing (should be already shown in *Loaded Parts* tab) or you can load the part files using the *Open* file options in the dialog box. This will load the selected part file into the *Loaded Parts* dialog box.

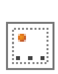
➢ Click on the **Open** icon and select the file **Impeller_upper-casing.prt**

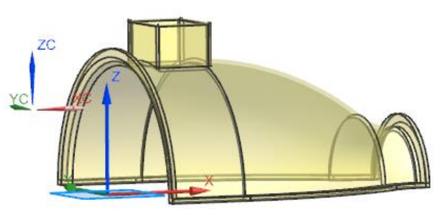
➢ Click **OK** in the part name dialog box

Then we need to set a location to place the coordinate system of the first component. In the *Location* group box, keep the default *Snap* for the *Assembly Location* option.

➢ Click **Select Object**

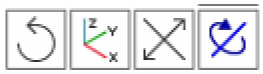
Now you should be able to see the part in a transparent mode, as shown in the figure on the right side.

➢ Click the **Point** Dialog icon  and create the coordinates of [0, 0, 0]



➢ Click **OK** to exit the **Point** Dialog

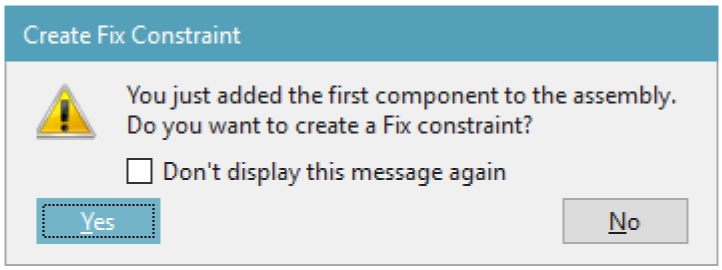
Note: Feel free to play with the **Cycle Orientation** options to set different orientations.

➢ Cycle Orientation 

In the *Placement* group box, we can define where and how we place this component. In this case, we will leave the options as default.

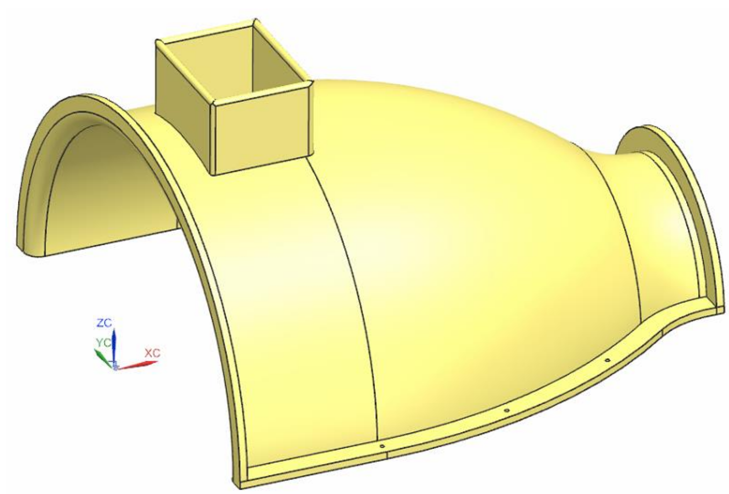
➢ Click **OK** to exit the **Add Component** Dialog

Then, you will see a pop-up dialog appears as shown in the figure below.



➢ Click **Yes** to create a Fix constraint to this part.

Now what you have should like the figure below.

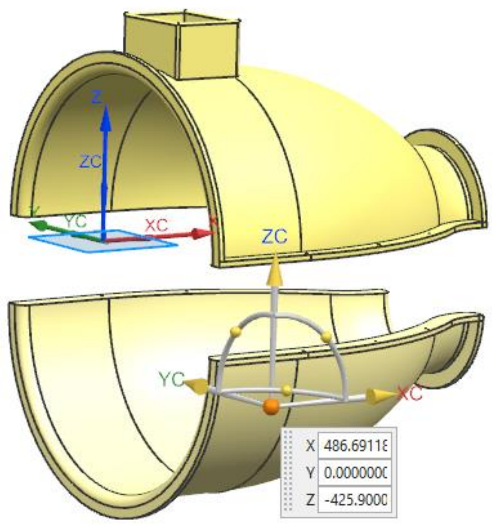


Let's move on to add the second component, the lower casing.

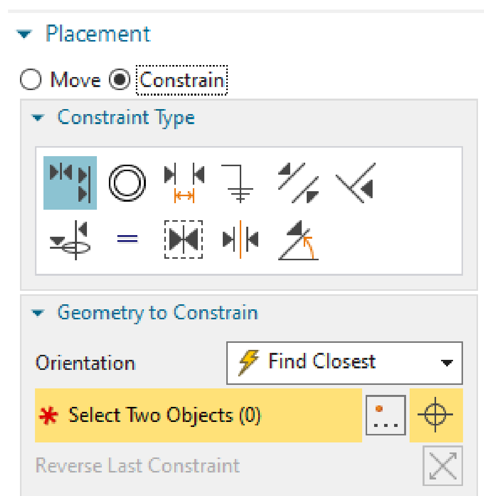
➢ Click on **Add** in the assembly section

➢ Select the file **Impeller_lower-casing.prt** from the *Loaded Parts* or *Open*

- In the **Location** group box change the option to **Absolute – Work Part** to place the new part at the absolute origin of the current work part
- In the **Placement** group box, first toggle the **Move** radio button and move the lower casing away from the upper casing to have enough space for selecting the mating surfaces. What you will have should like the figure below.



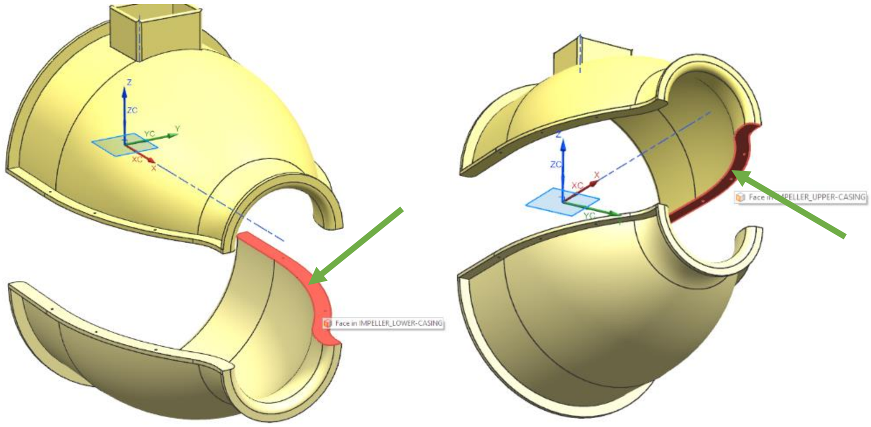
- Then, toggle the **Constrain** radio button.



Now let's mate the upper and the lower casings. You can access all the constraints in the drop-down menu in the *Constraint Type* box.

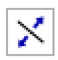
Here you can see the different Mating Types, which were explained above in the previous section.

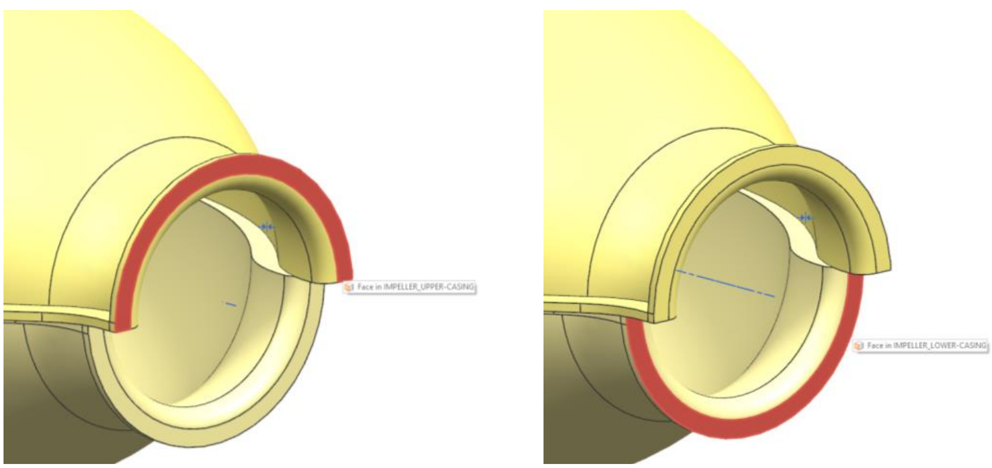
- Make sure the **Touch Align** icon  is selected in the **Type** dialog box
- First, select the face that the arrow is pointing to as shown below in the figure on the left.
- Click on the face of the upper casing in the screen as shown in the figure on the right.



You may have to adjust the view in order to select the faces. After you choose these two faces, the Touch Align constraint will be automatically added.

Let’s add another **Touch Align** constraint.

- Click on the **Flange** of the upper casing
- Click on the **Flange** of the lower casing, you may need to inverse the direction of constraint by click on the **Inverse** icon 

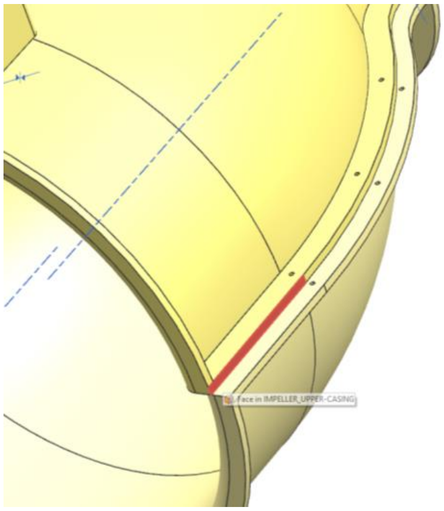


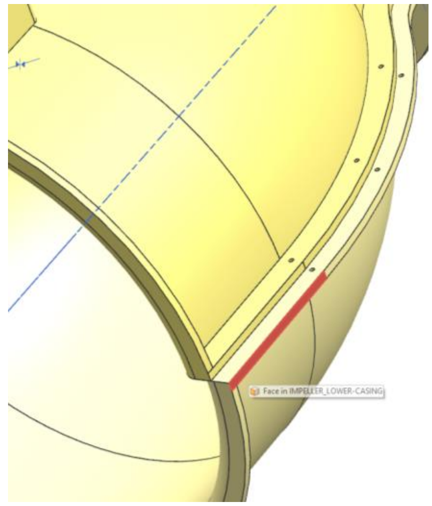
Note: If it is difficult for you to select the faces because of the position of the parts, you can move them by toggling the *Move* in the Placement group box and manipulate its handler.

Let’s add another **Touch Align** constraint.

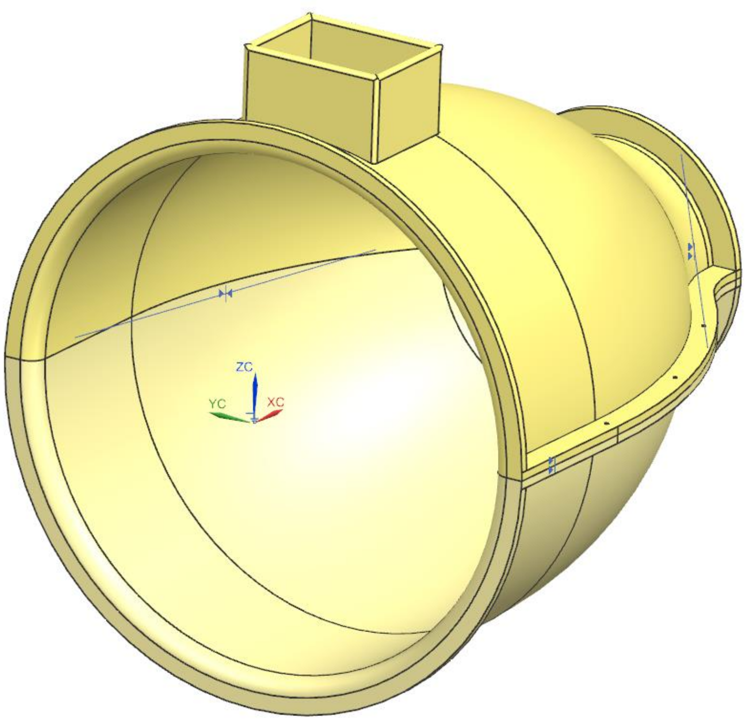
- Make sure you are still using **Touch Align**

➢ Click on the flat face of the upper casing as shown and then the corresponding face on the lower casing





The two assembled components will be seen as shown in the figure below.

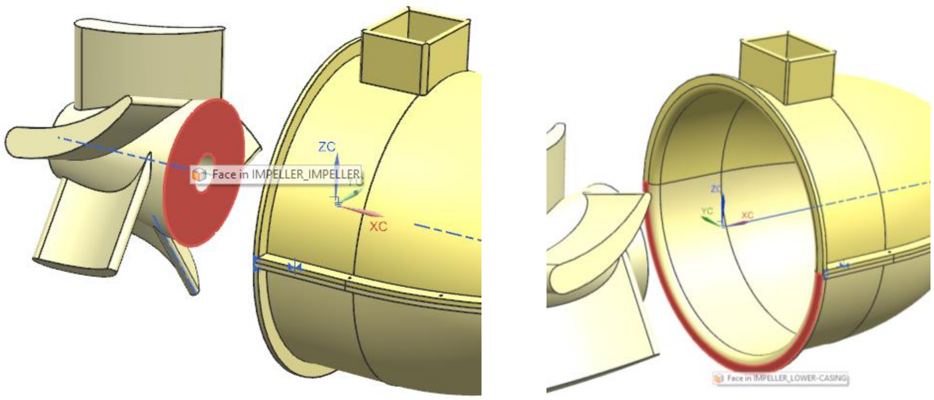


The lower casing is constrained with respect to the upper casing. Now let us add the impeller.

➢ Choose **Assemblies** → **Component** → **Add** to add a component to the current assembly

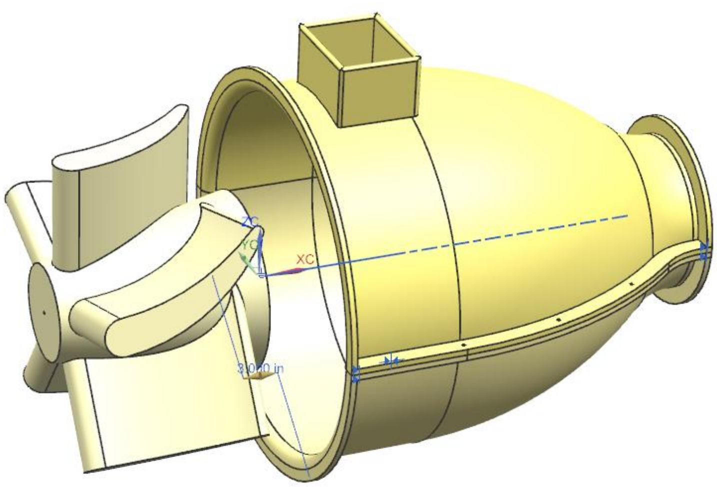
➢ Open the file **Impeller_impeller.prt**

- Click **OK** on the dialog box
- Choose *Absolute – Work Part* for Assembly Location
- Toggle the **Constrain** button
- Click on the **Distance** icon  in the **Constraint Type** box
- Select the two faces, first on the impeller and then on the casing, as shown in the figure below

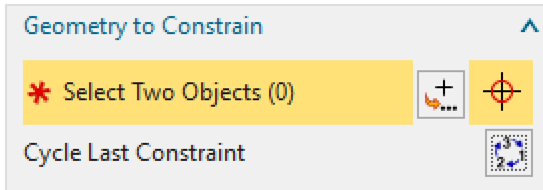


- In the **Distance** box in the **Placement** group, enter the value of **3**
- Press **Enter** to preview the current assembly

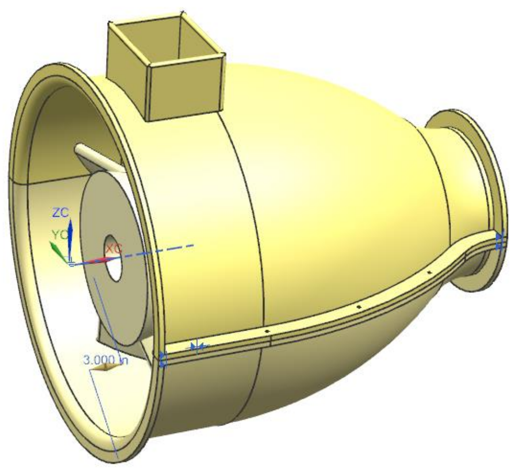
The preview may show the impeller oriented in the direction opposite to the one we want.



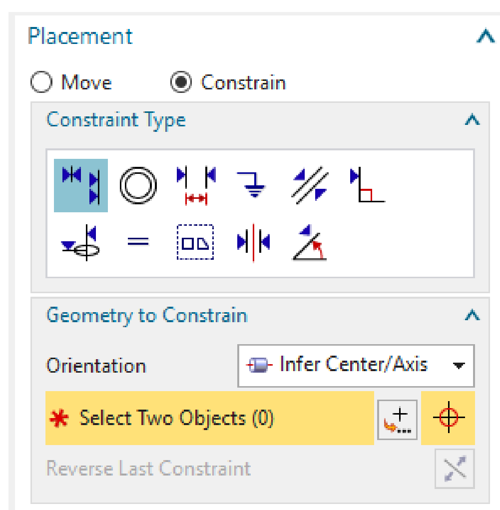
➢ To change the orientation of the part or the distance direction, in the **Placement** window, click on the **Cycle Last Constraint**  button in the **Geometry to Constrain** box, as shown below



Check the assembly status from its preview, you may have to click on the  button for several times to get the desired result. Now the impeller will be oriented in the right direction.



We will now add the shaft using the *Center* constraint.



➢ Click on **Assemblies → Components → Add**

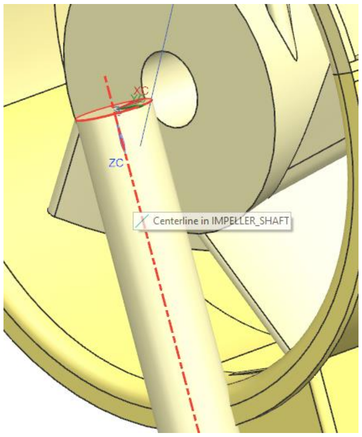
➢ Open the file **Impeller_shaft.prt**

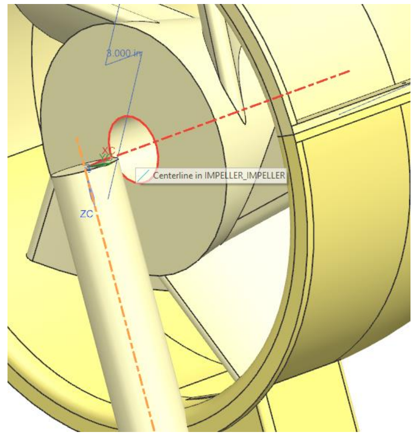
➢ Click **OK** on the dialog box

➢ Choose the **Touch Align** icon in the **Constraint Type** box

➢ Choose the **Infer Center/Axis** option in the **Geometry to Constrain** box

➢ Select the shaft and inner hole’s axis, first on the shaft and then on the impeller as shown in the figures below

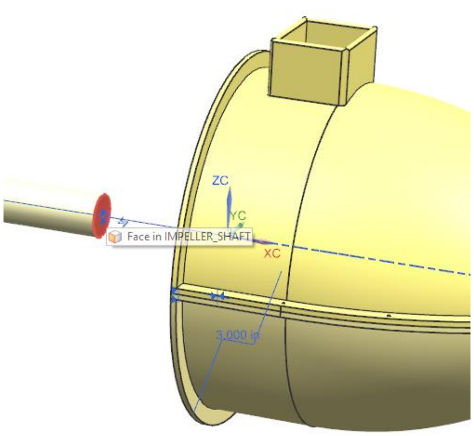


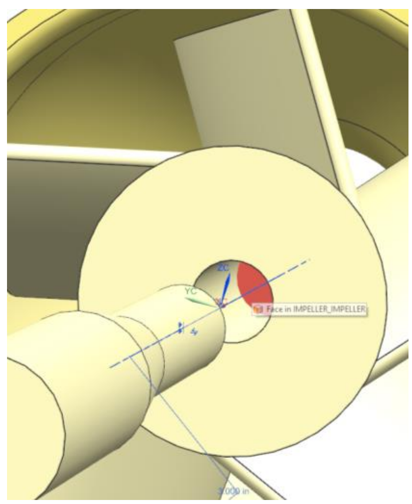


➢ Stay in the **Touch Align**  constraint

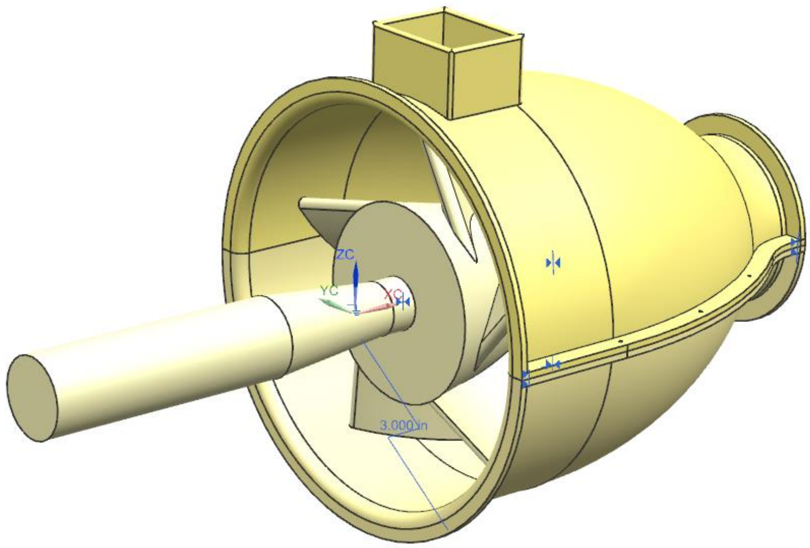
➢ Choose the **Prefer Touch** option in the **Geometry to Constrain** box

➢ First, select the face on the shaft and then select the bottom face of the hole in the impeller as shown

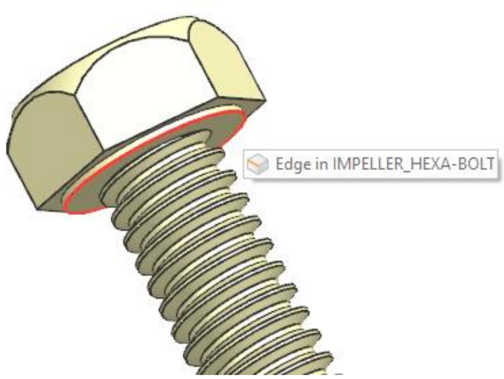


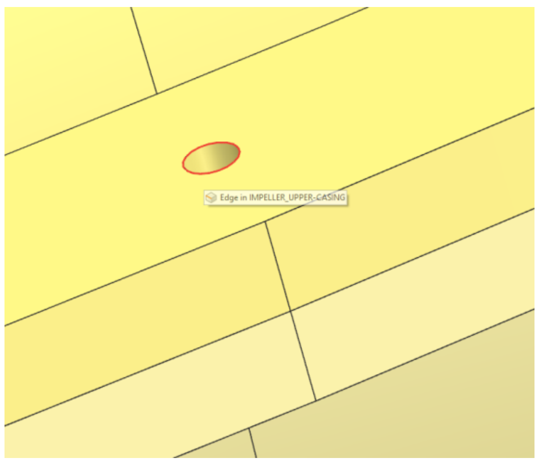


The assembly will now look like the figure below.

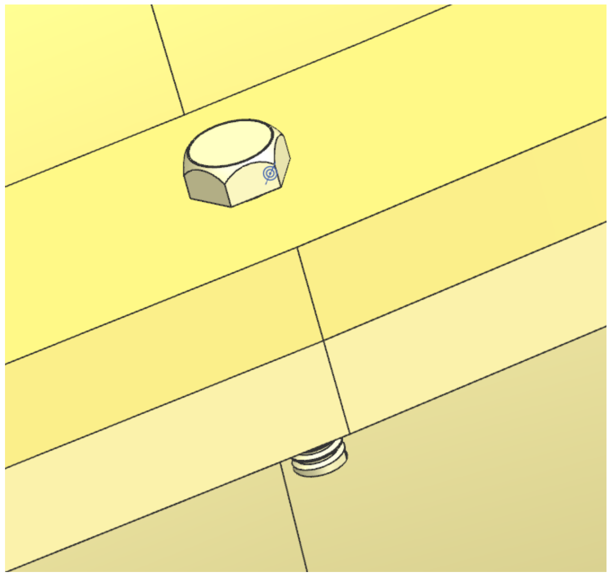


- Click on **Assemblies** → **Components** → **Add**
- Open the file **Impeller_hexa-bolt.prt**
- Choose the **Concentric** constraint. Use the **Infer Center/Axis** option in the **Geometry to Constrain** box
- First, select the outer circular edge of bolt and then select the hole's edge on the upper casing as show in the figures below.





The assembly is shown below.

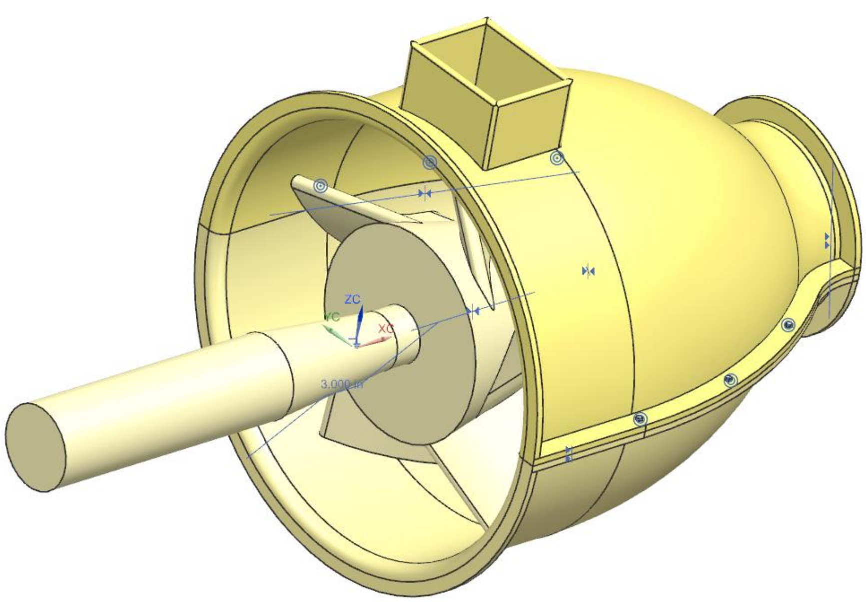


- ➢ Repeat the same procedure to add bolts and nuts to all the holes in the assembly.

This completes the assembly of the impeller.

Note: There is a simpler way to assemble the bolt and nut set. Instead of adding the three parts individually, you can assemble these components separately in another file. This will be a sub-assembly. You can insert this subassembly and mate it with the main assembly.

The Final Assembly will look as the shown below. Save the Model.

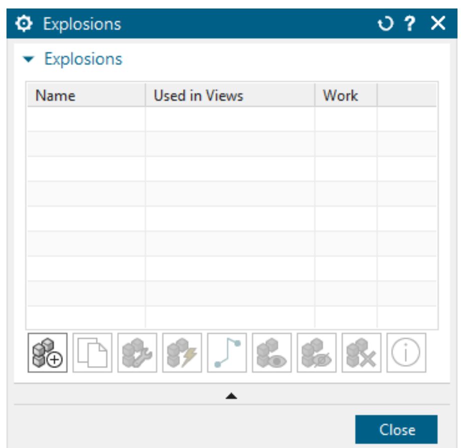


6.5.3 Exploded View

In this section, we are going to [create an *Exploded View*]() of the assembly to show a separated part-by-part picture of the components that make the assembly. In today's industrial practice, these kinds of views are very helpful on the assembly shop floor to get a good idea of which item fixes where. The user should understand that exploding an assembly does not mean relocation of the components, but only viewing the models in the form of disassembly. You can *Unexplode* the view at any time you want to regain the original assembly view. Let us explode the Impeller Assembly.

- Choose **Menu** → **Assemblies** → **Explosions**

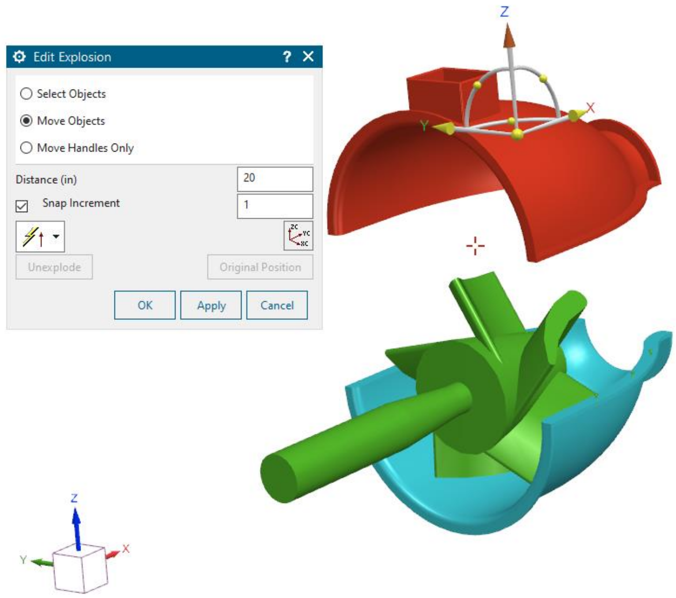
This will pop up a dialog box asking and click **New Explosion** .



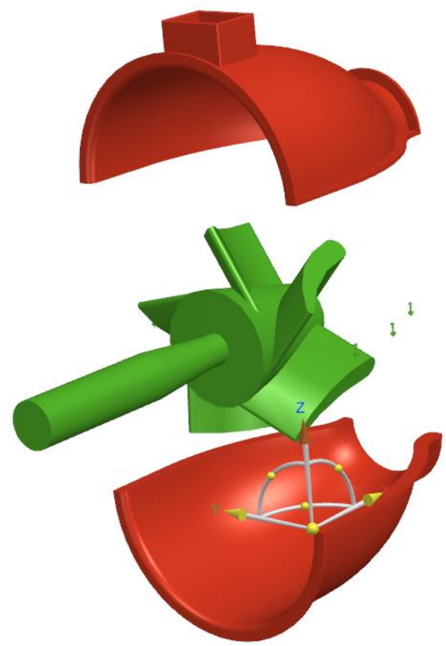
Now the NX environment is in *Exploded View* environment though you do not find any difference. When we start exploding an assembly, we should decide upon a component to keep that component as the reference. This component should not be moved from its original position. In the case of the impeller assembly, the impeller will be the right option as it is central to the entire assembly. Now let us start exploding the components.

- **Edit Explosion** dialog box will pop up

The *Edit Explosion* window will pop up along with coordinate handles on the component.

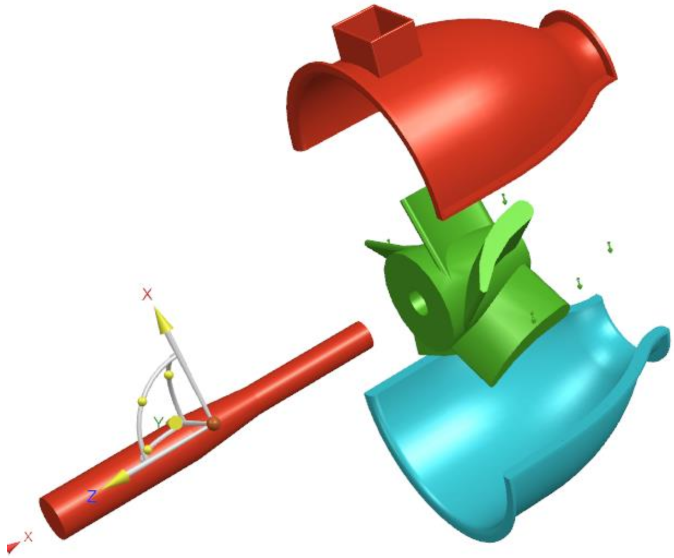


- Click on the **Z** axis; hold the mouse and drag upwards until the reading in the **Distance** shows -**20 (substitute +20 if you have designed in opposite direction)**
- Click **OK**
- Right click on the lower casing and choose **Edit Explosion**

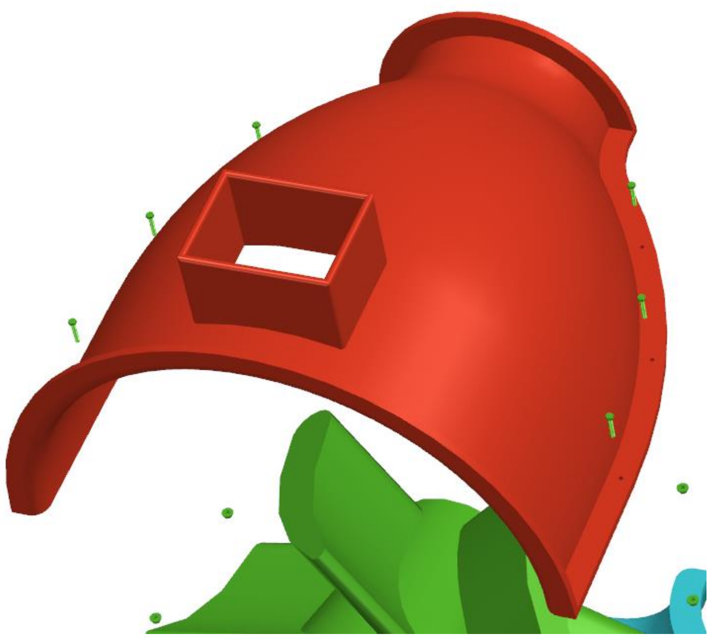


Again, this will pop up a dialog window for *Edit Explosion* and a coordinate system on the component.

- Click on the **Z-axis**; hold the mouse and drag downwards until the reading in the Distance shows **20** as shown in the figure on the right side.
- Right click on the shaft and choose **Edit Explosion**
- This time click on the **Z-axis**; hold the button and drag to the right side until the reading in the distance shows **-25 or 25** as shown in the following figure
- Choose **OK**

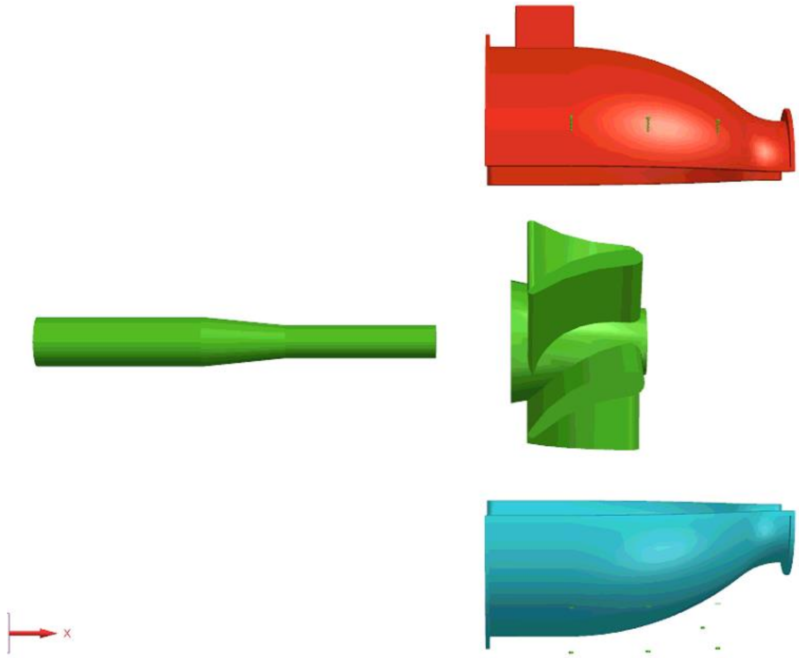


- Select all the six hexagonal bolts in the assembly by clicking on them
- Right click on one of them and choose **Edit Explosion**
- This time click on the **Z-axis**; hold the button and drag upwards until the reading in the Distance shows **25** as shown in the following figure. This will move all the six bolts together to the same distance.
- Choose **OK**



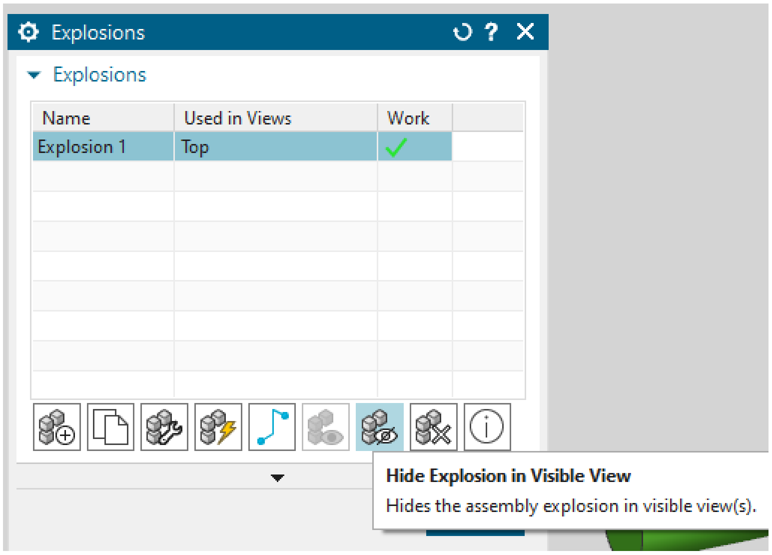
➢ Likewise, move the six washers and the six hexagonal nuts downwards to -30 and -35, respectively.

This is the Exploded view of the assembly. You can rotate and see how it looks like.



If you want to go back to the original unexploded view,

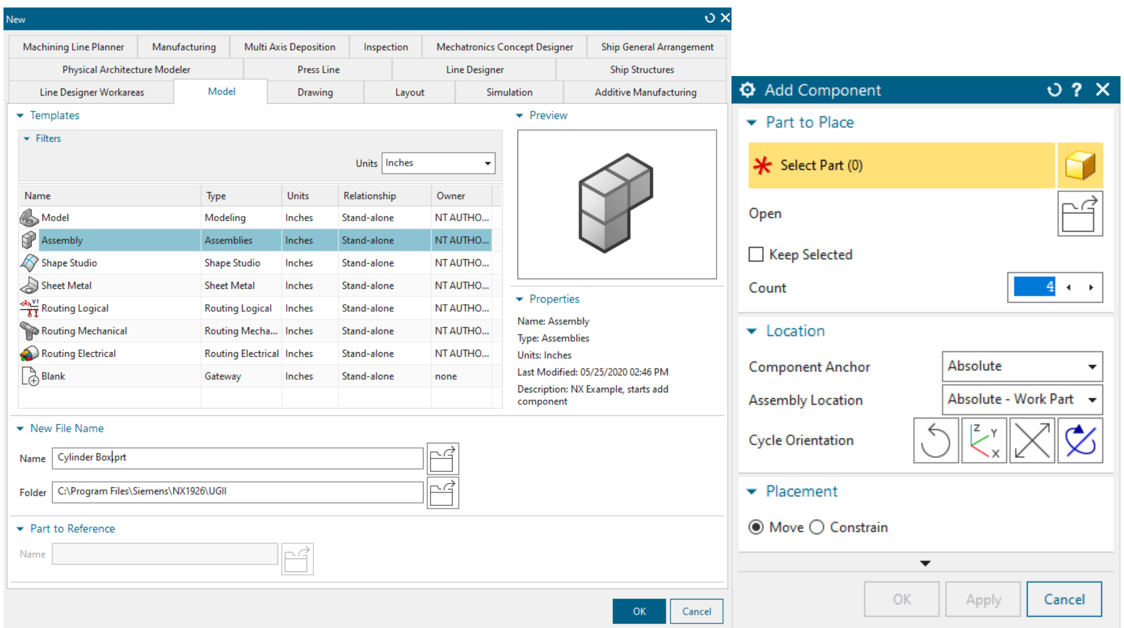
➢ Click **Hide Explosion in Visible View** in the **Explosions**



6.5.4 WAVE Geometry Linker

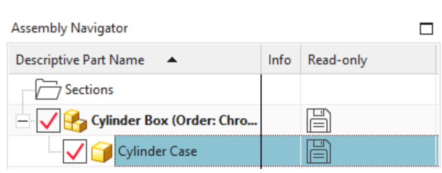
When designing parts in a design group, you have to cooperate with partners. The WAVE Geometry Linker can help you to use any kind of associativity between components and speed up your design efficiency. Here we will use a basic assembly on Cylinder Case as an example to create a Cylinder Cap using WAVE Geometry Linker. The dimension in Cylinder Cap were copied from geometries from the Cylinder Case. When the Cylinder Case dimension changes, the dimension of the Cylinder Cap will change accordingly and simultaneously because this Cap was modeled by referencing dimensions generated using WAVE Geometry Linker.

- Create a new **Assembly** and call **Cylinder Box**
- **Cancel** in **Add Component** window

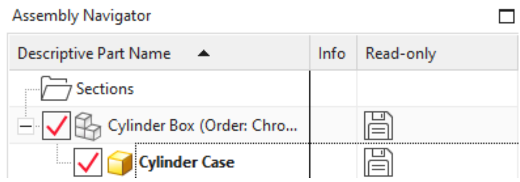


- Click **New Component**  in **Assemblies**
- Choose **Model** and name **Cylinder Case**
- **OK**

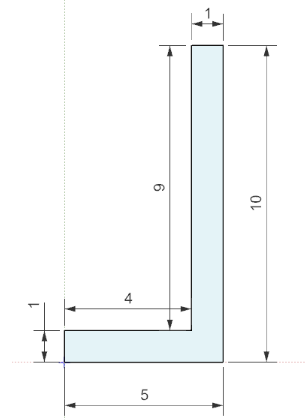
Now you will see **Cylinder Case** under **Cylinder Box** in **Assembly Navigator**



- Double clicks on **Cylinder Case** to highlight and start editing it

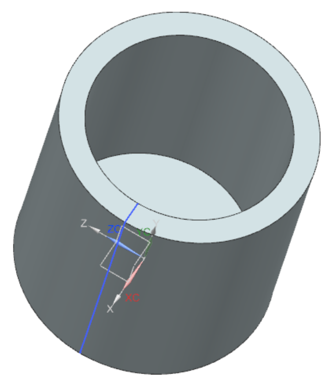


- Click **Sketch**: Specify Plane in **XC-YC Plane** in **Plane Dialog** and Specify Point **(0,0,0)** in **Point Dialog**

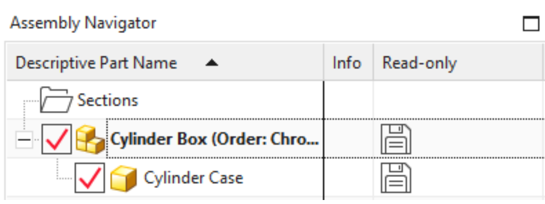


- Make a sketch as shown in the right figure
- **Finish**
- Click **Revolve**: Specify Vector in **YC Axis** and Specify Point **(0,0,0)** in **Point Dialog**
- **OK**

You will have a **Cylinder Case** model as shown in figure.

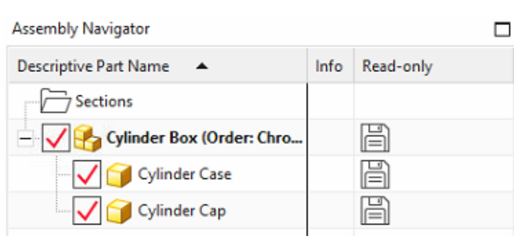


Before creating the Cap, double click on **Cylinder Box**

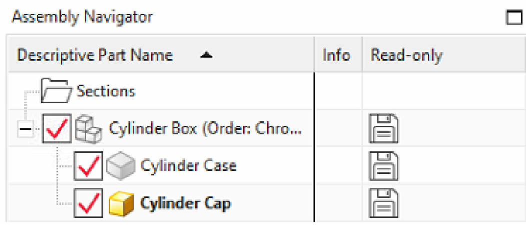


- Click **New Component** in **Assemblies**
- Choose **Model** and name **Cylinder Cap**
- **OK**

Now you will see **Cylinder Case** and **Cylinder Cap** under **Cylinder Box** in **Assembly Navigator**

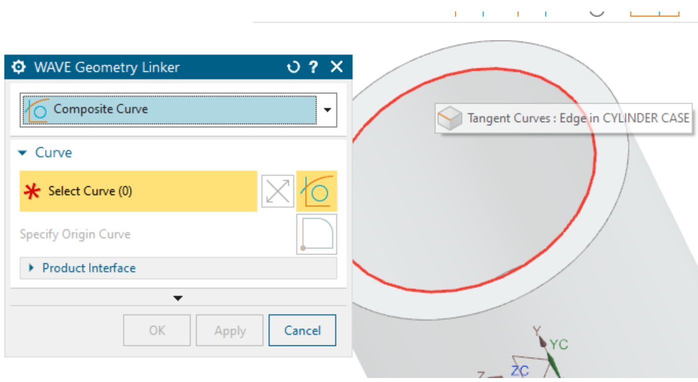


➢ Double clicks on **Cylinder Cap** to highlight and start editing it

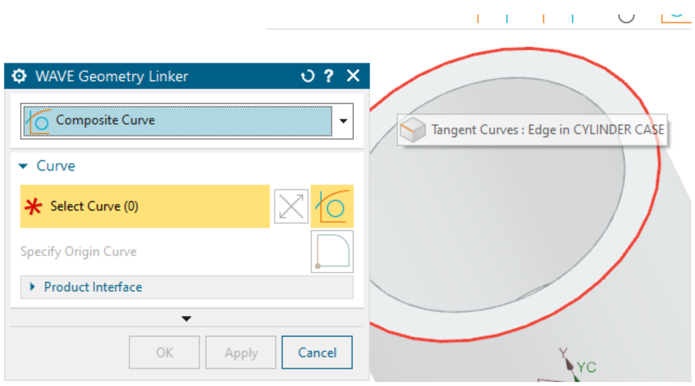


Open Insert → Associate Copy → WAVE Geometry Linker

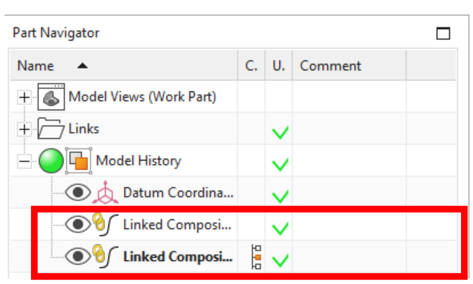
➢ Click inner circle and **Apply**



➢ Click Outer circle and **OK**

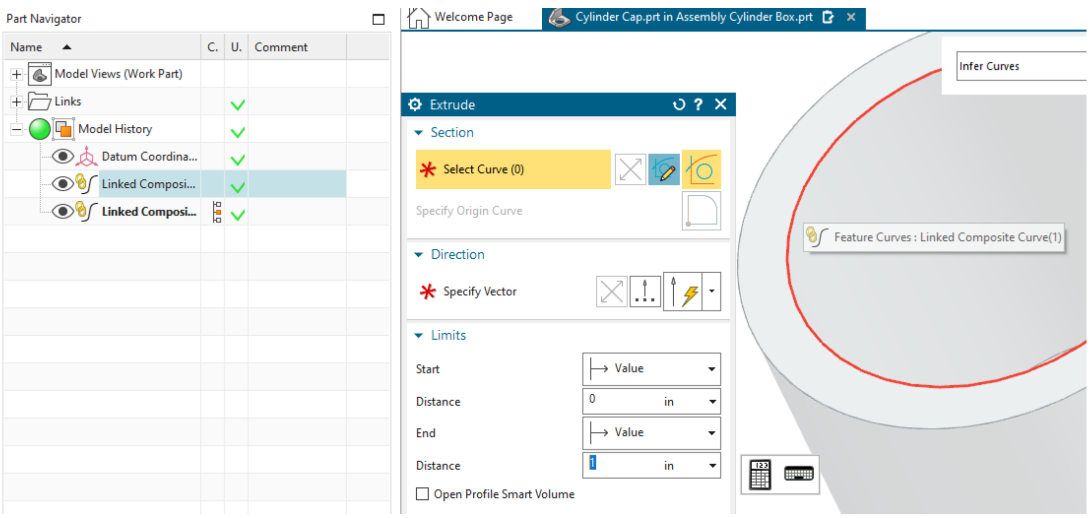


In Part Navigator, you will see two curves you just copied using WAVE Geometry Linker.



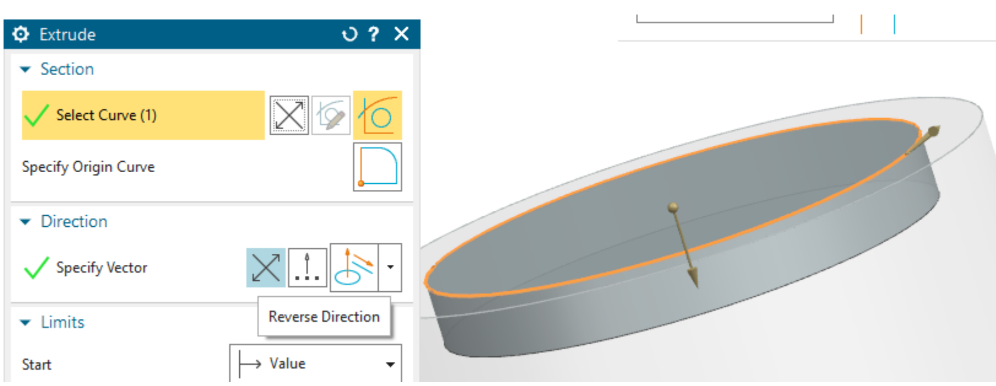
Now, let's create the cap with these two curves.

➢ Click **Extrude** and select inner circle (Linked Composite Curve)

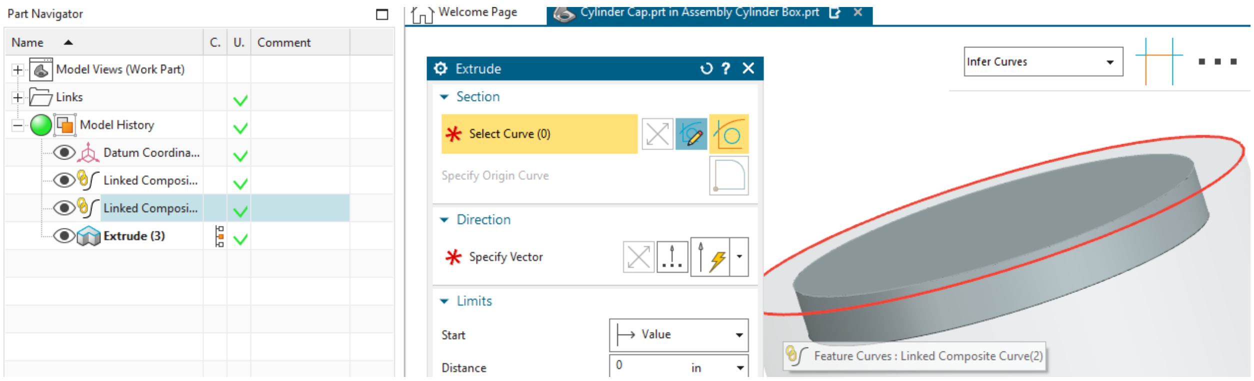


➢ Click **Reverse Direction** to reverse extrusion direction if the default extrusion direction is incorrect

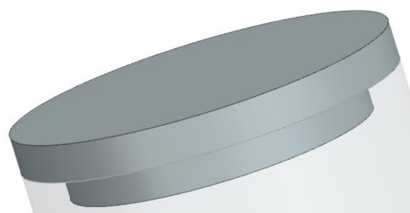
➢ **Apply**



➢ Click **Extrude** and select outer circle (Linked Composite Curve)

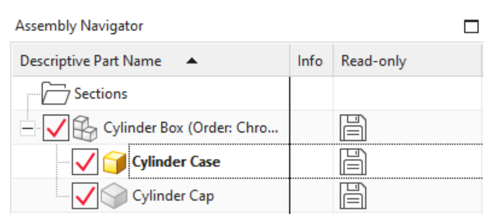


Now, the **Cylinder Cap** is modeled as shown below

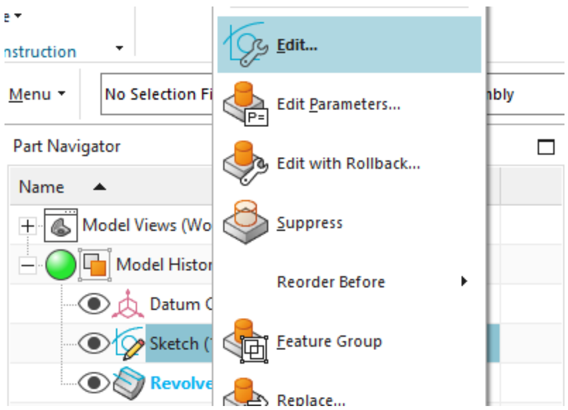


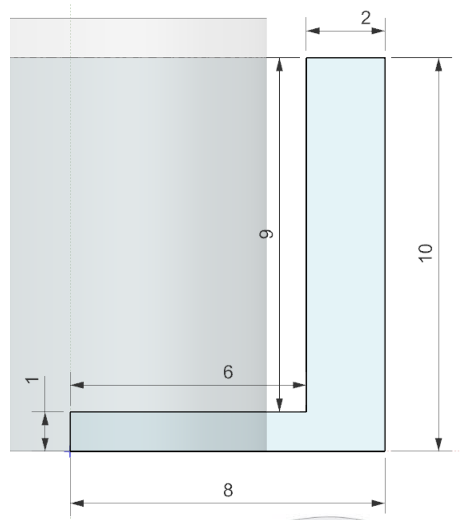
The **Cylinder Cap** were created by referencing the curve from the **Cylinder Case** using the WAVE Geometry Linker. When the curves of Cylinder Case changes, the dimensions of Cylinder Cap will change simultaneously. Let’s change the size of Cylinder Case.

- Double clicks on **Cylinder Case** to edit it in **Assembly Navigator**



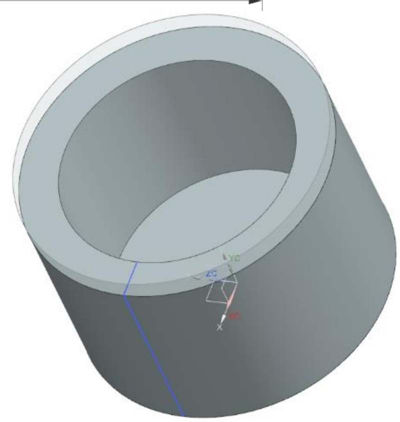
➢ Right click on **Sketch** to edit its dimensions as shown in figures





➢ **Finish**

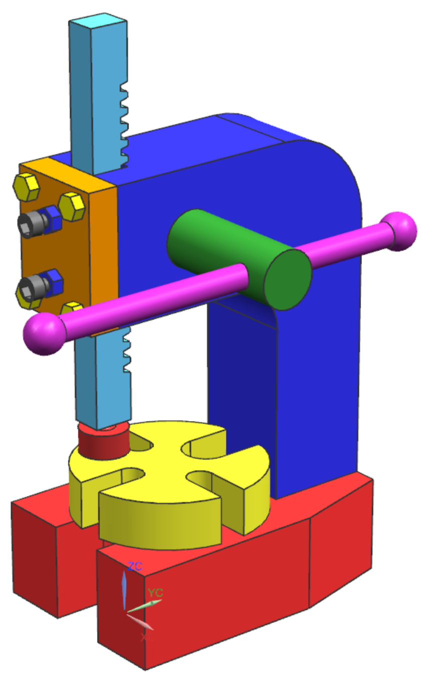
You can see the dimensions of Cylinder Cap are changed immediately. Therefore, using the WAVE Geometry Linker can make your modeling efficiently, especially dealing with complex modeling design.



6.6 EXERCISES

6.6.1 Arbor Press

In the previous sections of this tutorial, we have modeled various parts, some of which are components of the arbor press shown below. Assemble the arbor press using the components that you have modeled in addition to ones that are provided to you, which you have not modeled before. The complete list of parts of the arbor press assembly is provided below. All these parts are provided in a folder that can be accessed along with this tutorial in the same internet address (https://web.mst.edu/~mleu/).



- Allen Bolt
- Allen Nut
- Base
- Circle base
- End clip
- Handle

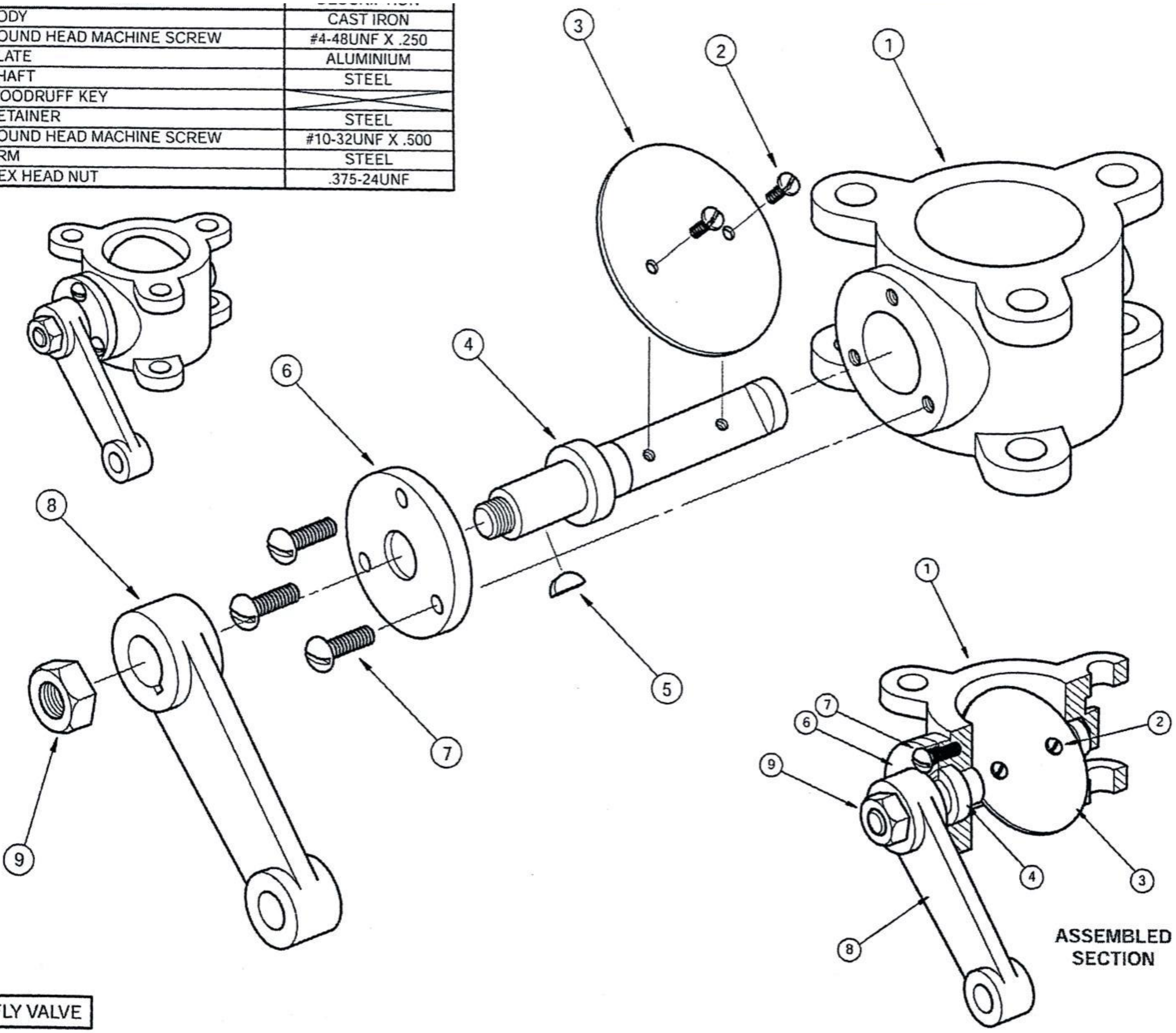
- Hexagonal Bolt
- L-bar
- Pin
- Pinion
- Pinion handle
- Plate
- Rack
- Sleeve

6.6.2 Butterfly Valve

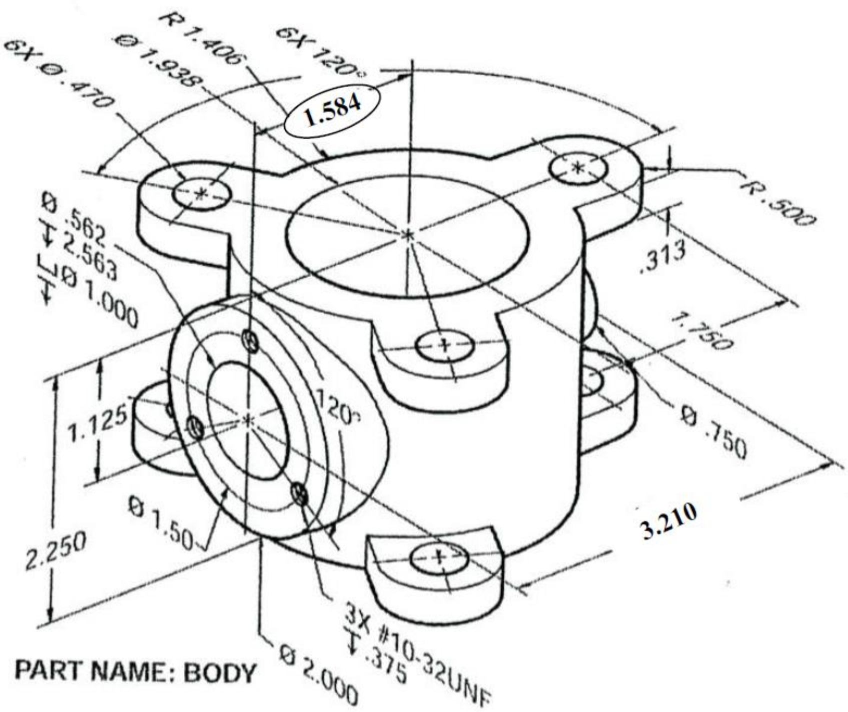
Butterfly valve is one of the most commonly used devices for controlling flow. The Butterfly valve consists of a rotating disk positioned in the pipe. The disc is attached to the shaft that is connected to an actuator on the outside of the valve. Rotating the actuator turns the disc either parallel or perpendicular to the flow. When the valve is closed, the disc is turned so that it completely blocks off the passageway. When the valve is fully open, the disc is rotated a quarter turn so that it allows an almost unrestricted passage of the fluid. The valve may also be opened incrementally to regulate flow.

In this exercise, you will model and assemble the Butterfly valve consisting of 9 different parts for a total of 12 components. The names, drawings and dimensions of the different part items, as well as how the various components fit together, are provided. The parts for which the drawings are not given are to be designed or selected, for example, the round head machine screw. All dimensions are in inches.

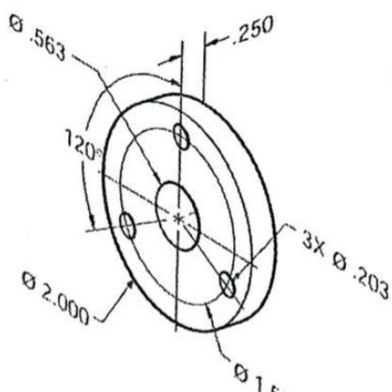
1	1	BODY	CAST IRON
2	2	ROUND HEAD MACHINE SCREW	#4-48UNF X .250
3	1	PLATE	ALUMINIUM
4	1	SHAFT	STEEL
5	1	WOODRUFF KEY	
6	1	RETAINER	STEEL
7	3	ROUND HEAD MACHINE SCREW	#10-32UNF X .500
8	1	ARM	STEEL
9	1	HEX HEAD NUT	.375-24UNF



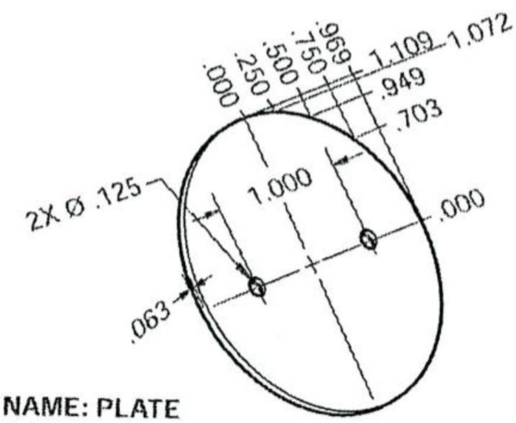
BUTTERFLY VALVE



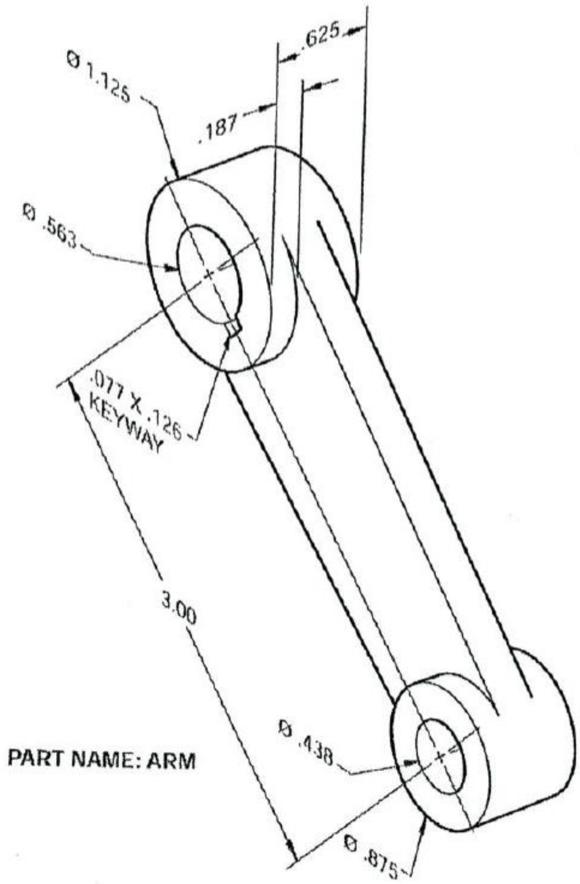
PART NAME: BODY

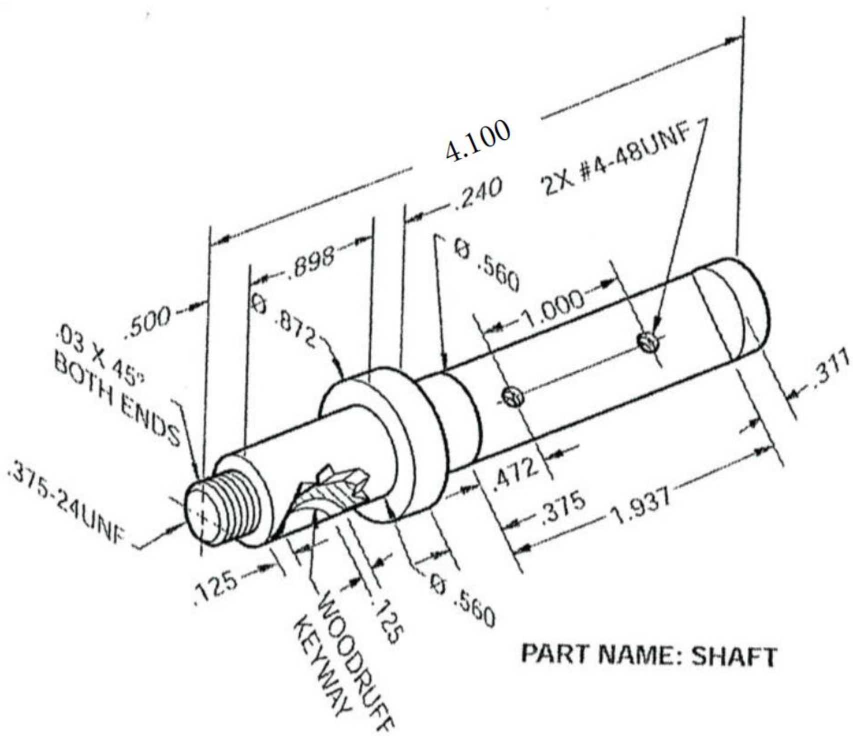


PART NAME: RETAINER

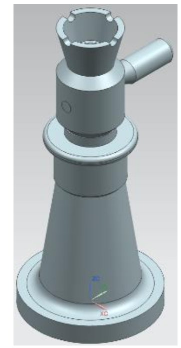


PART NAME: PLATE
(ELLIPTICAL)

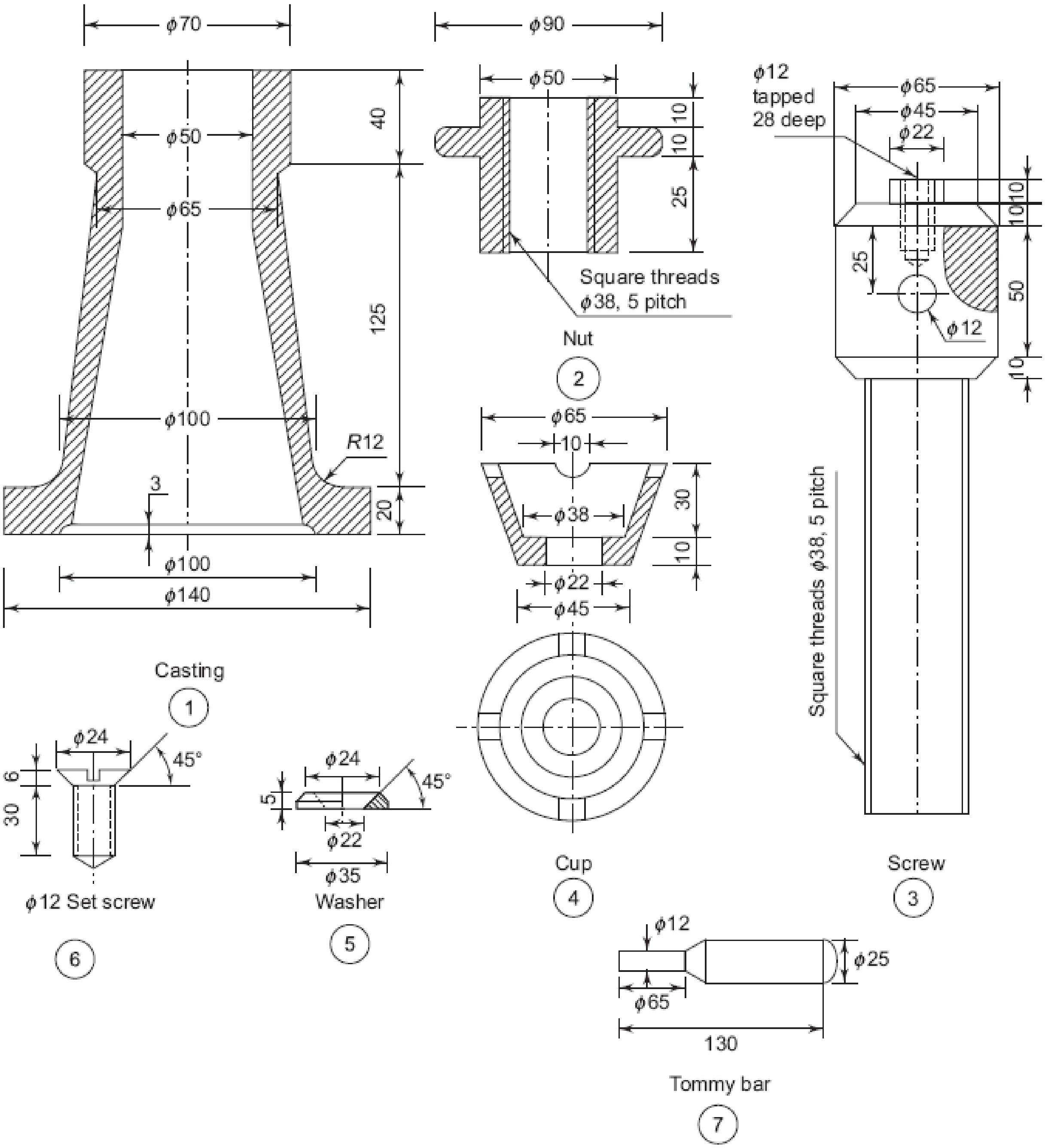

Ø 1.125
.187
.625
Ø .563
.077 X .126
KEYWAY
3.00
Ø .438
Ø .875
PART NAME: ARM

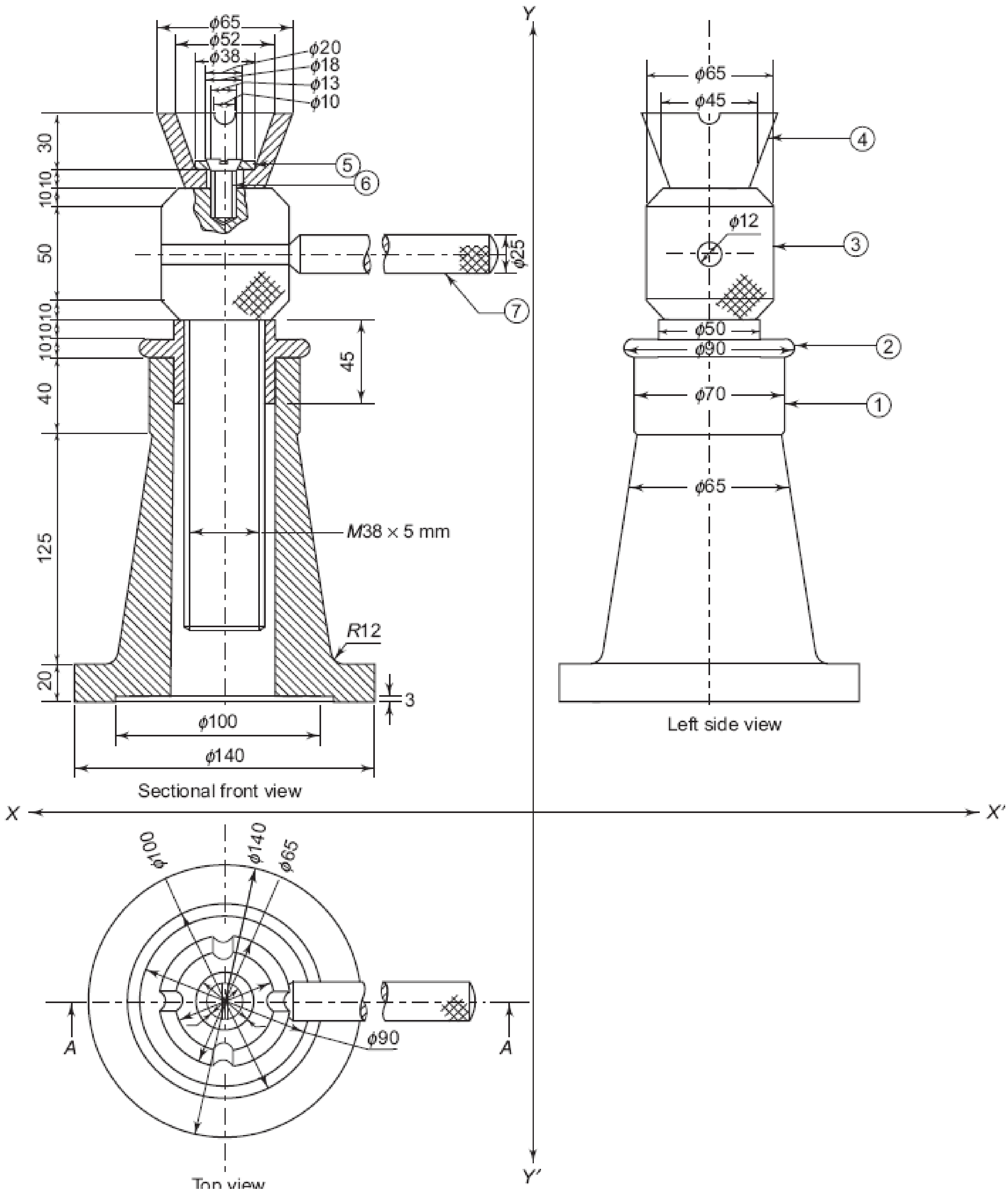

4.100
.240
2X #4-48UNF
.898
Ø .560
.500
Ø .872
1.000
.03 X 45°
BOTH ENDS
.311
.472
.375
1.937
.375-24UNF
.125
.125
Ø .560
WOODRUFF
KEYWAY
PART NAME: SHAFT

6.6.3 Jackscrew



A jackscrew is a type of jack that is operated by turning a leadscrew. In this exercise, you are asked to model, assemble and prepare drawings of parts. All dimensions should be in millimeters. Create individual drafts for each component. Draft the final assembly and make a table, listing out individual components. The Assembly draft should have an exploded view.

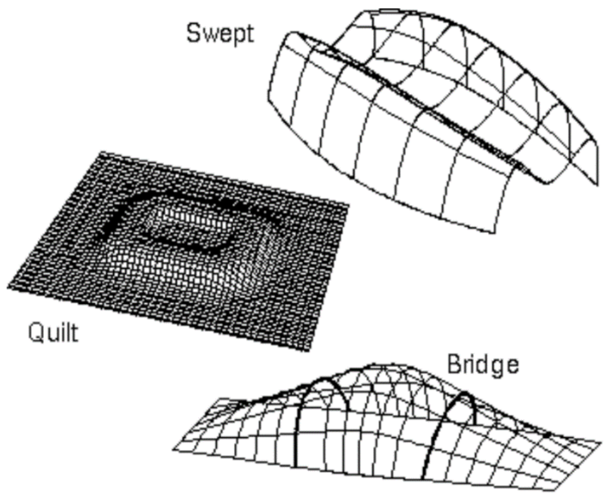




φ65
φ52
φ38
φ20
φ18
φ13
φ10
30
10
10
50
10
10
10
40
125
20
5
6
7
φ25
45
M38 × 5 mm
R12
3
φ100
φ140
Sectional front view
Y
φ65
φ45
4
φ12
3
φ50
2
φ90
φ70
1
φ65
Left side view
X
X′
φ100
φ140
φ65
φ90
A
A
Top view
Y′

CHAPTER 7 – FREEFORM SURFACE MODELING

In this chapter, you will learn how to create freeform surfaces in NX. Up to this point, you have learned different ways to create models by using *Form Features* or by *Sketching*. *Freeform surface* modeling involves creating models in the form of surfaces for aesthetic or functional purposes, such as car bodies and turbine blades. A few freeform features are shown below.



To create *Freeform Features*, you first need a set of points, curves, edges of sheets or solids, faces of sheets or solids, or other objects. The following sections cover some of the methods that you can use to create models using freeform features.

7.1 OVERVIEW

In NX, the *Freeform Features* options are located at various places like ***Menu*** → ***Insert*** → ***Surface/Mesh Surface/Sweep/Flange Surface*** **and** ***Menu*** → ***Edit*** → ***Surface*** for more advanced operations. There are a lot of ways in which you can create *Freeform Features* from the existing features you have like points, edges, curves, etc. A few of commonly used functions are discussed in the following sections.

7.1.1 Creating Freeform Features from Points

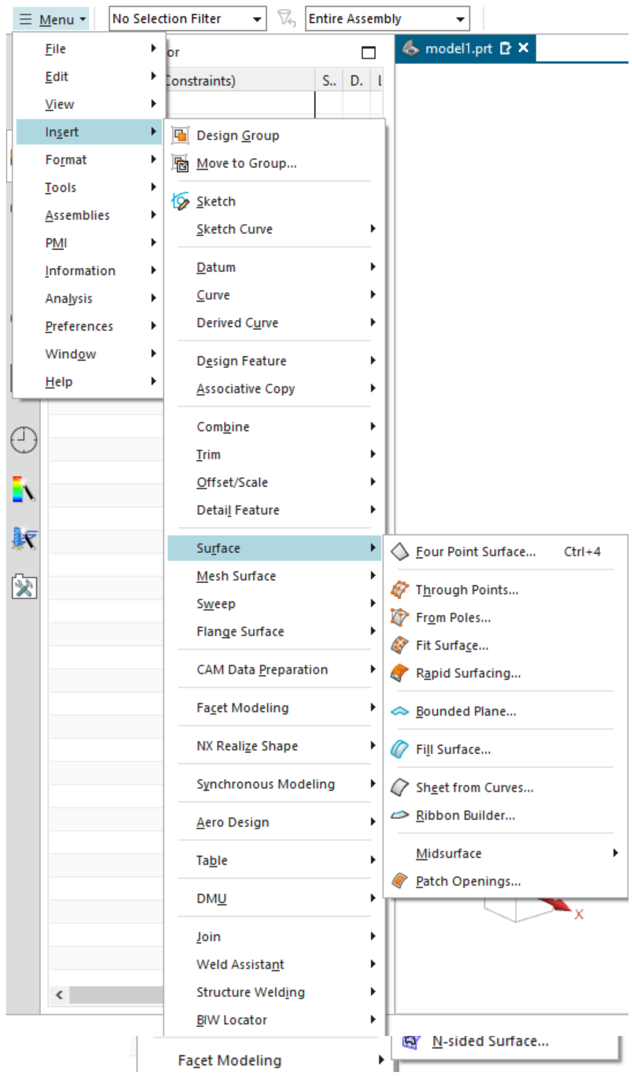
In the case where the geometry you are constructing or pre-existing data includes only points, you can try using one of the following three options to build a surface from the given points.

- From **Menu**, click on **Insert** → **Surface**

 **Four Point Surface:** if you have four corner points.

 **Through Points:** if the points form a rectangular array.

 **From Poles:** if defined points form a rectangular array tangential to the lines passing through them.

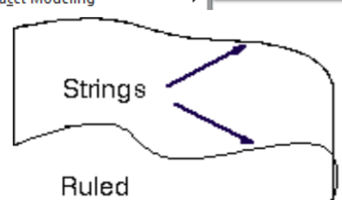


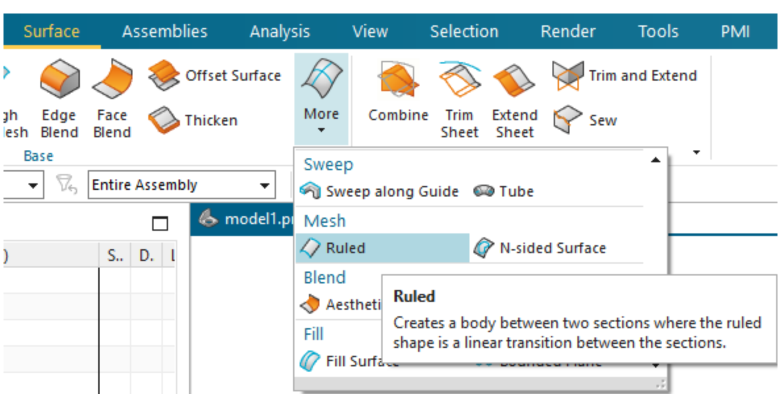
7.1.2 Creating Freeform Features from Section Strings

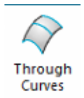
If construction geometry contains strings of connected objects (curves and edges), you can use one of the following two options to create a freeform surface.

- ➢ From **Base** group in **Surface**, click on **More**

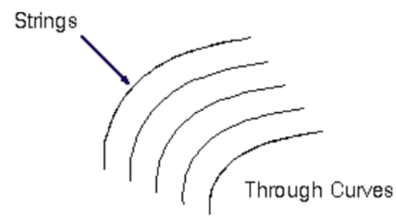
Ruled: if you have two strings which are roughly parallel.





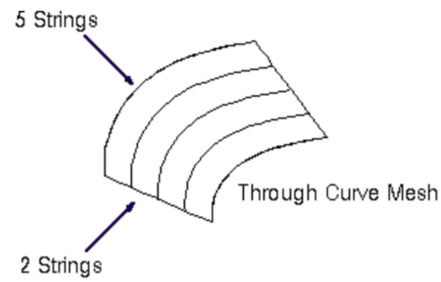


Through Curves: if three or more strings are roughly parallel.

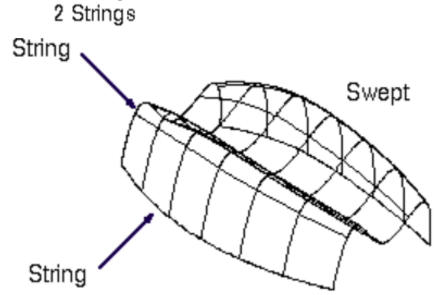


If the construction geometry contains two or more strings (curves, faces, edges) that are roughly parallel to each other, and one or more section strings that are roughly perpendicular to the first set of curves (guides), you can try using one of these following options to build a freeform surface.

Through Curve Mesh: used if at least four section strings exist with at least two strings in each direction (parallel and perpendicular).

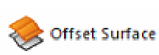


Swept: used if at least two section strings are roughly perpendicular (from **Menu**, choose **Insert** → **Sweep**).

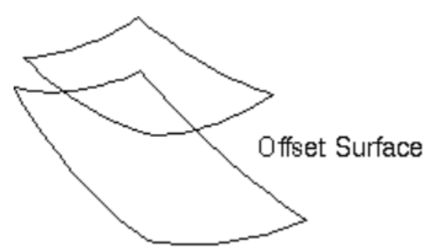


7.1.3 Creating Freeform Features from Faces

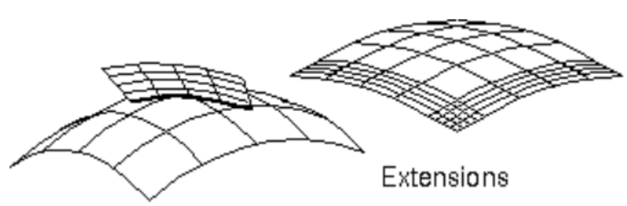
If the construction geometry contains a sheet or face, you may be able to use one of the following two options to build the freeform surface.



Offset Surface: use this option if you have a face to offset. (From **Menu**, click on **Insert** → **Offset/Scale)**



 **Extension:** use this option if you have a face and edges, edge curves, or curves on the face. (Click on the **Insert** → **Flange Surface** → **Extension**)

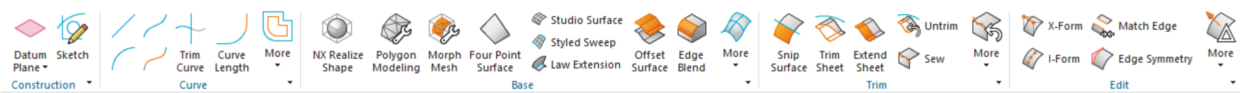


7.2 FREEFORM FEATURE MODELING

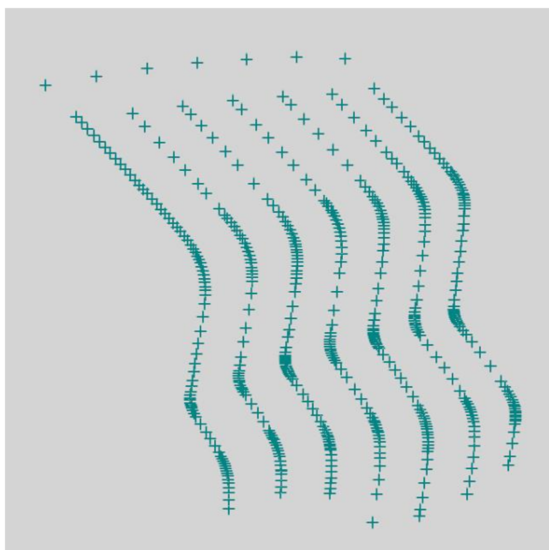
Let’s do some exercises on freeform modeling with structured points, a point cloud, curves and faces. Structured points are a set of point’s defined rows and columns. A point cloud has a set of scattered points that form a cloud.

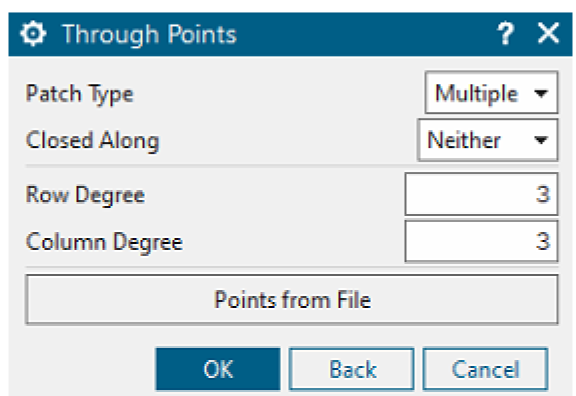
7.2.1 Modeling with Points

- Open the file **freeform_thrupoints.prt**
- Click **Shape Studio** in **Application** and change the **Home** Toolbar like below



You will see seven rows of points.



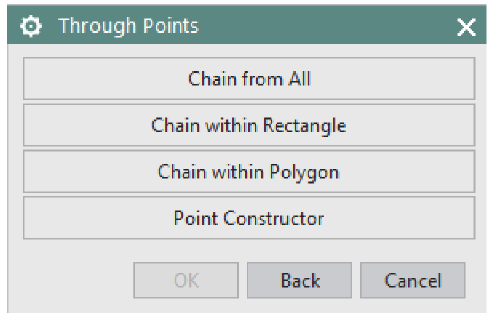


- Choose **Insert** → **Surface** → **Through Points**

The dialogue box will pop up as shown in the right.

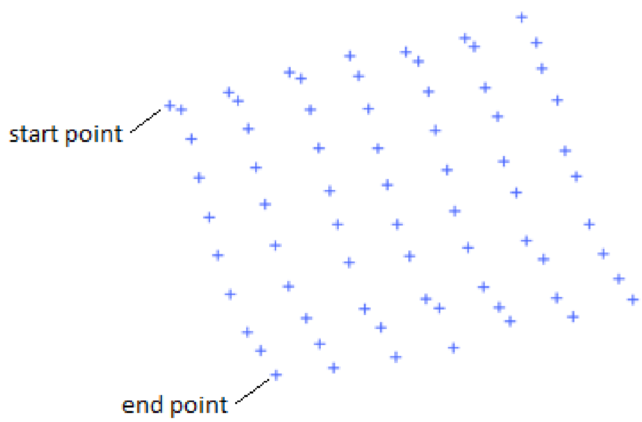
- For **Patch Type**, select **Multiple**

- For **Closed Along → Neither**
- For **Row Degree** and **Column Degree**, enter **3.**
- Click **OK**



The next dialogue box will be as shown in the right figure.

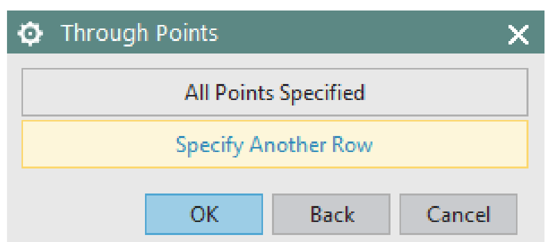
- Click **Chain from All**
- Select the top starting point and the bottom ending point of the left most row as shown in the following figure



The first row of points will be highlighted.

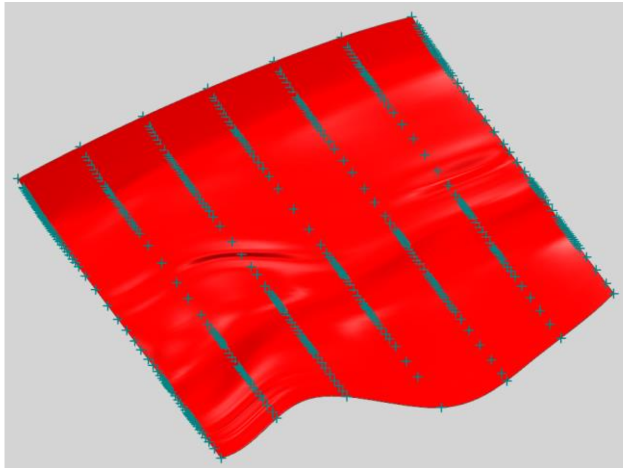
- Repeat the same procedure to select the first four rows of points.

After that, a window should pop-up asking if all points are specified or if you want to specify another row.



- Select **Specify Another Row** until all rows are specified
- When all the rows are specified, choose **All Points Specified**
- Click **Cancel** on the **Through Points** window

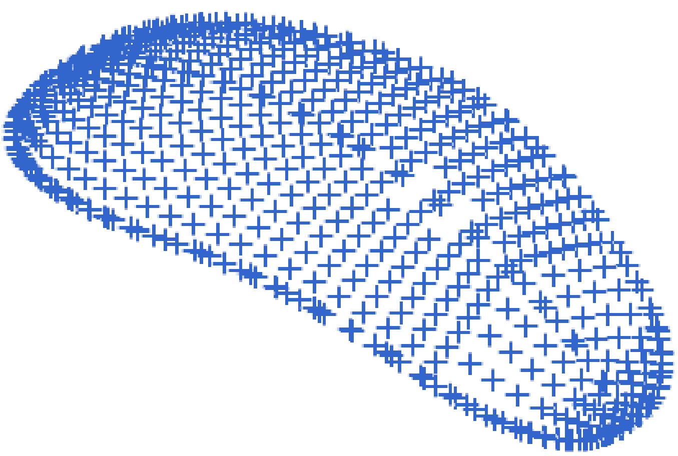
You will see the surface as shown below.



7.2.2 Modeling with a Point Cloud

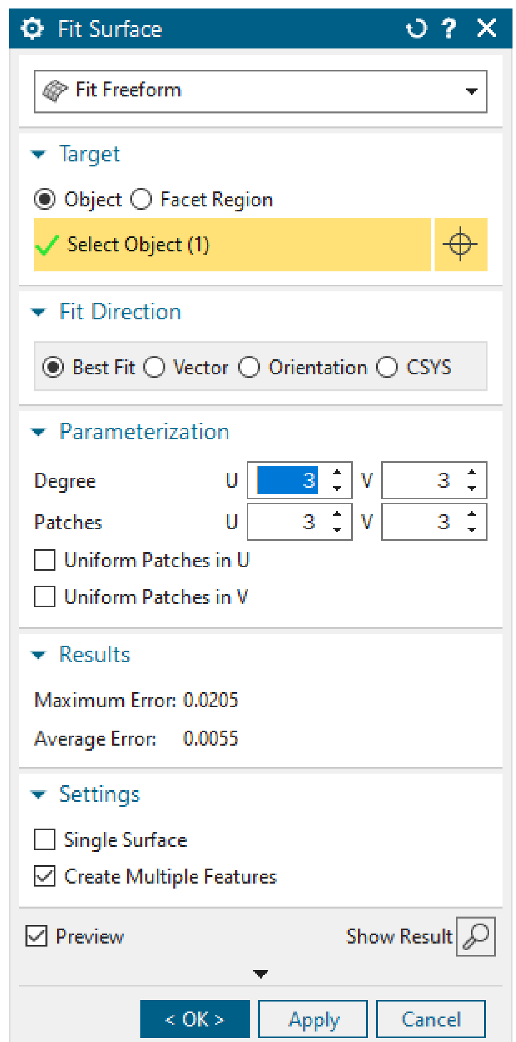
- ➢ Open the file named **freeform_cloud.prt**

The point cloud will be seen as follows.



- ➢ Choose **Insert** → **Surface** → **Fit Surface**

The right dialogue box will appear.

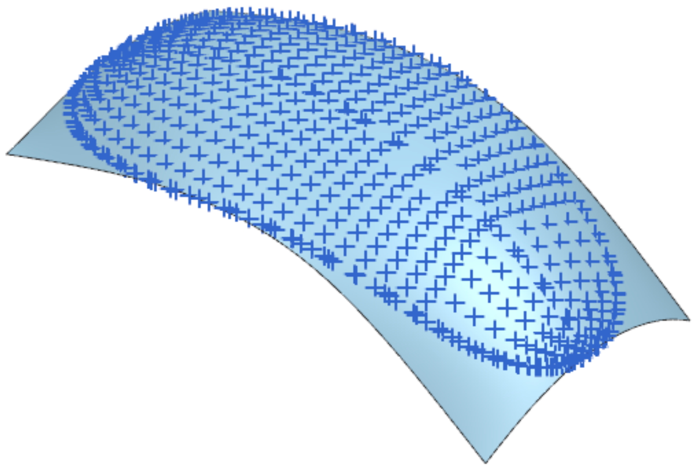


- ➢ Select all the points on the screen by clicking on the point cloud.
- ➢ In the **Fit Direction** drop-down menu, choose **Best Fit** for. This matches the point cloud coordinate system with original system

➢ Change the default values for **U** and **V** degrees to 3

➢ Click **OK**

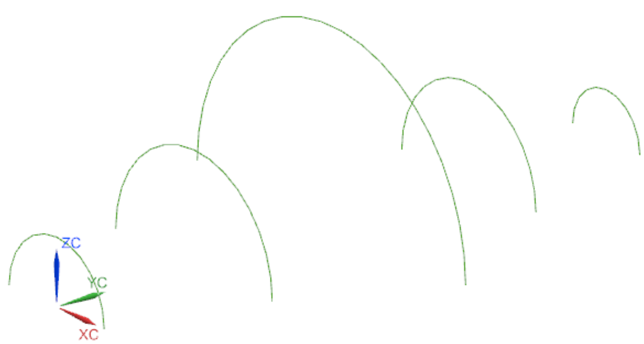
The final surface will look like the following.



7.2.3 Modeling with Curves

➢ Open the file named **freeform_thrucurves_parameter.prt** where we will create a surface using through curve option.

The curves will be seen as in the figure below.



➢ Choose **Insert → Mesh Surface → Through Curves**

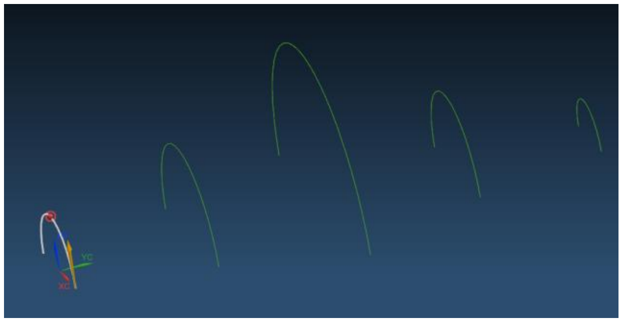
OR

➢ Click on this Icon 

on the Toolbar

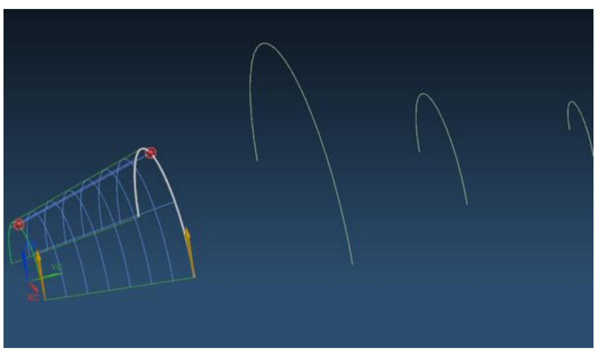
➢ Select the first **section string** as shown below. Be sure to select somewhere on the left side of the arc.

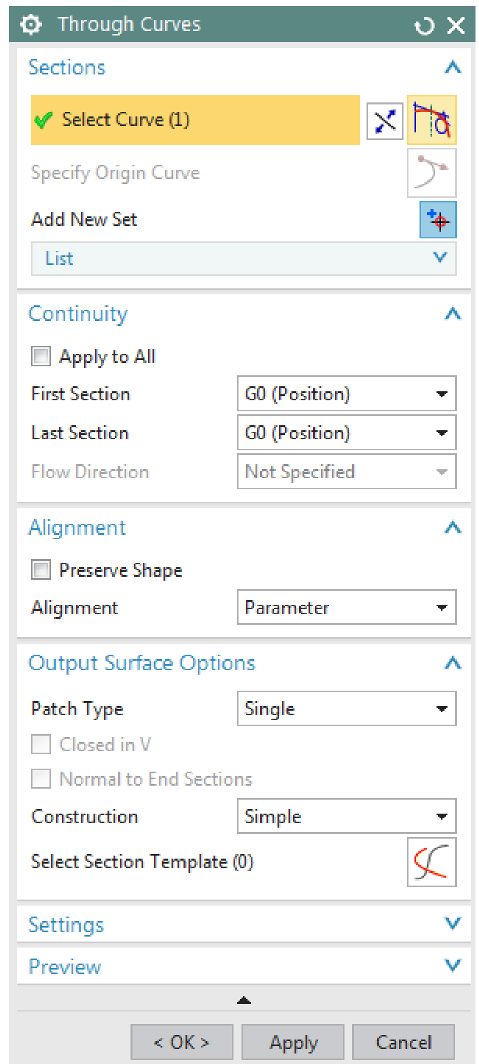
A direction vector displays at the end of the string.



➢ Click the middle mouse button **MB2** or click **Add New Section** 

➢ Click on the next curve similar to first one and click the middle mouse button **MB2**. You can see a surface generated between the two curves as shown in the figure





➢ Repeat the same procedure to select the remaining strings. Remember to click **MB2 (or Add New Set)** after selecting each curve.

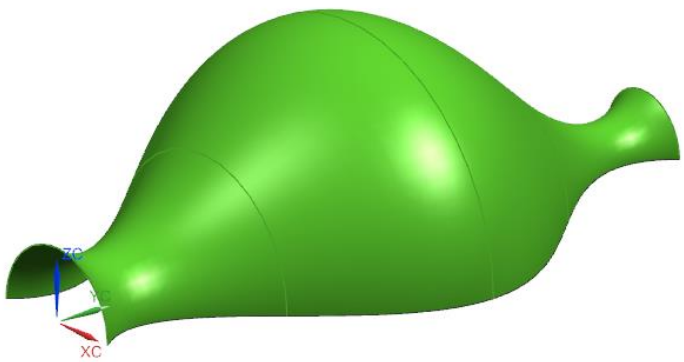
➢ For **Alignment** → **Parameter**

➢ For **Patch Type** → **Single**

➢ For **Construction** → **Simple**

When the *Simple* option is selected, the system tries to build the simplest surface possible and minimize the number of patches.

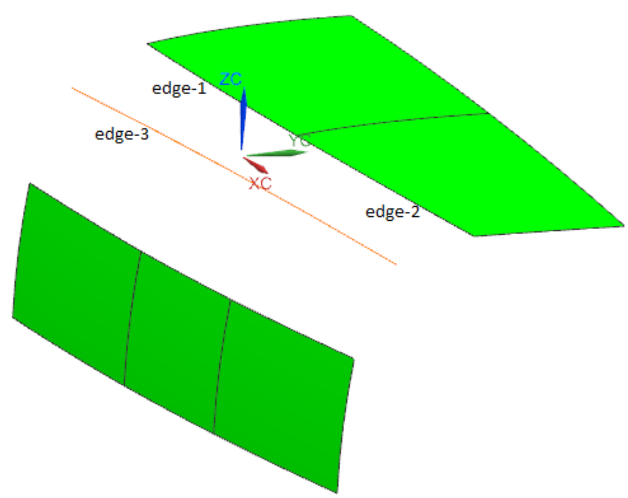
➢ Click **OK**



7.2.4 Modeling with Curves and Faces

➢ Open the file named **freeform_thrucurves_faces.prt**

The curve and faces will be seen as shown on the right.



➢ Choose **Insert** → **Mesh Surface** → **Through Curves**

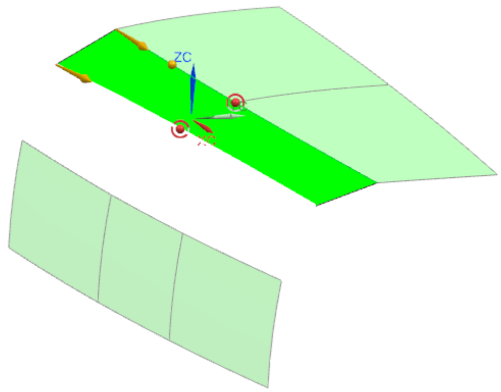
➢ Select edge-1 of the top plane

➢ Select edge-2 and click **MB2**

➢ Select edge-3

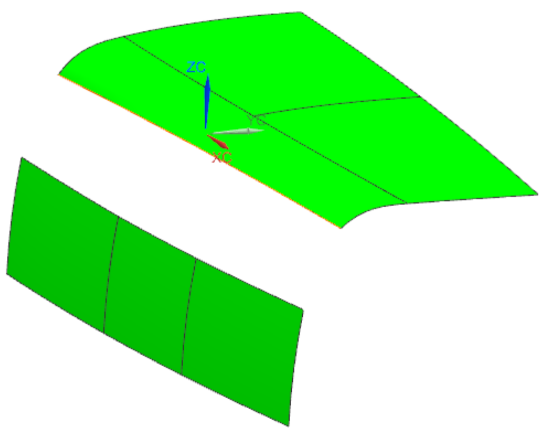
➢ In the Dialog box, under the **Alignment** section, uncheck the **Preserve Shape** check box

You would get the following shape displayed on your screen.



Make sure that all the arrows are pointing in the same direction (if they are not, double click on either of the arrows to flip its direction).

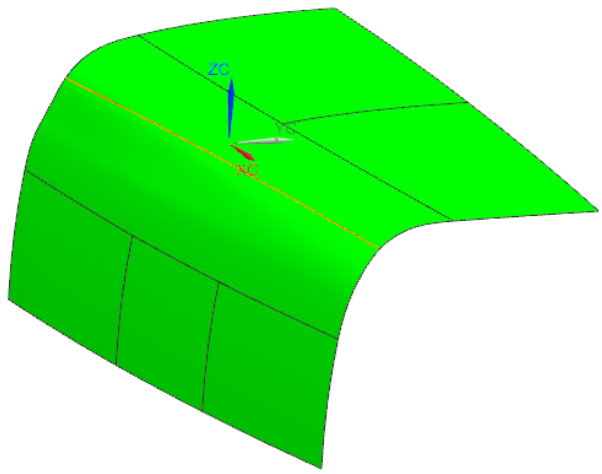
- In the **Alignment** dialog box choose **Parameter**
- In the **Continuity** dialog box, **for First Section**, select **G2 (Curvature)** option and select the two patches of the top surface
- Click **APPLY**



- Now select edge-3 and click **MB2**
- Select the three edges of the lower plane
- Change the option to **G2 (Curvature)** in the **Continuity** dialog box for **First Section**
- Select the surface you just created and click **MB2**

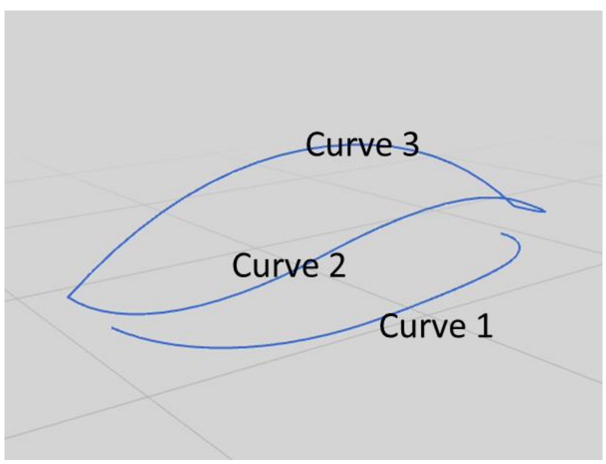
- For **Continuity** of the **Last Section**, choose **G2 (Curvature)** and then select the bottom three patches as references
- Click **OK** to exit

The final freeform surface should be seen as shown below.



7.3 EXERCISES

7.3.1 An Exercise on Curves



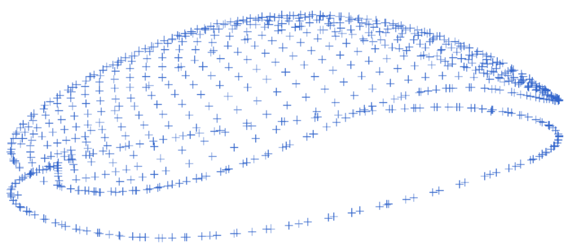
The above figure shows three curves created using points as “Control Points” or “Through Points”. The corresponding points and modeling type of each curve are listed in the following table.

	Curve 1	**Curve 2**	**Curve 3**
Points	(0, 0, 0) (0.117, -0.765, 0) (1.953, -0.525, 0) (2.5, -0.196, 0) (2.5, 0, 0)	(-0.191, 0, 0.181) (-0.191, -0.757, 0.181) (1.435, -0.709, 0.577) (2.609, -0.415, 0.147) (2.610, 0, 0.147)	(-0.191, 0, 0.181) (0.591, 0, 0.596) (1.632, 0, 0.622) (2.610, 0, 0.147)
Type	Control Points with 2nd degree		Through Points with 2nd degree

(a) Create the ruled surface between Curve-1 and Curve-2.
(b) Extrude Curve-3 along the +Y direction to create a reference surface. Then create the surface between Curve-2 and Curve-3, this surface should have G1 (Tangent) continuity to the reference surface you just extruded.

Tips: You can import those points from a text file to NX. First, save the points coordinates into a text file. Then, use NX File -> Import -> Points from File to import it.

7.3.2 An Exercise on Surfaces



Given 2 points sets that stored in the files of “Fit curve.pts” and “Fit surface.pts” (available in the file folder).

(a) Import those two sets of points into NX. (Figure above shows the expected result)
(b) Create a spline curve based on points in the file of “Fit curve.pts”. You can use **Fit Curve** to create it, adjust the number of **Degree** and **Segments** to get a better fitting.
(c) Create a freeform surface based on points in the file of “Fit surface.pts”. You can use **Fit Surface** to create it, adjust the number of **Degree** and **Patches** to get a better fitting.
(d) Use the spline curve as the boundary to trim the freeform surface along Z direction. The expected result is like the upper surface of a computer mouse.

7.3.3 Design a Computer Mouse

Model a computer mouse similar to the one shown below (feel free to search for more images as references), or you can come up with a new design and then model it. As a hint, create some boundary curves on different datum planes and use them to create freeform surfaces.

7.3.4 Design a Sport Water Bottle

Design a sport water bottle and use freeform features in NX (curves and surfaces) to model it.

CHAPTER 8 – FINITE ELEMENT ANALYSIS

Finite Element Analysis (FEA) is a practical application of the Finite Element Method (FEM) for predicting the response behavior of structures or fluids to applied factors such as forces, pressures, heats, and vibrations. Usually, the process starts with the creation of a geometric model. Then the model is subdivided (meshed) into small pieces (elements) of simple geometric shapes connected at specific node points. The material properties and the boundary conditions are applied to each element. Finally, software such as NX solves this FEA problem and outputs results and visualizations. It helps engineers to have a better understanding of the product performance before it is fabricated and tested.

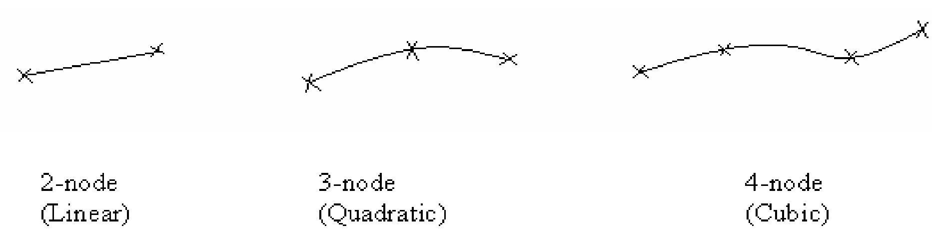
Some of the applications of *FEA* are *Structural Analysis, Thermal Analysis, Fluid Flow Dynamics*, and *Electromagnetic Compatibility*. Of these, *FEA* is most commonly used in structural and solid mechanics applications for calculating the mechanical behavior (e.g. stresses and displacements). These are often critical to the performance of the hardware and can be used to predict failures. In this chapter, we are going to deal with the structural stress and strain analysis of a solid part.

8.1 OVERVIEW

8.1.1 Element Shapes and Nodes

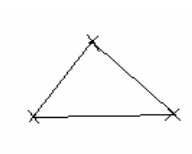
The elements can be classified into different types based on the number of dimensions and the number of nodes in an element. The following are some of the types of elements used for discretization.

<u>One-dimensional elements</u>



Two-dimensional elements

Triangular:

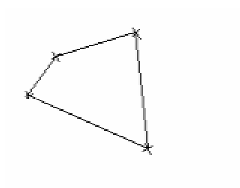


3-node
(Linear)

6-node
(Quadratic)

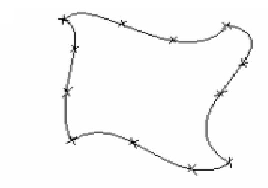
10-node
(Cubic)

Quadrilateral:



4-node
(Linear)

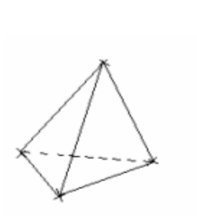
8-node
(Quadratic)



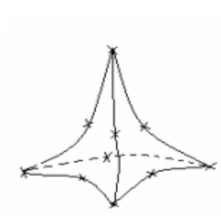
12-node
(Cubic)

Three-dimensional elements

Tetrahedral (a solid with 4 triangular faces):



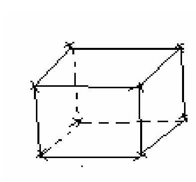
4-node
(Linear)



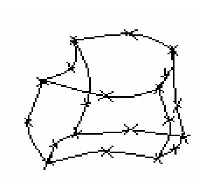
10-node
(Quadratic)

20-node
(Cubic)

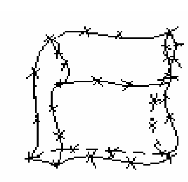
Hexahedral (a solid with 6 quadrilateral faces):



8-node
(Linear)



20-node
(Quadratic)



32-node
(Cubic)

Types of nodes

- Exterior nodes
  - Corner nodes
  - Side nodes
- Interior nodes

Usually, FEA can have a more accurate solution as the size of finite element becomes smaller, but the computing time becomes longer as well.

8.1.2 Solution Steps

Starting the Simulation: You can select a solver from one of these: NX Nastran, NX Nastran Acoustic, NX Nastran Vibro-Acoustic, NX Nastran Design, Samcef, NX Thermal/Flow, Simcenter Electronic Systems Cooling, Simcenter Space Systems Thermal, NX Multiphysics, Simcenter Acoustics BEM, MSC Nastran, Ansys, Abaqus, and LS-DYNA. In addition, you can choose the type of analysis to be performed. In this tutorial, only Structural Analysis will be covered with NX Nastran Design.

Choosing the Material Properties: This allows you to change the material properties that will be assigned to the model. For example, if we use steel to manufacture the impeller, we can enter the material properties such as density, Poisson's ratio, etc. These material properties can also be saved in the library for future use or can be retrieved from Library of Materials.

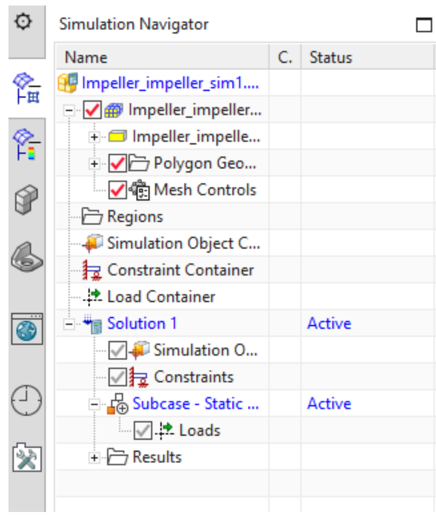
Applying the Loads: This option allows you to apply different types of loads, such as forces or pressures on the solid along with the directions and magnitudes.

Applying the Boundary Conditions: Simply speaking, boundary conditions constrain the degrees of freedom of the elements. Some elements can be rotationally fixed and some can be constrained from translational movement.

Meshing the Bodies: This is used to discretize the model into finite elements. Normally, we select tetrahedral shapes of elements for approximation. You can still select the 2-D and 1-D elements depending on the situation and requirements by choosing these options from the drop-down menu.

Solution and Results: This is the command to solve all the governing equations by the selected solver and all the above options. This solves and gives the results of the analysis of the problem.

8.1.3 Simulation Navigator



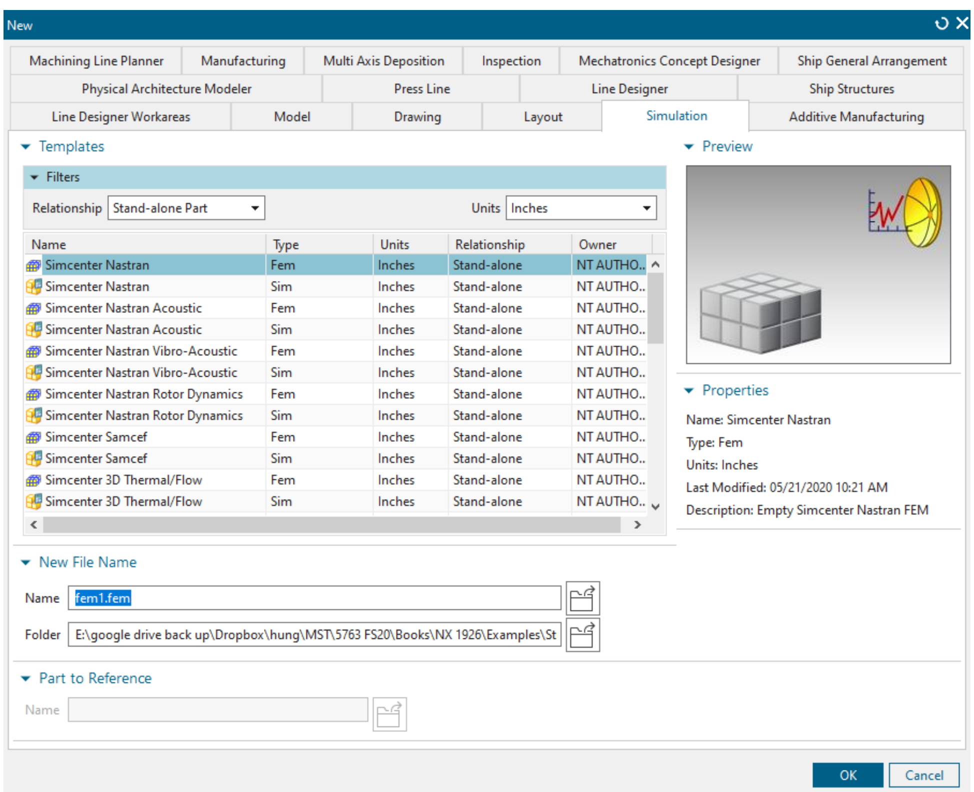
The *Simulation Navigator* provides the capability to activate existing solutions, create new ones, and use the created solution to build mechanisms by creating and modifying motion objects. To display the *Simulation Navigator*,

- ➢ Click the **Simulation Navigator** tab in the **Resource bar** as shown in the figure

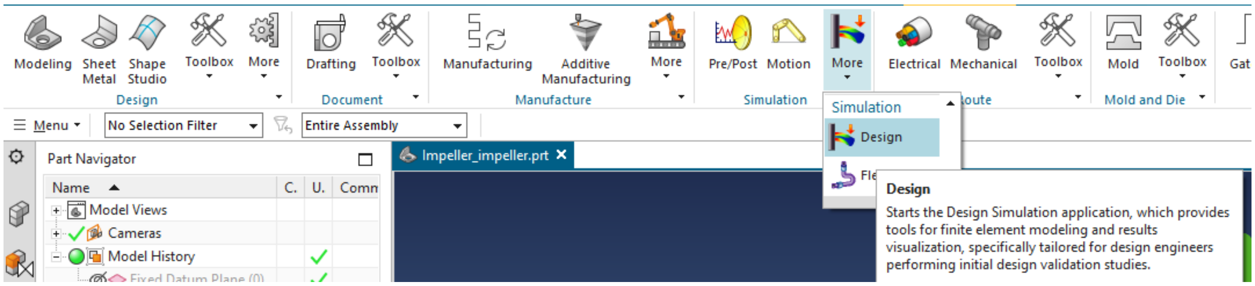
It shows a list of the simulations created for the model. In each simulation, it displays a list of loads, boundary conditions, types of meshes, results, reports generated and so on.

8.2 SIMULATION CREATION

- ➢ Copy and paste the file **Impeller_impeller.prt** into a new folder to avoid changes being made to the assembly
- ➢ Click on **New** → **Simulations** if the part is NOT already opened in the NX window

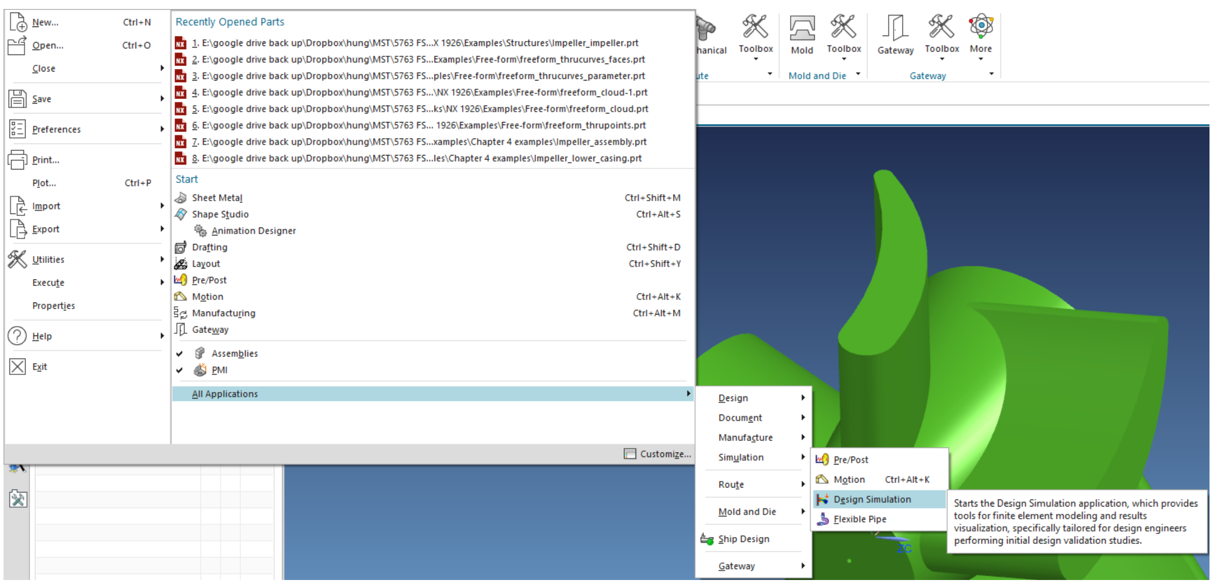


- ➢ Open this newly copied file
- ➢ If the part is already opened in NX, then from the top ribbon bar, click on **Application** → **Design**



Or

- ➢ click on **File** → **All Applications** → **Simulation** → **Design Simulation**



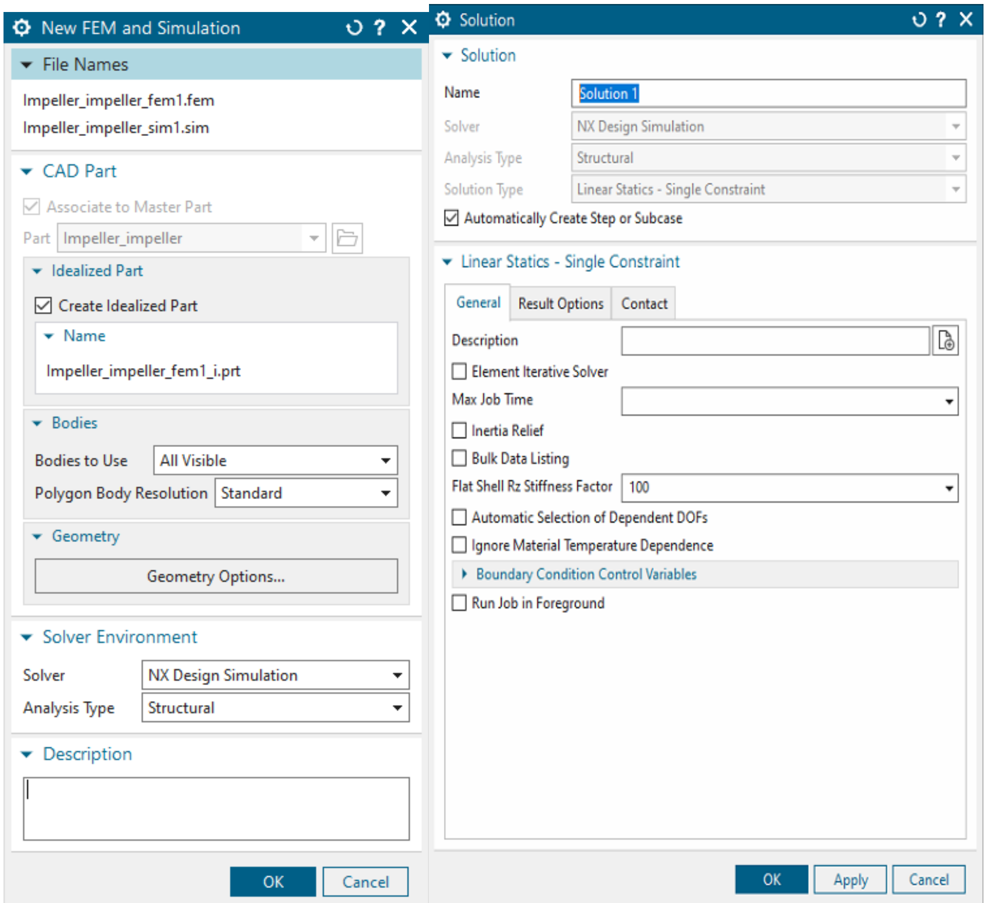
When you first open any file in *Design Simulation* module, it will automatically pop up the New FEM and *Simulation* dialog to create a simulation.

- ➢ In the popup dialog, click OK to create a new simulation.

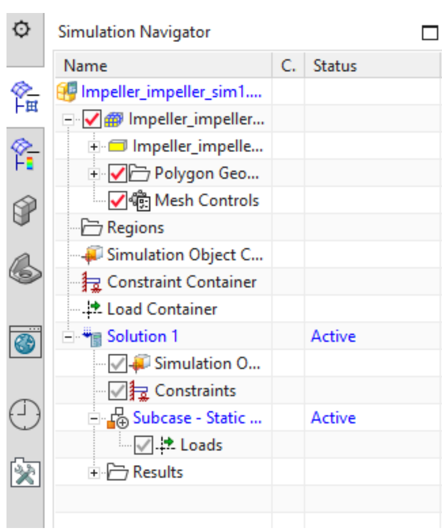
Then in the next popup *Solution* window, you can select the *Solver* and the *Analysis Type*.

The default Solver type is *NX Nastran Design* and Analysis type is *Structural*.

➢ Choose **OK** to create a new **Create Solution** called Solution 1, which will be displayed in the **Simulation Navigator.** Here we keep the other items as default.

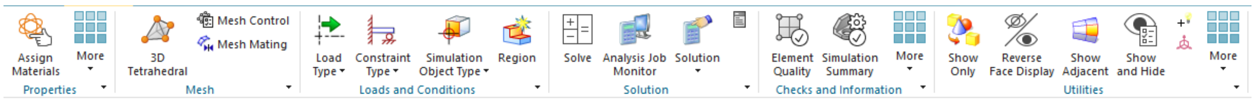


Now the Simulation Navigator will look like the following figure.



8.3 MATERIAL PROPERTIES

The next step is to assign the material properties to the solid model for this simulation. Because we do not have any data in the library to retrieve for standard material, we will create one. Let us assume that we will use steel to manufacture the impeller.



- ➢ Click on **Assign Materials** from the ribbon bar shown above

The *Assign Material* window will pop up. You have the option of choosing the pre-defined materials from the *Library* or create a new material.

- ➢ Select the **Impeller** for **Select Body**

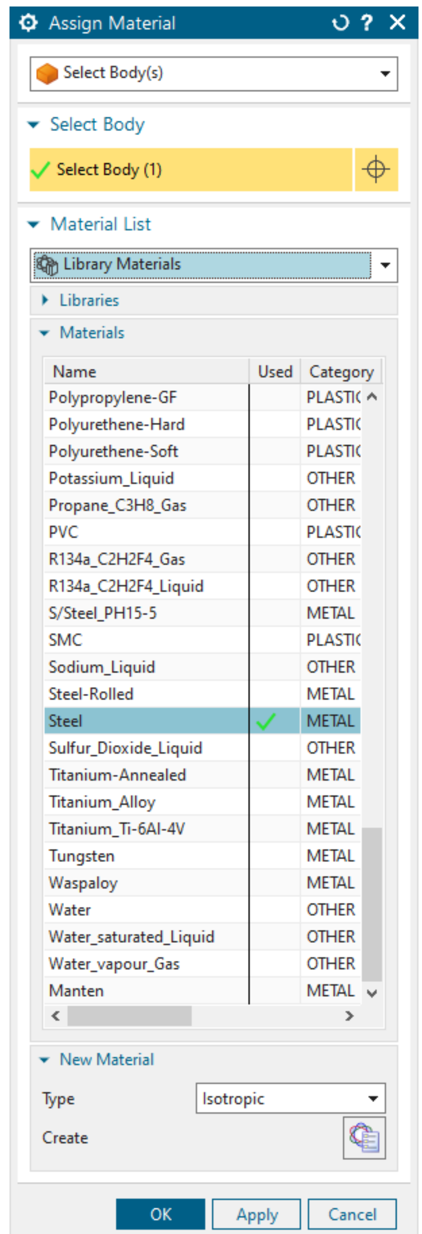
- ➢ Assign the **Steel** as the material (Double left-click to see the detail mechanical properties)

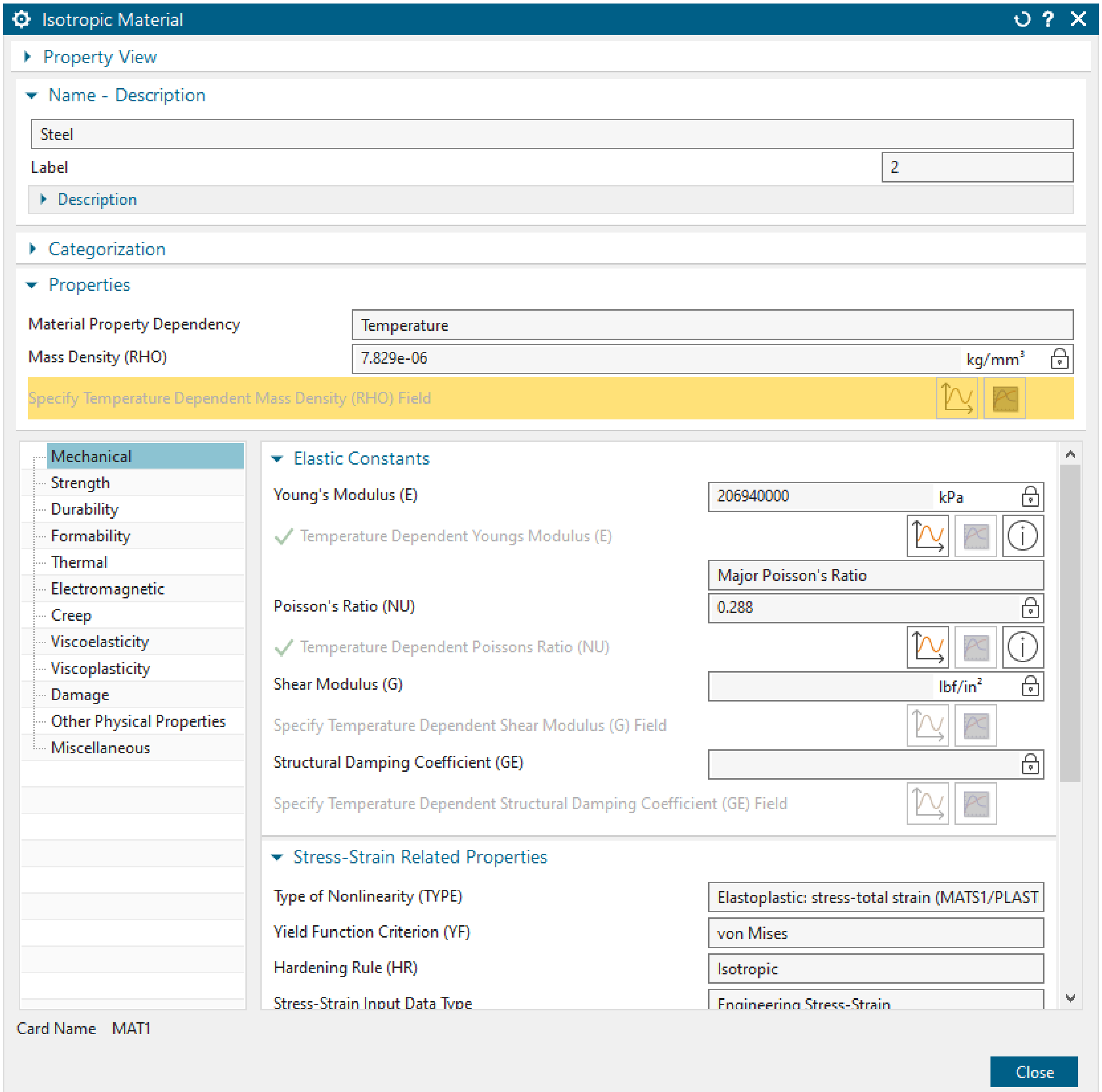
The name and values as shown in the following figure. Pay attention to the units.

(Note that 7.829e-6 represents 7.829×10^{-6})

- ➢ Choose **Close** to exit the **Isotropic Material** window

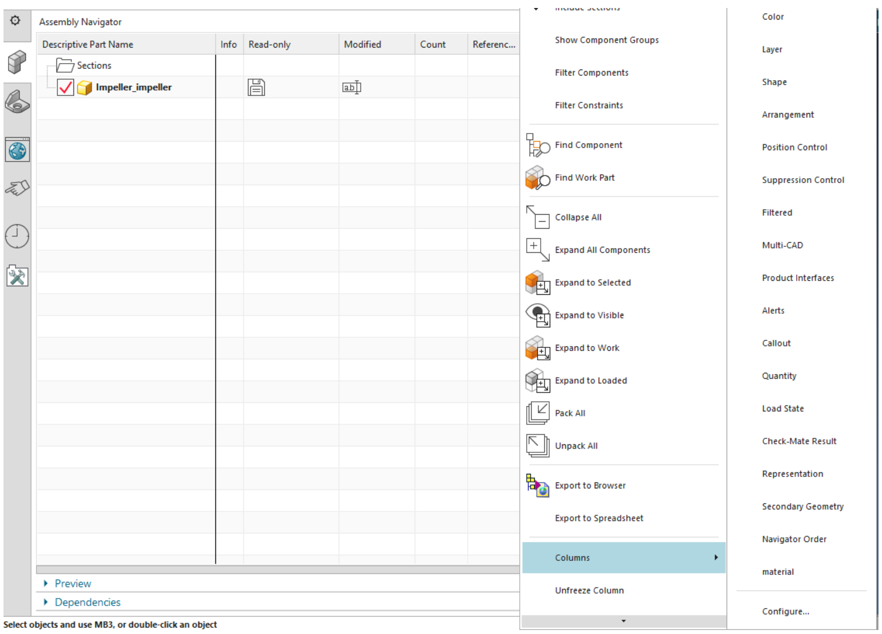
Now we have assigned the material to the impeller model.



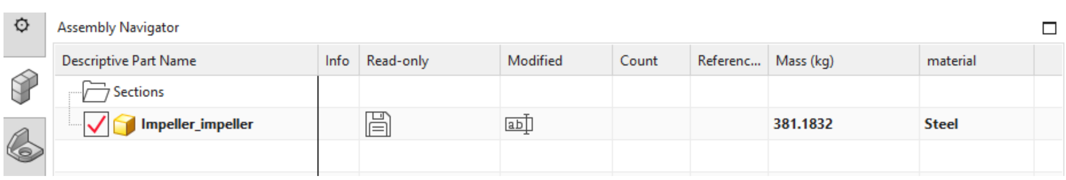

Isotropic Material
Property View
Name - Description
Steel
Label
2
Description
Categorization
Properties
Material Property Dependency
Temperature
Mass Density (RHO)
7.829e-06
kg/mm³
Specify Temperature Dependent Mass Density (RHO) Field
Mechanical
Strength
Durability
Formability
Thermal
Electromagnetic
Creep
Viscoelasticity
Viscoplasticity
Damage
Other Physical Properties
Miscellaneous
Elastic Constants
Young's Modulus (E)
206940000
kPa
Temperature Dependent Youngs Modulus (E)
Major Poisson's Ratio
Poisson's Ratio (NU)
0.288
Temperature Dependent Poissons Ratio (NU)
Shear Modulus (G)
lbf/in²
Specify Temperature Dependent Shear Modulus (G) Field
Structural Damping Coefficient (GE)
Specify Temperature Dependent Structural Damping Coefficient (GE) Field
Stress-Strain Related Properties
Type of Nonlinearity (TYPE)
Elastoplastic: stress-total strain (MATS1/PLAST
Yield Function Criterion (YF)
von Mises
Hardening Rule (HR)
Isotropic
Stress-Strain Input Data Type
Engineering Stress-Strain
Card Name MAT1
Close

To display the Mass of your model, open **Impeller_impeller.prt** in **Modeling Application**

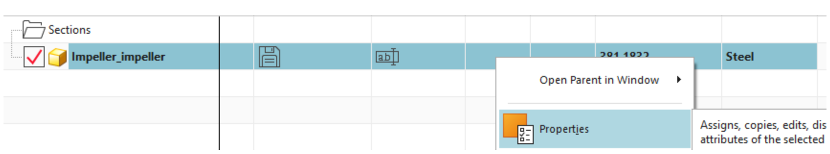
- Assign Material: Select **Menu** → **Tools** → **Materials** → **Assign Material**
- Select the model and Assign the **Steel** as the material and **OK**
- Right-click in the Window of Assembly Navigator and choose **material** (this tag can be created using **Configure**) and **Mass (kg)** in **Columns**



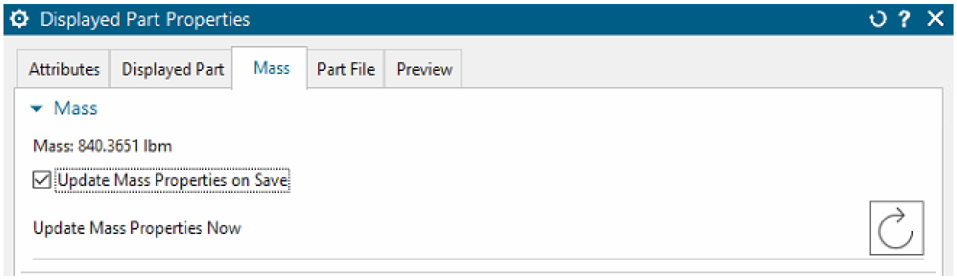
Now, you should be able to see Mass and Material like figure below:



If the Mass is not correct, try right-click on the part and open **Properties**.



Use **Update Mass Properties Now** to display correct mass in the Mass option



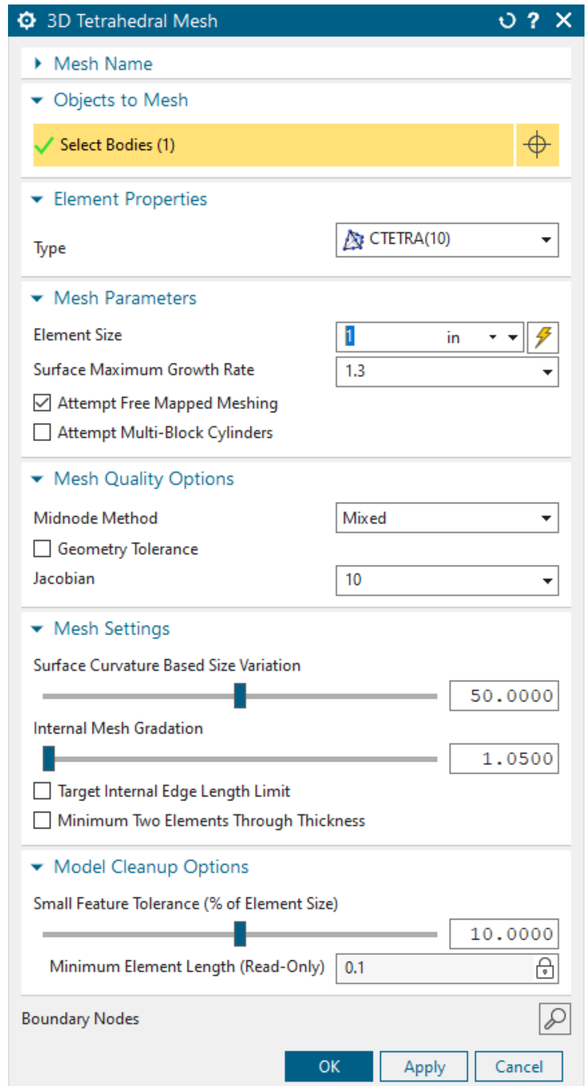
8.4 MESHING

The *Mesh* option discretizes the model into small elements.

- Click on the **3D Tetrahedral** icon 

A window will pop up asking for the type and size of the elements.

- First, click on the **impeller model** on the screen for **Select Bodies**.

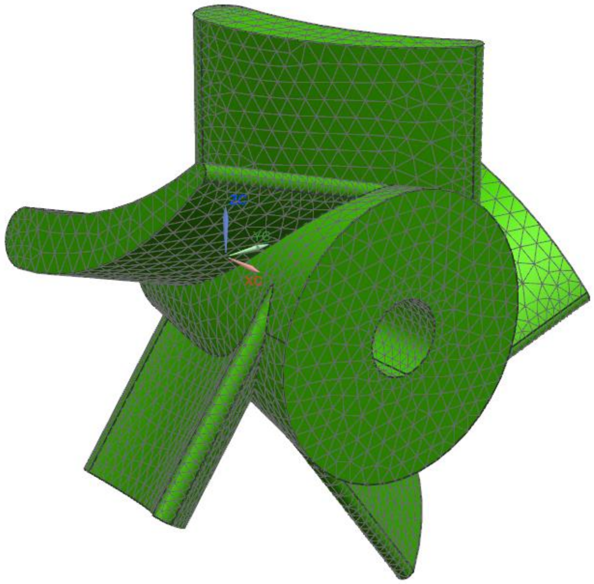


Then, there are two types of *Tetrahedral Elements* available in NX. One is 4-node and the other is 10-node.

- Choose the **Type** to be **CTETRA(10)**
- Enter the **Element Size** as **1.0 inch**
- Click **OK**

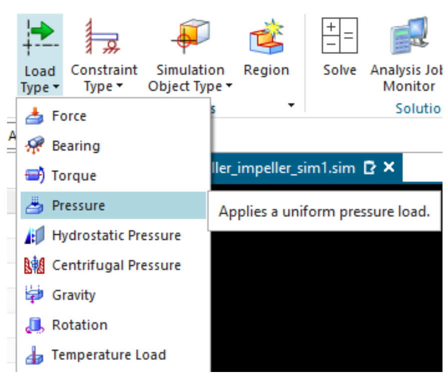
You can find the model with small tetrahedral elements. It will look like the figure shown below.

Note: While meshing the solid, there is a trade-off you need to consider. If you choose a smaller element with higher nodes you will get better accuracy in your analysis than larger element. However, the time required to solve the model with smaller elements will much greater than with larger element. Hence, based on the accuracy requirement of the study and how critical the component is in terms of the end product, choose the appropriate parameters for the elements and nodes.

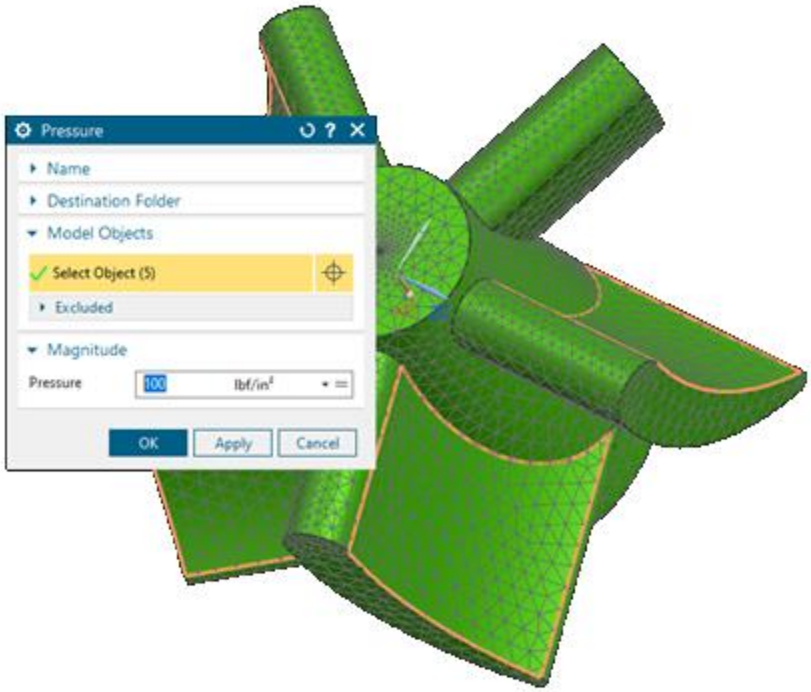


8.5 LOADS

The loads applied on the solid model should be input to the system. For the impeller, suppose the major force acts on the concave surfaces of the turbine blades. This loading can be approximated by normal pressure on all the five surfaces. Since we are not concerned about the magnitude of the load, let us take the value to be 100 lbf/in^2 inch to exaggerate the deformation of the blades.

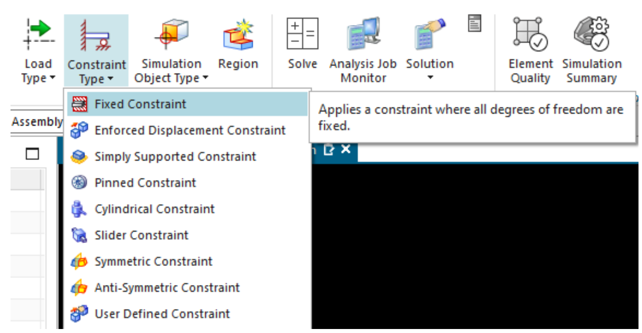


- Click on **Load Type** and choose **Pressure**
- Click on the five concave surfaces of the blades as shown in the following figure
- Enter the value for **Pressure** as **100** and keep the unit as lbf/in^2 (psi)

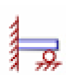


8.6 BOUNDARY CONDITIONS

The impeller rotates about the axis of the cone with the shaft as you can see in the assembly in the previous chapters. It is not fixed but our concern is the deformation of the blades with respect to the core of the impeller. The conical core is relatively fixed, and the deformations of the blades are to be analyzed accordingly.



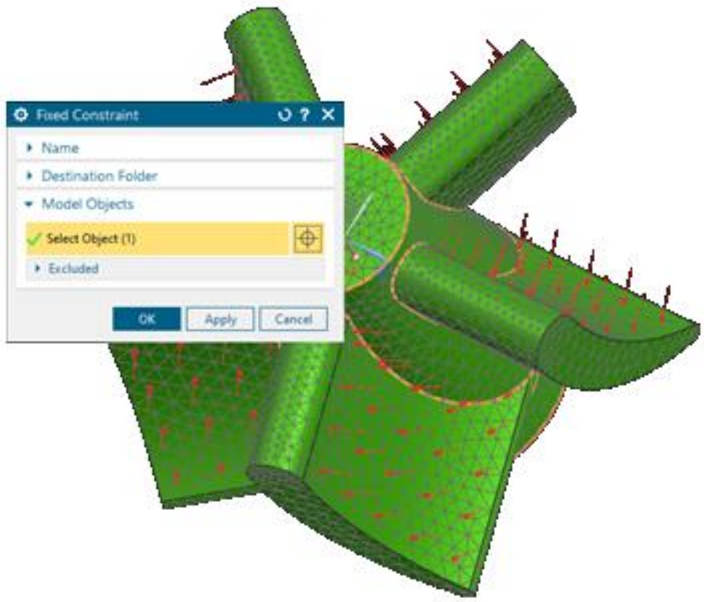
- ➢ Click on the **Constraint Type** icon



- ➢ Select the **Fixed Constraint**

This type of constraint will restrict the selected entity in six DOF from translating and rotating. You can see the different constraints available by clicking the *Constraint Type* drop-down menu on the toolbar.

- ➢ Click on the conical surface of the impeller as shown in the following figure
- ➢ Click **OK**



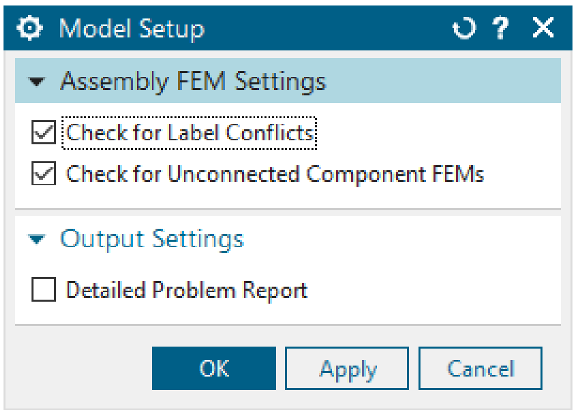
8.7 RESULT AND SIMULATION

8.7.1 Solving the Simulation

The Finite Element Model is now ready for solving and analysis. It is a good practice to first check for model completion before we get into solving the model. To check the model

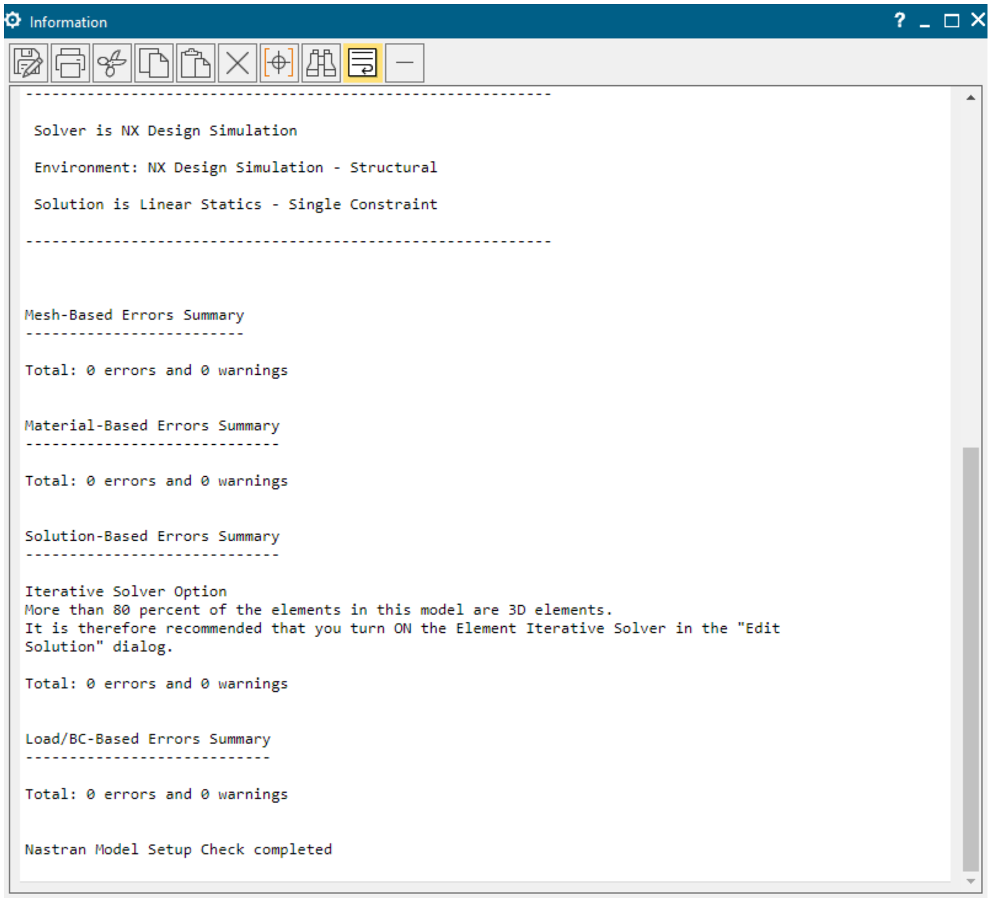
- Click on the **Menu** → **Analysis** → **Finite Element Mode Check** → **Model Setup** or click the **Model Setup** icon in the **Checks and Information** group in the ribbon bar

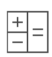
This will pop up a window as shown on the right.



- Choose **OK**

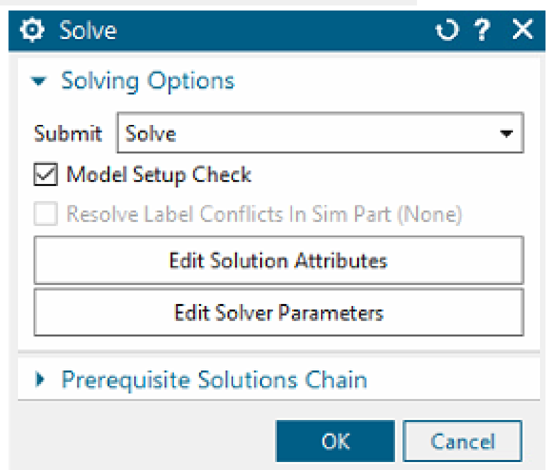
This will display the result of the *Check* operation**.** You will see any errors and warnings in a separate window. In case you get errors or warnings go back to the previous steps and complete the required things. If you do not get errors or warnings, you are ready to solve the FEA problem.



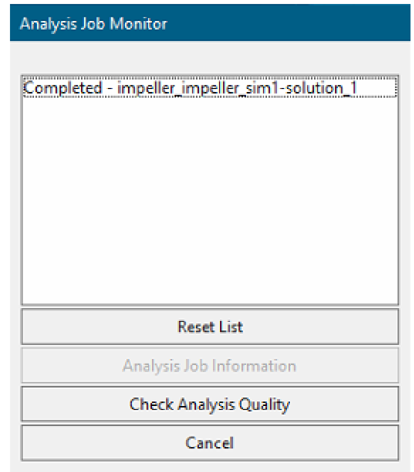
- ➢ Click on the **Solve** icon 

This will open the *Solve* window.

- ➢ Click **OK** without making any changes



It may take a while to solving the job. Wait until the *Analysis Job Monitor* window appears, showing the job to be *Completed*. While the solver is doing computations, the *Analysis Job Monitor* will show as *In Progress*

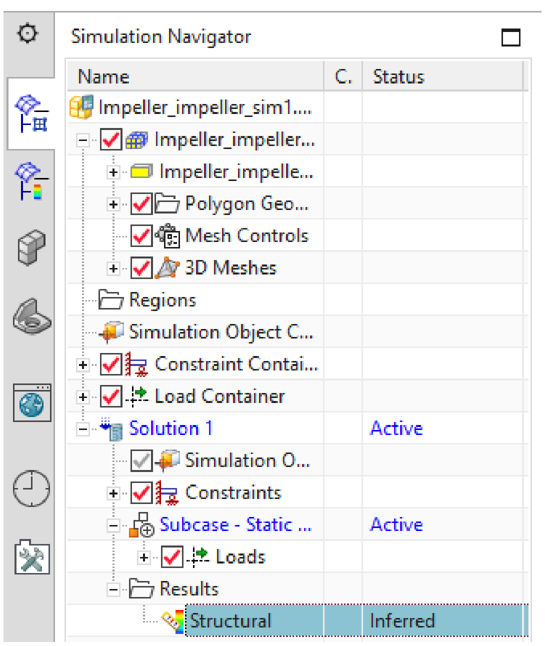


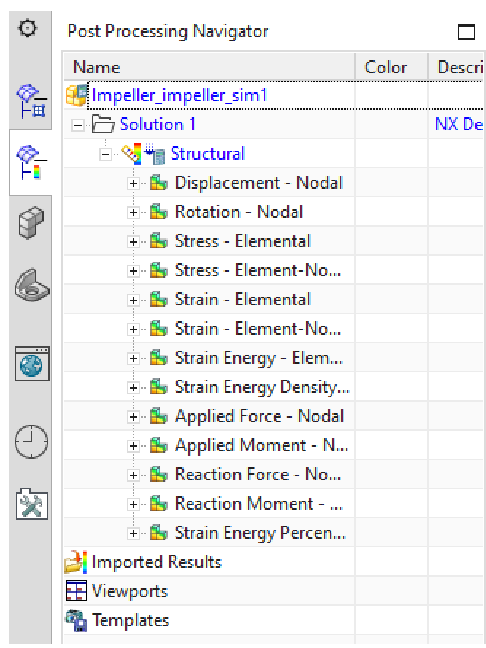
- Click on **Cancel** when the **Analysis Job Monitor** window shows **Completed**

8.7.2 FEA Result

- From **Simulation Navigator**, double click on **Structural** under **Results**
- You will be directed to the **Post Processing Navigator**

The *Post-Processing Navigator* shows all the *Solution* you just solved. If you click the '+' sign in front of the *Solution*, you will see the different analyses that have been performed on the model.

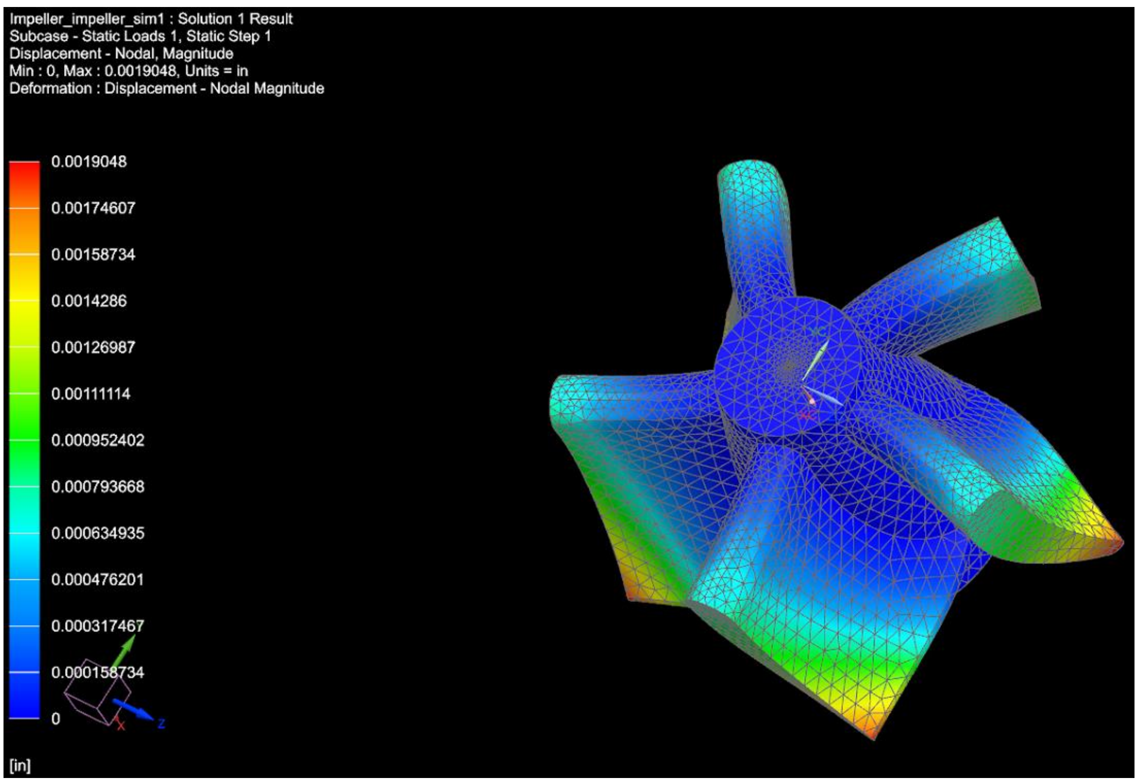


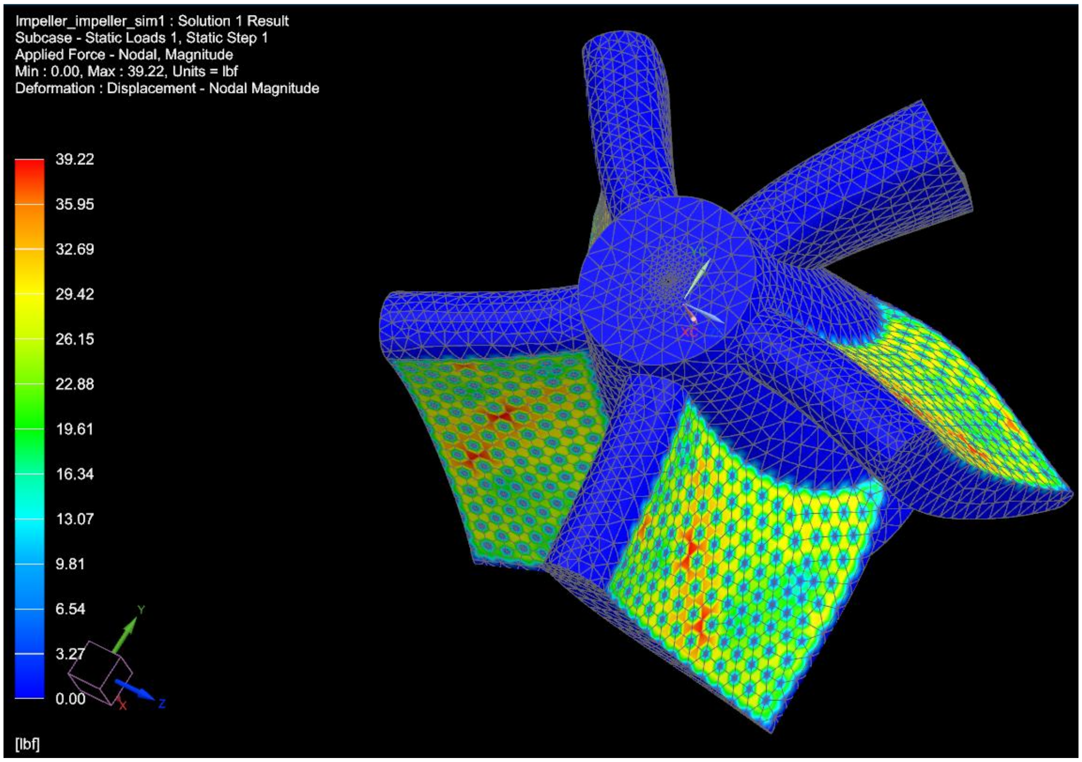


- Double-click on the **Displacement-Nodal**

The screen will now appear as shown below. You can easily interpret the results from the color-coding. The orange-red color shows the maximum deformation zones and the blue area shows the minimum deformation zones. You can observe that because the conical core is fixed, it experiences zero deformation. The analysis also shows that the maximum deformation experienced at the tip of the blades is approximately 1.9×10^{-3} inches.

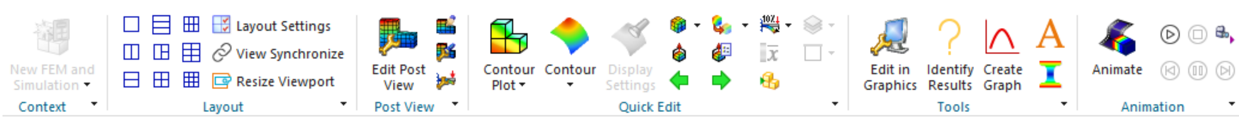
On the *Post-Processing Navigator*, you can keep changing the results by double clicking each option as shown below. You can click on the other inactive marks to see various results. Some of the other results are shown below.

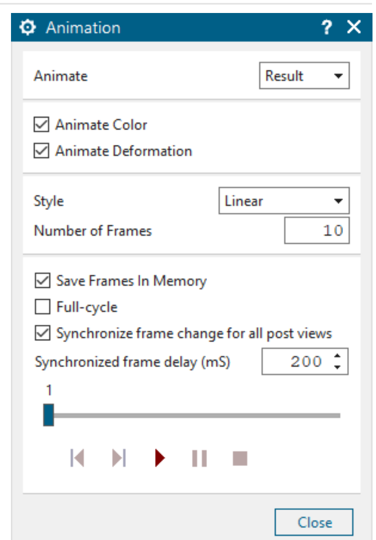




8.7.3 Simulation and Animation

Click on the **Results** tab from the top ribbon bar. A group for **Animation** can be seen on it as follows

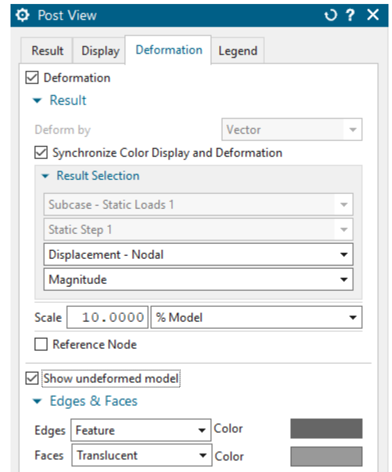




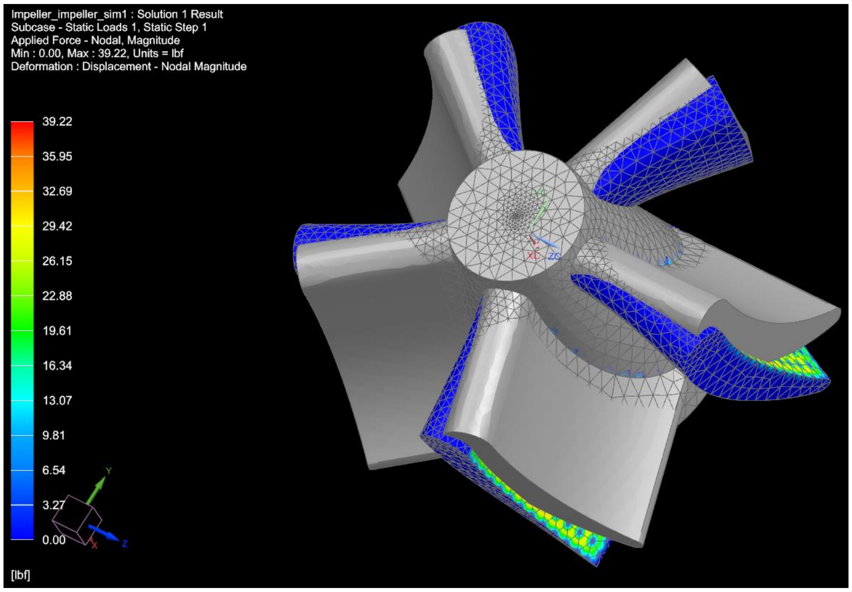
- ➢ Click on the **Animate** icon.
- ➢ In the **Animation** window, change the number of frames to **10** and click on the **Play** button  to see the animation of the deformation

You can now see an animation of how the impeller is deformed as the loads are applied to the blades.

➢ To make any setting changes in the results display, click on the **Edit Post View** icon 

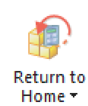


➢ Check the **Show undeformed model** and click **OK**



Now press on the **Play** button to see the animation. This will show the animation of deformation with the original shape in grey color, as shown in the figure below.

➢ Click on the **Stop** button to stop the animation

➢ Click on **Return to Home** to go back to the FEA model 

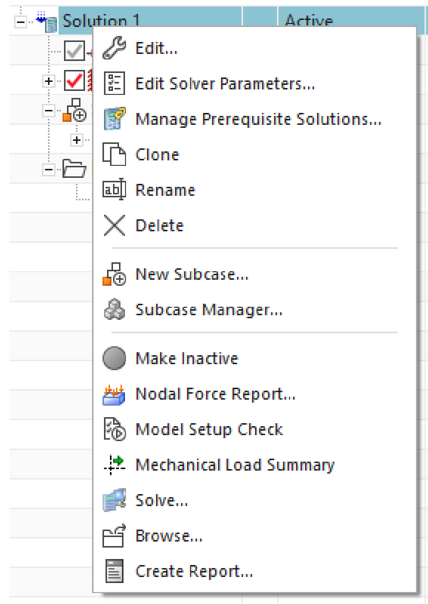
There are two ways to improve the accuracy of FEA results:

- Reduce the size of element
- Increase the order of interpolation polynomial (i.e. use quadratic or even cubic instead of linear polynomials)

Let us try creating a simulation using different settings.

➢ Right-click on **Solution 1** in the **Simulation Navigator**

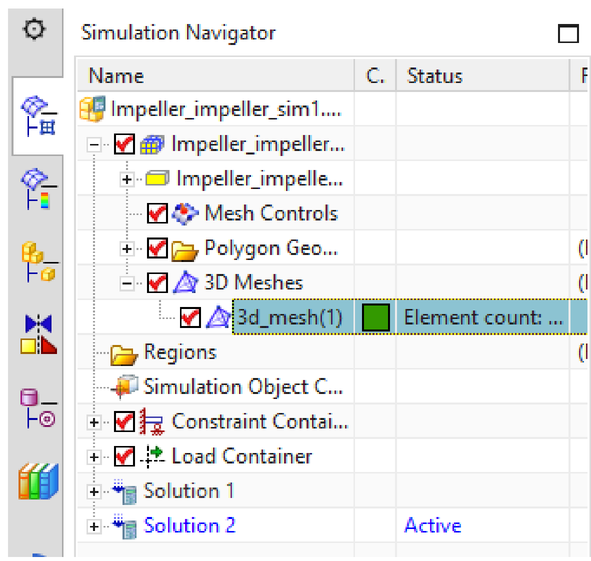
- Choose **Clone** to copy the first simulation
- Once **Copy of Solution 1** is created, rename it to **Solution 2**
- From the **Simulation Navigator**, under **3D Meshes**, double left-click on the **3D Mesh (1)**
- In the popup dialog, change the **Type** to **TETRA4**
- Click **OK**



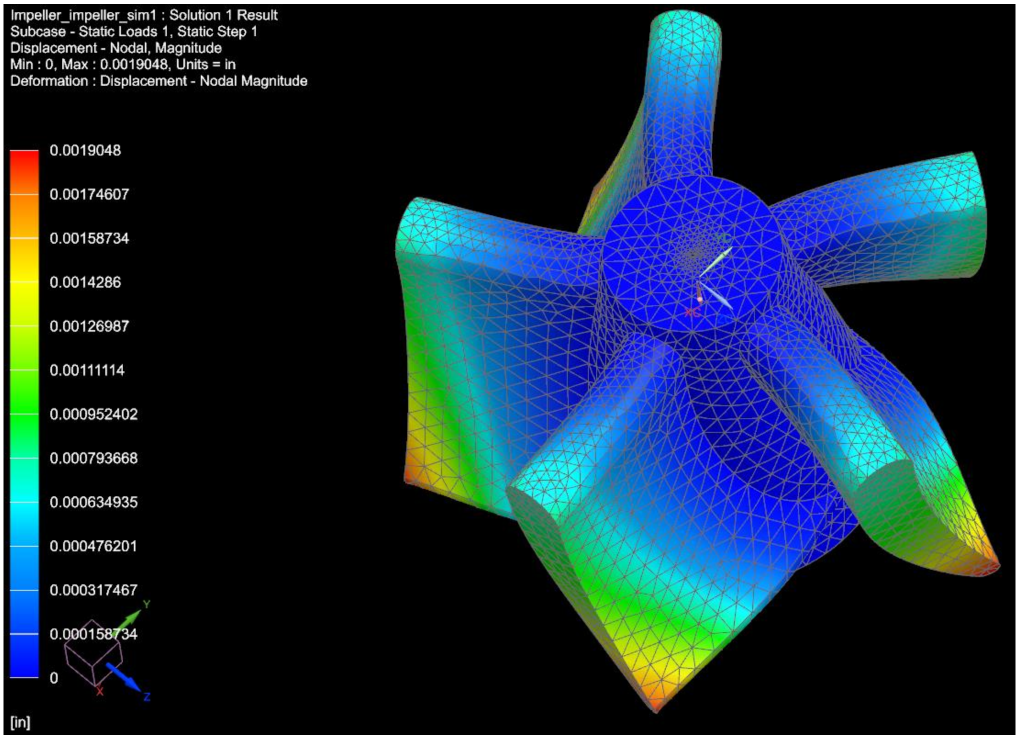
- Click on the **Solve** icon to solve the simulation
- Click **OK**

The *Analysis Job Monitor* should show the status of *Solution 2* to be *Completed***.**

- Click **Cancel**
- In the **Simulation Navigator**, double-click on **Results for Solution 2**



The figure below shows the analysis. You can observe the change in the maximum value. Save all the simulations and close the files.

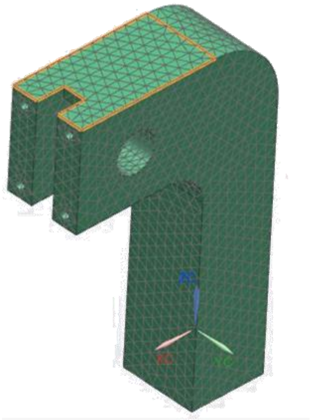


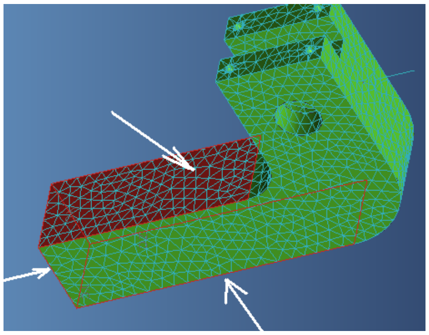
8.8 EXERCISES

8.8.1 Arbor Press Bar

Open the file '**Arborpress_L-bar.prt**' and do a similar structure analysis, considering the material as steel. For the mesh, the element size and type should be '10' and 'Tetra(10)'. For the loads, apply a normal pressure with a magnitude of 500 on the top surface as shown in the first figure below.

For the boundary conditions, fix the three flat faces (the front highlighted face, the face parallel to it at the backside and the bottom face) as marked in the second figure below.



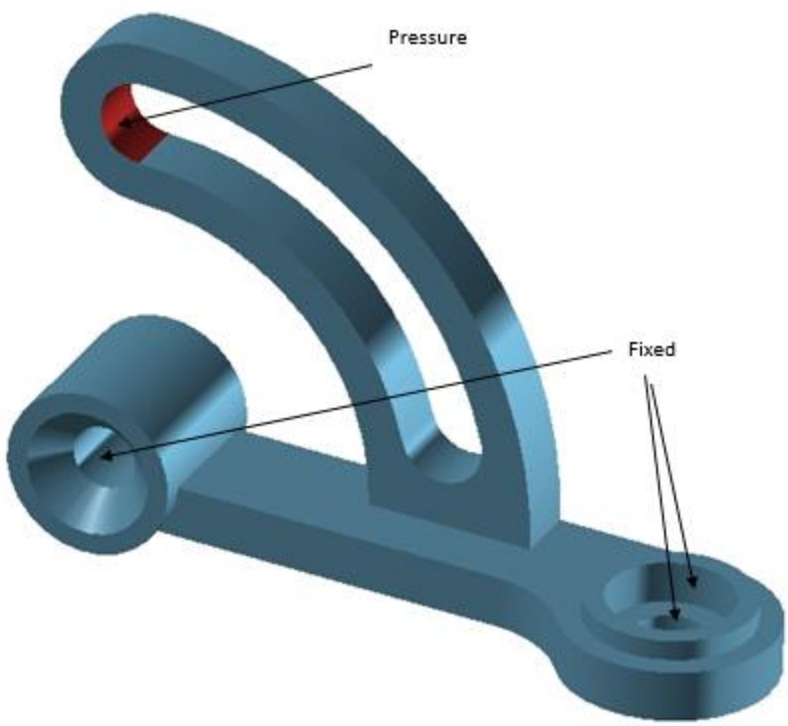


8.8.2 Rocker Arm

In this exercise, you will examine the effect of element type and mesh size on the results of finite element analysis. Open the Rocker Arm you modeled in chapter 4. Assign the following material properties: Young's modulus = 3.0×10^7 psi, Poisson's ratio = 0.29, and mass density = 7.35×10^{-4} slug/in^3. Fix both the counter bore hole and the counter sunk hole (i.e. fix the cylindrical faces) as shown in the figure below and apply a pressure load of magnitude 600 psi normal to the face shown. For each of the four following cases, obtain the deflection contour and Von-Mises stress contour.

a) **Tetra 4** elements, element size – **0.2**
b) **Tetra 4** elements, element size – **0.05**
c) **Tetra 10** elements, element size – **0.2**
d) **Tetra 10** elements, element size – **0.05**

Please answer the following questions: What are the maximum deflection and the maximum stress for each of the four cases? Which of the four cases provides the most accurate results? How are the FEA results affected by the type and size of elements?



CHAPTER 9 – MANUFACTURING

As we discussed in Chapter 1 about the product realization process, the models and drawings created by designers will undergo manufacturing processes to get to the finished products, which is the essence of CAD/CAM integration. The most widely and commonly used technique is to generate control codes for CNC machines to mill the desired part. This technique reduces the amount of human programing in creating CNC codes, which facilitates the designers to create products with complex geometries. In this chapter, we will cover the *Manufacturing Module* of NX to generate CNC codes for *3-Axis Vertical Machining Centers*. This module allows us to program and do some post-processing on drilling, milling, turning and wire-cut EDM tool paths.

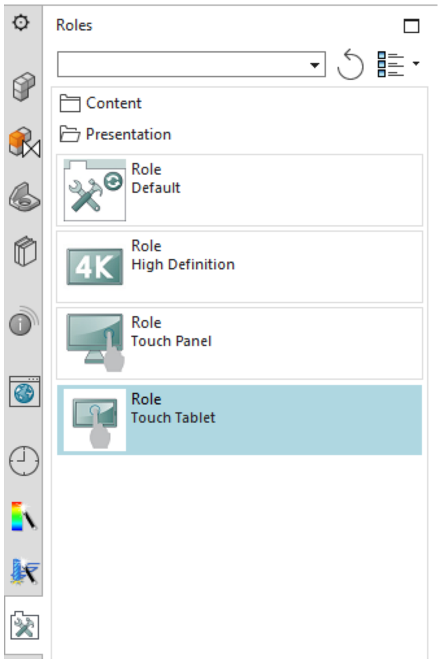
9.1 GETTING STARTED

A few preparatory steps need to be performed on every CAD model before moving it into the CAM environment. In this chapter, we are going to work with a model that was finished in the previous exercise problem. All the units are followed in **millimeters** in this model and manufacturing of the component.

Before getting started, it would be helpful if you can get into a *CAM Advanced Role*. To do this, go to the *Roles* menu on the *Resource Bar*. Click Content and a list of choices will pop up in which the *CAM Advanced* role can be seen as shown in the figure.

9.1.1 Creation of a Blank

After completing the modeling, you should decide upon the raw material shape and size that need to be loaded on the machine for the actual fabricating. This data has to be the input in NX. It can be achieved in two ways. The first method is creating or importing the model of the raw material as a separate solid in the same file and assigning that solid as the *Blank*. The second method is letting NX decide the extreme dimensions of the designed part and some offset values if needed. The later method allows a quick way of assigning the size details but it can only be used for prismatic shapes.



➢ Open the file **Die_cavity.prt** for the exercise problem in Chapter 4

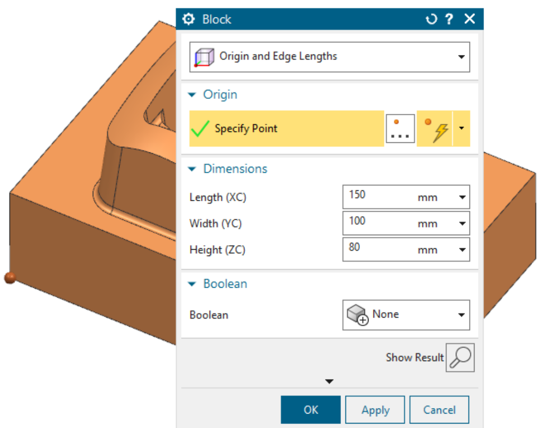
➢ To create a *Blank*, insert a block in with the following dimensions (Menu Insert **Design Feature**

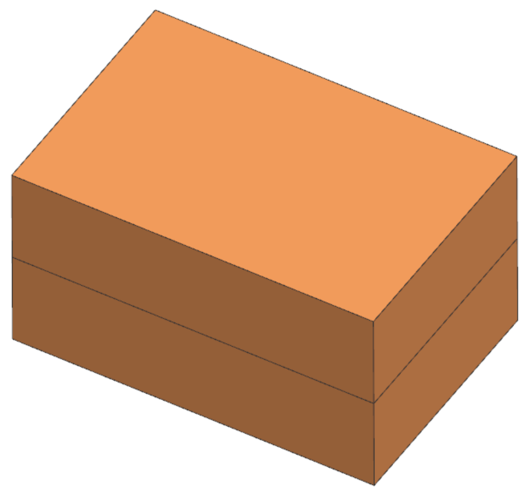
Length = **150** mm

Width = **100** mm

Height = **80** mm

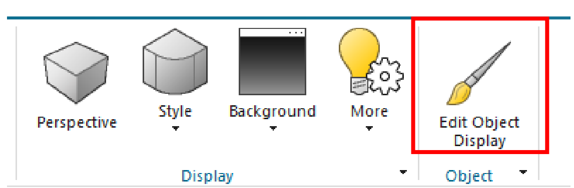
➢ For the **Origin** option, choose the lower most corner of the base block, so that the new block created can wrap up the whole previous model as shown below

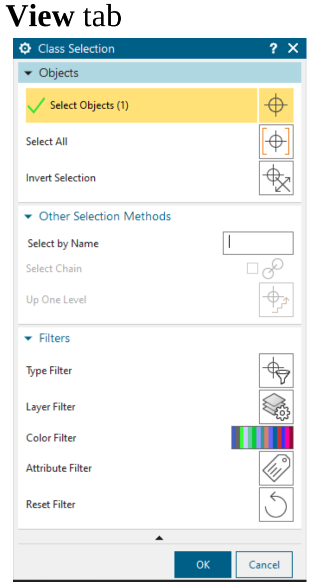




This block encloses the entire design part, so we need to change the display properties of the block to have a better visualization.

➢ Click on the **Edit Object Display** icon in the **Object** group of the **View** tab

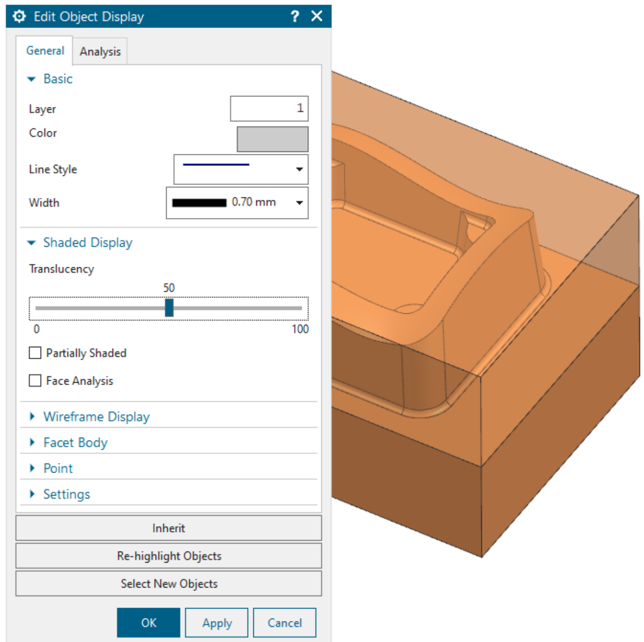




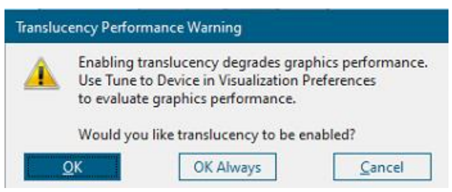
➢ Select the block you just created and click **OK**

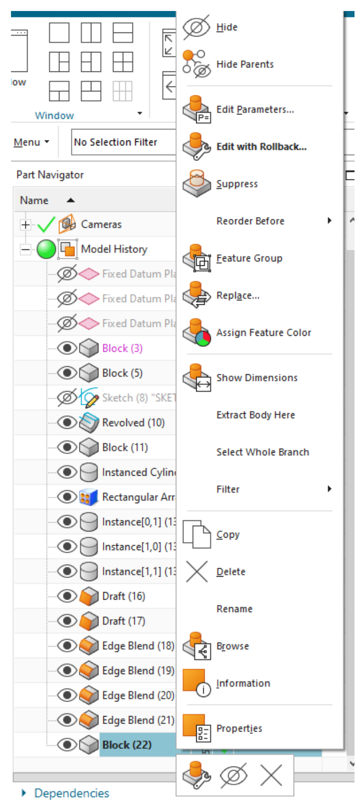
➢ When a window pops up, change the display **Color** and change the **Translucency** to **50**

➢ click **OK**

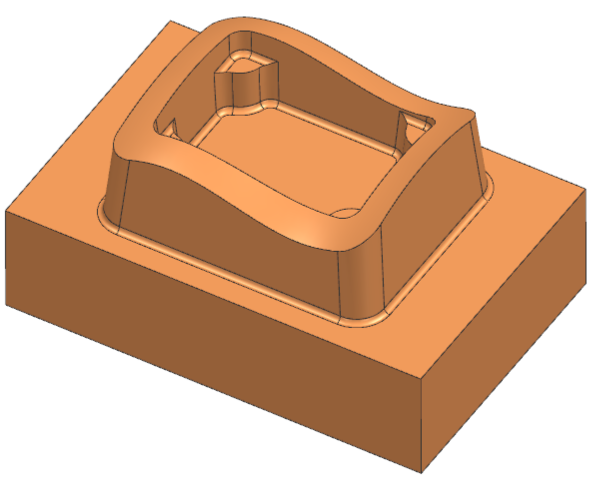


➢ Click OK when Translucency Performance Warning pops up.





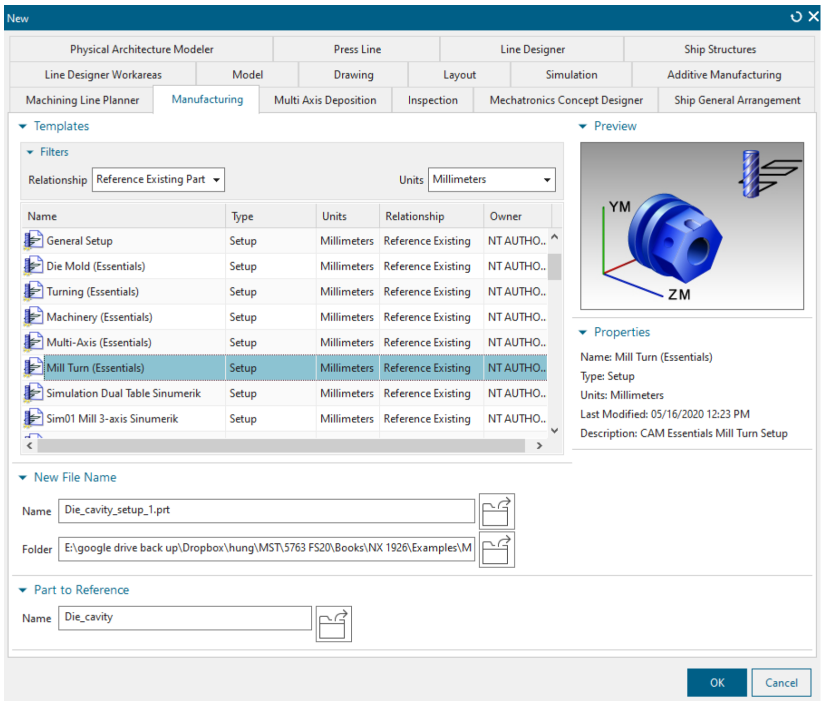
➢ **Hide** the block you just created by right clicking on the block in the **Part Navigator** and click **Hide** (you may need to use **Select from List…** to choose the blank). This will make the raw block disappear from the working environment.



9.1.2 Setting Machining Environment

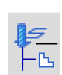
Now we are set to get into the **Manufacturing** module.

- Select **File** → **New** → **Manufacturing** → **Mill Turn**
- There are many different customized CAM templates available for different machining operations. Here, we are only interested in the milling operation.



9.1.3 Operation Navigator

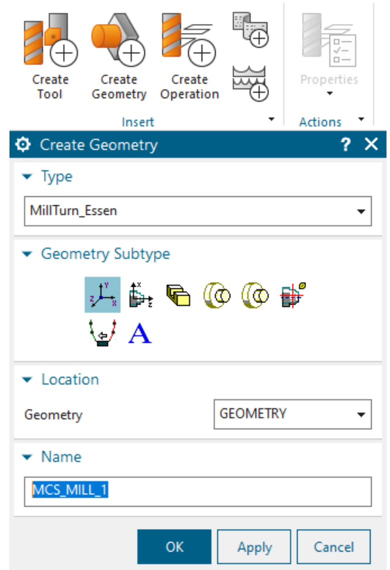
As soon as you get into the *Manufacturing* environment, you will notice many changes in the main screen such as new icons that are displayed.

- Click on the **Operation Navigator** tab  on the left on the **Resource Bar**

The *Operation Navigator* gives information about the programs created and corresponding information about the cutters, methods, and strategies. A list of programs can be viewed in different categorical lists. There are four ways of viewing the list of programs in the *Operation Navigator*, which are *Program Order view*, *Machine Tool view*, *Geometry view* and *Machining Method view*.

- Click on **Geometry View** 

9.1.4 Machine Coordinate System (MCS)



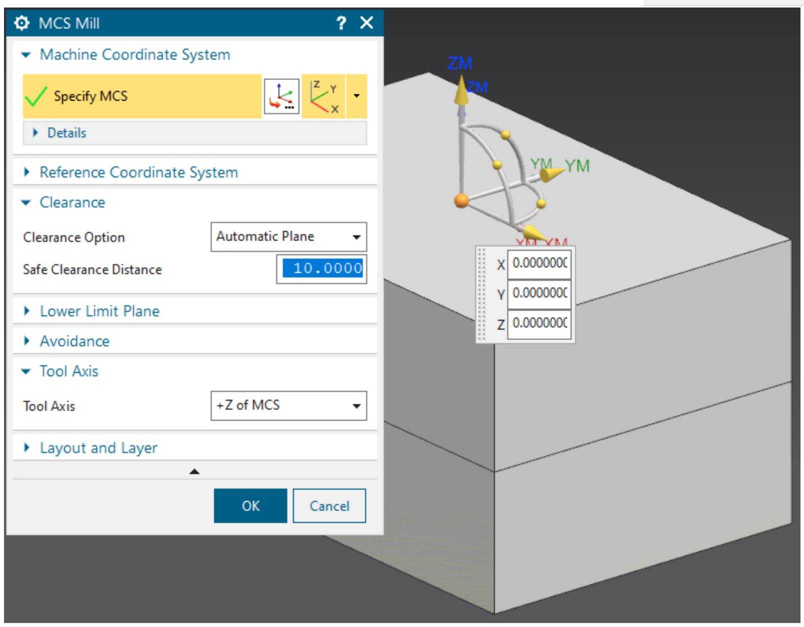
- ➢ Click on the **Create Geometry** icon in the **Insert** group to initiate setup for programming

You will see a *Create Geometry* popup dialog.

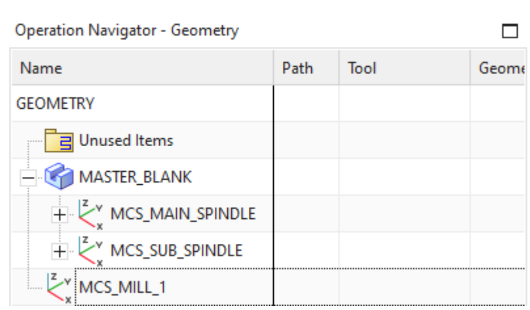
- ➢ Click **OK**

Another popup window will allow you to set the *MCS* wherever you want. By default, NX takes the original Absolute-CS as the MCS.

- ➢ Click **OK** to select the default choice as the **MCS**



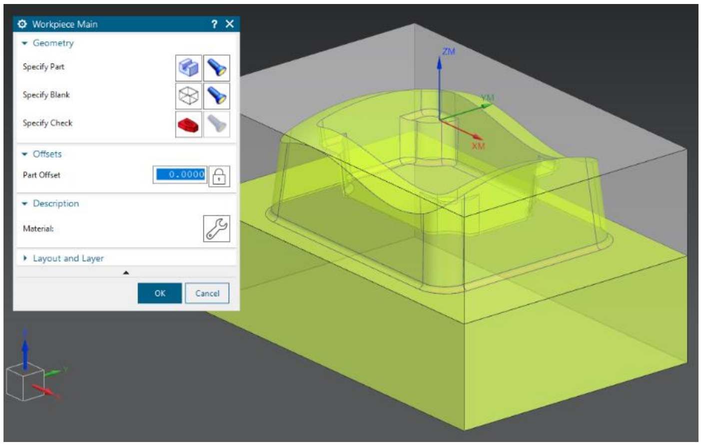
9.1.5 Geometry Definition



- ➢ Click on **Geometry View**
- ➢ Expand **MCS_MAIN_SPINDLE** by clicking on the '+' sign of MASTER_BLANK in the **Operation Navigator**
- ➢ Double click on **WORKPIECE_MAIN** in the **Operation Navigator**. If you do not see it, try clicking on related '+' signs

The popup window *Workpiece Main* appears. This is where you can assign the *Part* geometry, *Blank* geometry, and *Check* geometry.

- Click on the **Part** icon
- Select the design part (green part) and click **OK**



Now we will select the *Blank* Geometry.

- Click on the **Blank** icon

This will open the *Blank Geometry* Window. As mentioned earlier there are several ways to define the *Blank*. You can use a solid geometry as the *Blank* or can allow the software to assign a prismatic block with desired offsets in the X, Y, and Z directions. As we have already created a block, here we can use that as the *Blank* geometry.

- Click on the **Block** and press **OK**

Now we are finished assigning the *Part* and *Blank* geometries. Sometimes it may be required to assign *Check* geometry. This option is more useful for shapes that are more complex or 5-Axes milling operations where the tool cutters have a higher chance of dashing with the fixtures. In our case, it is not necessary to assign a *Check* geometry.

- Click **OK** to exit the **Workpiece Main** window

9.2 CREATING OPERATION

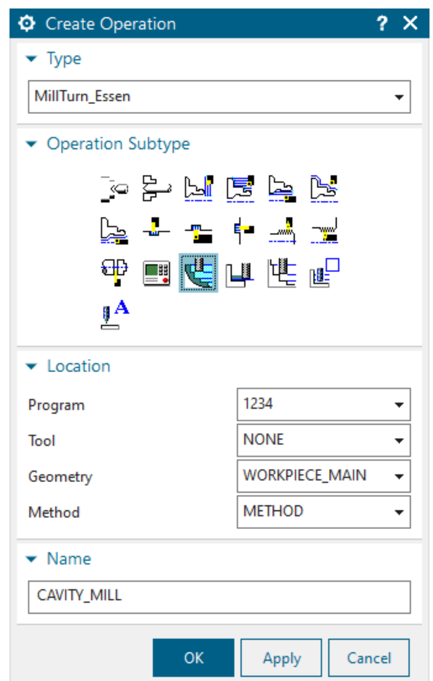
9.2.1 Creating a New Operation

The manufacturing setup is now ready for us to work further with *Programming Strategies*. There are many different manufacturing strategies involved in programming and it takes practice to know which one is optimal. Here, the basic guidelines are given for the most widely and frequently used strategies. This chapter will also cover some important parameters that are to be set for the programs to function properly.

- Click on the **Create Operation** icon in the toolbar 

The *Create Operation* window will pop up.

- Make sure the **Type** of Operation is **MillTurn_Essen**



There are a set of different subtypes under *Mill-Contour*, namely *Cavity Mill*, *Z-Level Follow Cavity*, *Follow Core*, *Fixed Contou*r, and so on. These different subtypes are used for different situations and profiles of the design part. As mentioned before, how you select a strategy for a certain situation depends on your knowledge and experience.

- Click on the **Cavity_Mill** icon at the top left as shown in the figure
- Choose the **Program** as **1234**
- Change the **Geometry** to **WORKPIECE_MAIN**
- Click **OK**

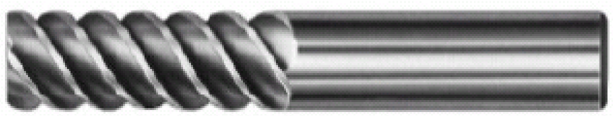
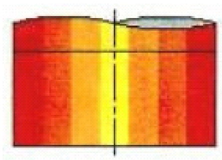
The program parameters window with *Cavity Mill* in the title bar will pop up. On this window, you can set all the parameters for this program. A brief introduction on every important parameter and terminology will be given as we go through the sequence.

9.2.2 Tool Creation and Selection

One of the most important decisions to make is to select a tool with the right shape and size to use. Before starting with the *Tool parameter* settings, we must first know about the types of *Tool* *cutters*. The *Milling Tool Cutters* are categorized into three forms of cutters. Hence, when selecting a cutter, it is important to take into consideration the size, shape, and profiles of the design parts. For example, if the corner radius of a pocket is 5 mm, the pocket should be finished by a cutter with a diameter less than or equal to 10 mm. Otherwise it will leave material at the corners. There are other special forms of cutters available in markets that are manufactured to suit different needs.

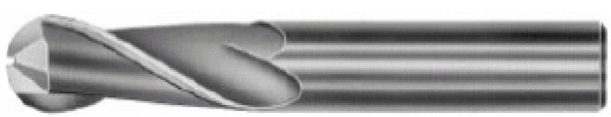
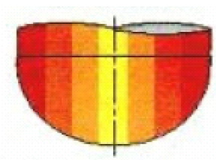
<u>Flat End Mill Cutters</u>

These cutters have a sharp tip at the end of the cutter as shown in the figure below. These cutters are used for finishing parts that have flat vertical walls with sharp edges at the intersection of the floors and walls.

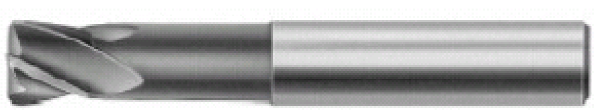
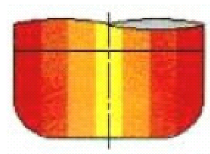
<u>Ball End Mill</u>

These cutters have the corner radii exactly equal to half the diameter of the shank. This forms the ball shaped profile at the end. These cutters are used for roughing and finishing operations of parts or surfaces with freeform features.

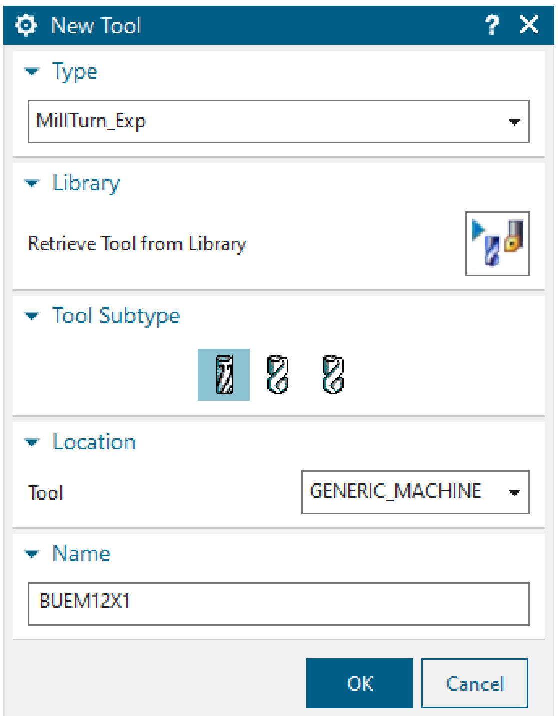
<u>Bull Nose Cutters</u>

These cutters have small corner radii and are widely used for roughing and/or semi-finishing the parts as well as for finishing of inclined and tapered walls.

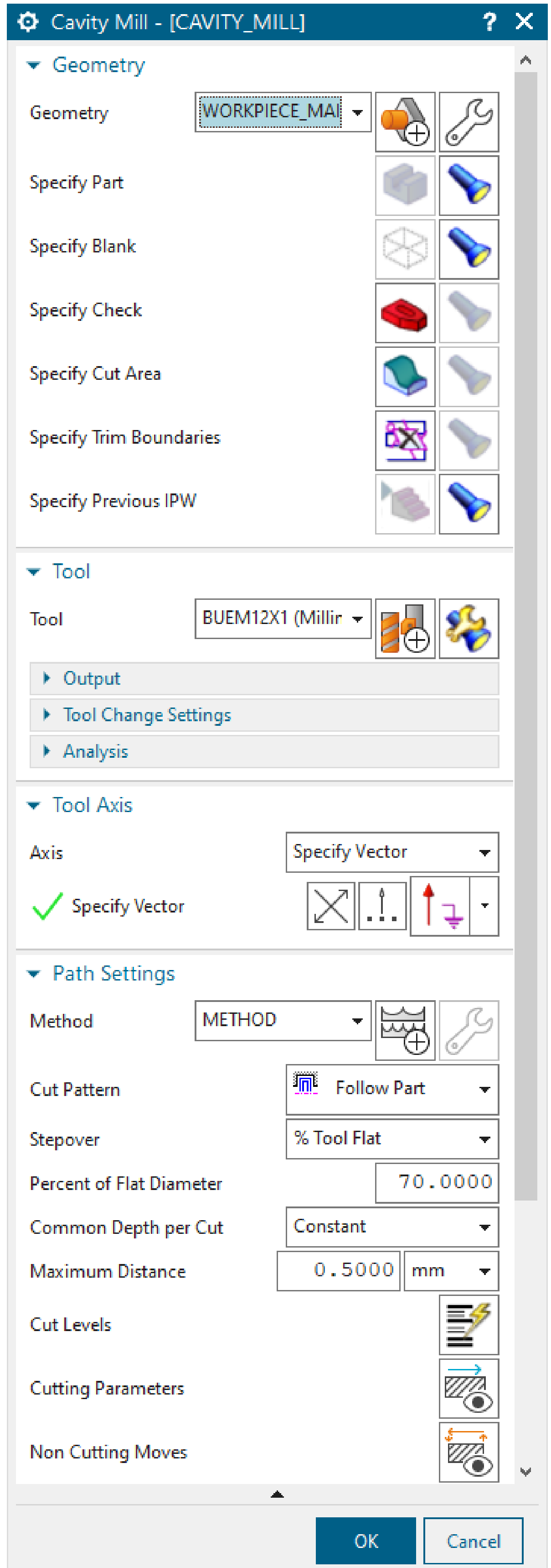
The cutter that we are going to use to rough out this huge volume is BUEM12X1 (Bullnose End Mill with 12 diameter and 1 corner radius).

- In the **Cavity Mill** popup menu click on the **Create New** button in the **Tool** dialog box
- Click **New**
- On the **New Tool** window, select the **Mill** icon

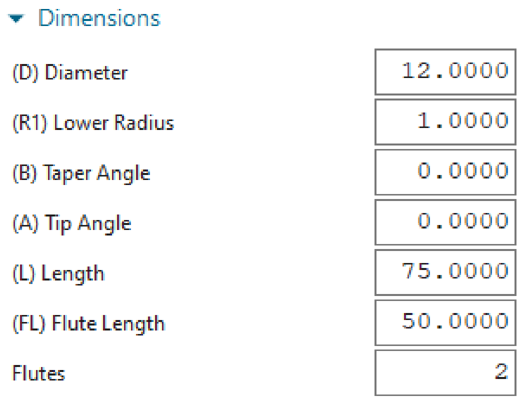


- Type in **BUEM12X1** as the **Name** and click **OK**

This will open another window to enter the cutter dimensions and parameters. You can also customize the list of tools that you would normally use and call the predefined cutters from the library.



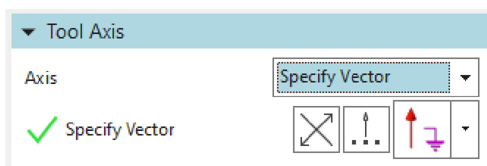
➢ Enter the values as shown in the figure below



➢ Click **OK**

9.2.3 Tool Path Settings

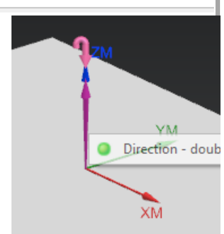
Make sure that the *Tool Axis* is perpendicular to the top surface on the block.



➢ Click on **Tool Axis** and choose **Specify Vector**

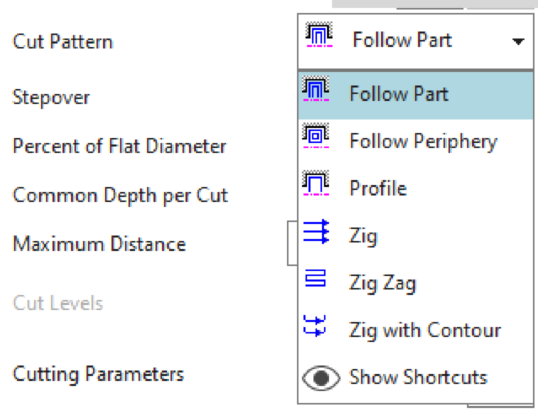
➢ Select the appropriate axis (**ZC**) as shown

➢ In the **Cavity Mill** menu click on the **Path Settings** option



There are different **Cut Patterns** in which the tool can move.

The following is a description of each.



 ***Follow Part***: This is the most optimal strategy where the tool path is manipulated depending on the part geometry. If there are cores and cavities in the part, the computer intelligently considers them to remove the materials in an optimal way. This is widely used for roughing operations.

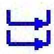
 ***Follow Periphery***: This takes the path depending upon the periphery profile. For example, if the outer periphery of our part is rectangular, the tool path will be generated such that it gradually

cuts the material from outside to inside with a Stepover value. This option is mostly used for projections and cores rather than cavities.

 ***Profile:*** This takes the cut only along the profile of the part geometry. It is used for semi-finishing or finishing operations.

 ***Zig:*** This takes a linear path in only one direction of flow.

 ***Zig Zag:*** This tool takes a zigzag path at every level of depth. It saves time by reducing amount of air cutting time (idle running). The climb and conventional cuts alternate.

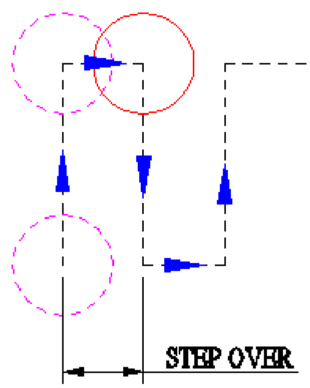
 ***Zig with Contour:*** This takes the path in one direction either climb or conventional. The unique thing is that it moves along the contour shape nonlinearly.

- For this exercise, select the **Follow Part** icon from the **Cut Pattern** drop-down menu since we have both projections and cavities in the part

9.2.4 Step Over and Scallop Height

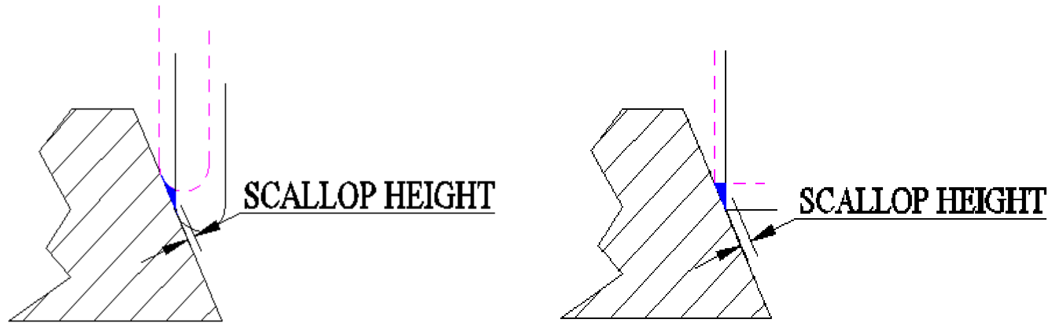
Step Over

This is the distance between the consecutive passes of milling. It can be given as a fixed value or the value in terms of cutter diameter. The *Stepover* should not be greater than the effective diameter of the cutter, otherwise it will leave extra material at every level of cut and result in an incomplete milling operation. The numeric value or values required to define the *Stepover* will vary depending on the *Stepover* option selected. These options include *Constant, Scallop, Tool Diameter*, etc. For example, *Constant* requires you to enter a distance value in the subsequent line.

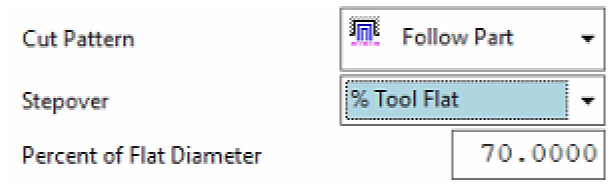


Scallop Height

Scallop Height controls the distance between parallel passes according to the maximum height of material (scallop) you specify to be left between passes. This is affected by the cutter definition and the curvature of the surface. *Scallop* allows the system to determine the *Stepover* distance based on the scallop height you enter.



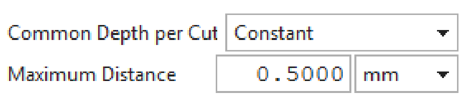
➢ For the **Stepover**, select **%Tool Flat** and change the ***Percent of Flat Diameter*** to **70**



9.2.5 Depth Per Cut

This is the value to be given between levels to slice the geometry into layers and the tool path cuts as per the geometry at every layer. The cut depth value can vary for each level. Levels are horizontal planes parallel to the XY plane. If we do not give cut levels, the software will unnecessarily try to calculate slices for the entire part and machine areas that are not in our interest.

➢ Change the **Common Depth per Cut** value to **0.5**

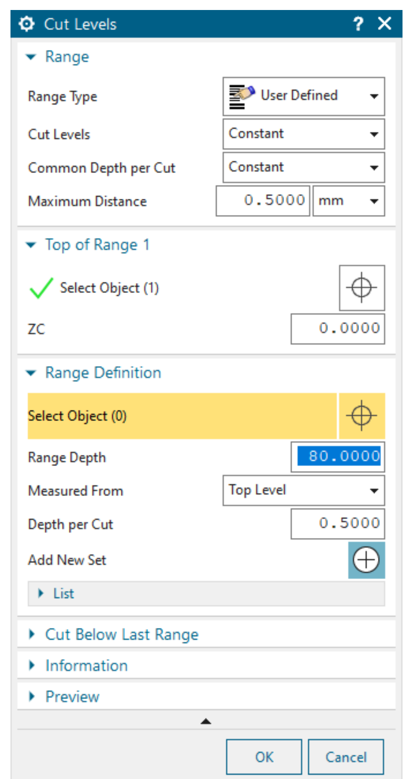


Now we will add the level ranges. This will split the part into different levels along the Z-direction to be machined.

➢ Click on **Cut Levels**

This will pop up a dialog box of *Cut Levels*. You need to set the level of the cut. You can either point to the object till which the cut level is or provide it as a *Range Depth* value. We are not going to mill up to the bottommost face of the part, but up to the floor at 40 mm from top. Therefore, we must delete the last level.

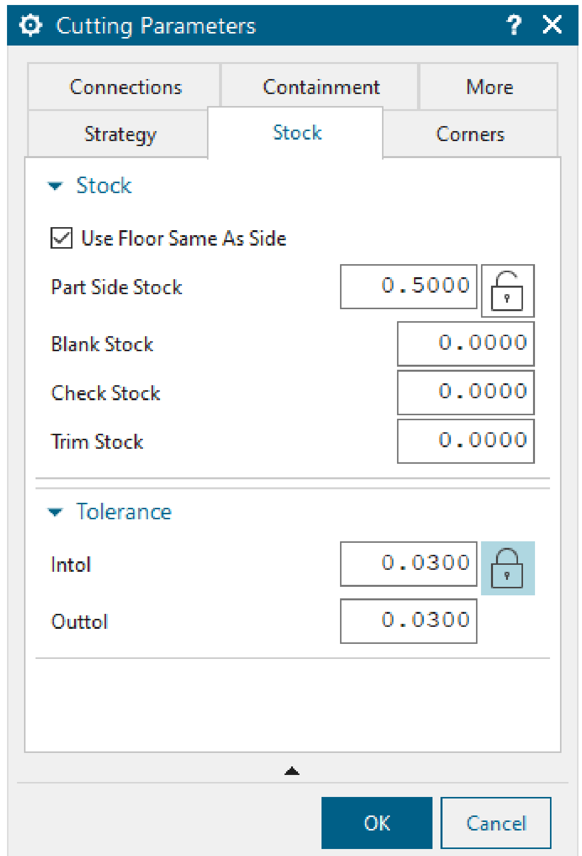
➢ Change the **Range Type** to **User Defined**



- Change the **Range Depth** to **80**
- Select **OK**

9.2.6 Cutting Parameters

- On the **Path Settings** menu, click **Cutting Parameters**
- Under the **Strategy** tab button, change the **Cut Order** from **Level First** to **Depth First**



Changing the cut order to *Depth First* orders the software to generate the tool path such that it will mill one island completely up to the bottom-most depth before jumping to another island. The *Depth First* strategy reduces the non-cutting time of the program due to unnecessary retracts and engages at each depth of cut.

- Click on the **Stock** tab
- Change the value of the **Part Side Stock** to **0.5**

This value is the allowance given to every side of the part. If you want to give different values to the floors (or the flat horizontal faces) uncheck the box next to *Use Floor Same As Side* and enter a different value for *Part Floor Stock*.

- Click **OK**

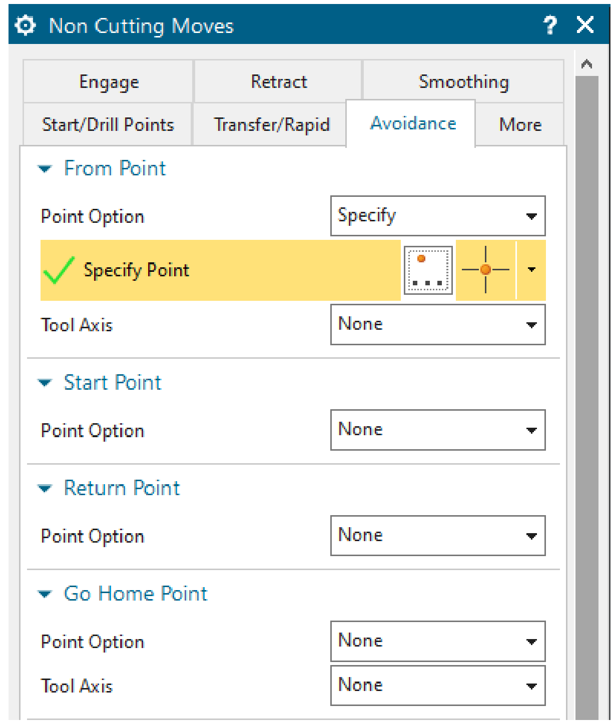
9.2.7 Avoidance

- Click the **Non Cutting Moves**
- Click the **Avoidance** tab

This window consists of several avoidance points of which we are concerned with the following points:

From Point

This is the point at which the tool change command will be carried out. The value is normally 50 or 100 mm above the Z=0 level to enhance the safety of the job when the cutter is changed by the Automatic Tool Changer (ATC).



- Click **From Point**
- Choose **Specify** in the **Point Option** field
- In the **Point Dialog**, enter the coordinates of XC, YC and ZC as **(0, 0, 50)**
- Click **OK**

Start Point

This is the point at which the program starts and ends. This value is also 50 or 100 mm above the Z=0 level to enhance safety. It is also the point at which the machine operator checks the height of the tool mounted on the spindle with respect to the Z=0 level from the job. This cross checks the tool offset entered in the machine.

- Click on **Start Point**
- Choose **Specify**
- Enter the coordinates **(0, 0, 50)** in the **Point Dialog**
- Click **OK** to exit the **Point Constructor**

Clearance Plane is the plane on which the tool cutter will retract before moving to the next region or island. This is also known as *Retract Plane*. Sometimes the *Clearance Plane* can be the previous cutting plane. However, when the tool has to move from one region to another, it is necessary to move to the *Clearance Plane* before doing so. The value of the *Clearance Plane* should be at least 2 mm above the topmost point of the workpiece or fixture or whichever is fixed to the machine bed.

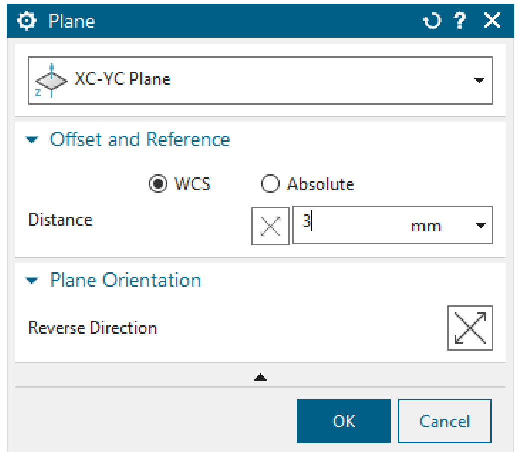
- Click on the **Transfer/Rapid** tab

➢ Choose **Plane** in the **Clearance Option**

➢ Choose the **XC-YC Plane** in **Plane Dialog**

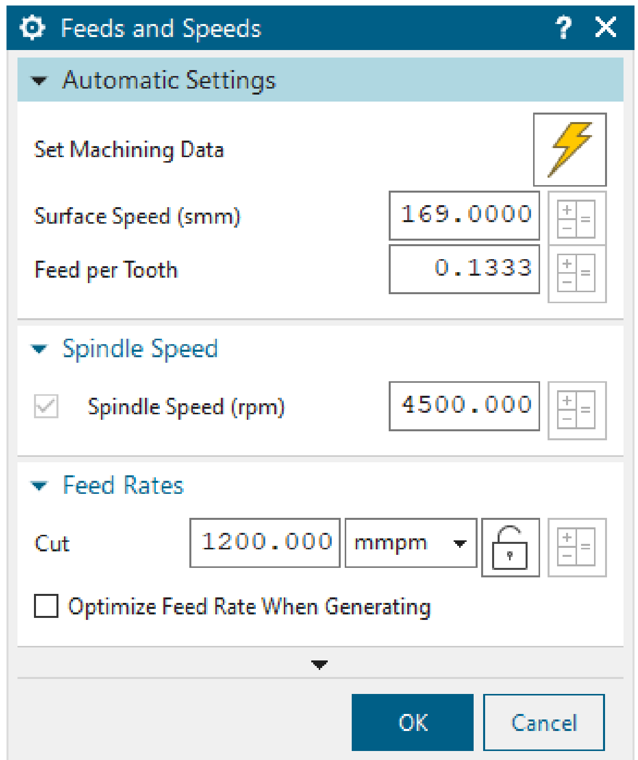
➢ Under the **Offset and Reference** tab enter the value of **3** as the **Distance**

➢ Click **OK** twice to go back to the **Cavity Mill** parameters window



9.2.8 Speeds and Feeds

➢ Choose **Feeds and Speeds** to enter the feed and speed parameters



Speed

Speed normally specifies the revolutions per minute (rpm) of the spindle (spindle speed). However, technically the speed refers to the cutting speed of the tool (surface speed). It is the linear velocity of the cutting tip of the cutter. The relative parameters affecting this linear speed are rpm of the spindle and the diameter of the cutter (effective diameter).

➢ Enter the **Spindle Speed** value as **4500 rpm**

For the Surface Speed and the Feed per Tooth, you should enter the recommended values given by the manufacturers of the cutter (for this example, click on the calculator button near spindle speed). By entering these values, the software will automatically calculate the cutting feed rate and spindle speed. You can also enter your own values for feed rates and spindle speeds.

Feeds

There are many feeds involved in a single program. The most important is the *Cutting* feed. This is the feed at which, the tool will be in engagement with the raw work-piece and actually cutting the material off the work-piece. It is the relative linear velocity, at which the cutter moves with respect to the job.

The other feeds are optional. Some machine control systems use their default retracts and traverse feed. In those cases, even if you do not enter the values of other feeds, there would not be any problems. Some control systems may look for these feed rates from the program. It can be slightly less than the machine's maximum feed rate.

- Enter the **Cut** value as **1200 mmpm**
- Click **OK**

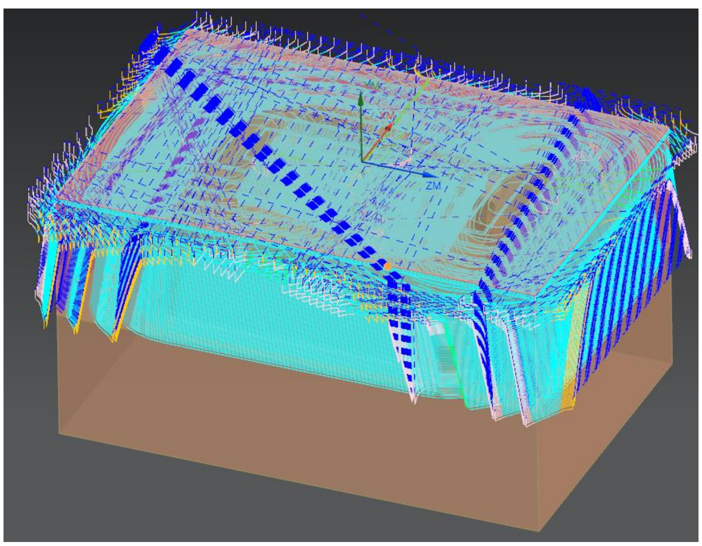
9.3 PROGRAM GENERATION AND VERIFICATION

9.3.1 Generating Program

Now we are done entering all the parameters required for the roughing program. It is time to generate the program.

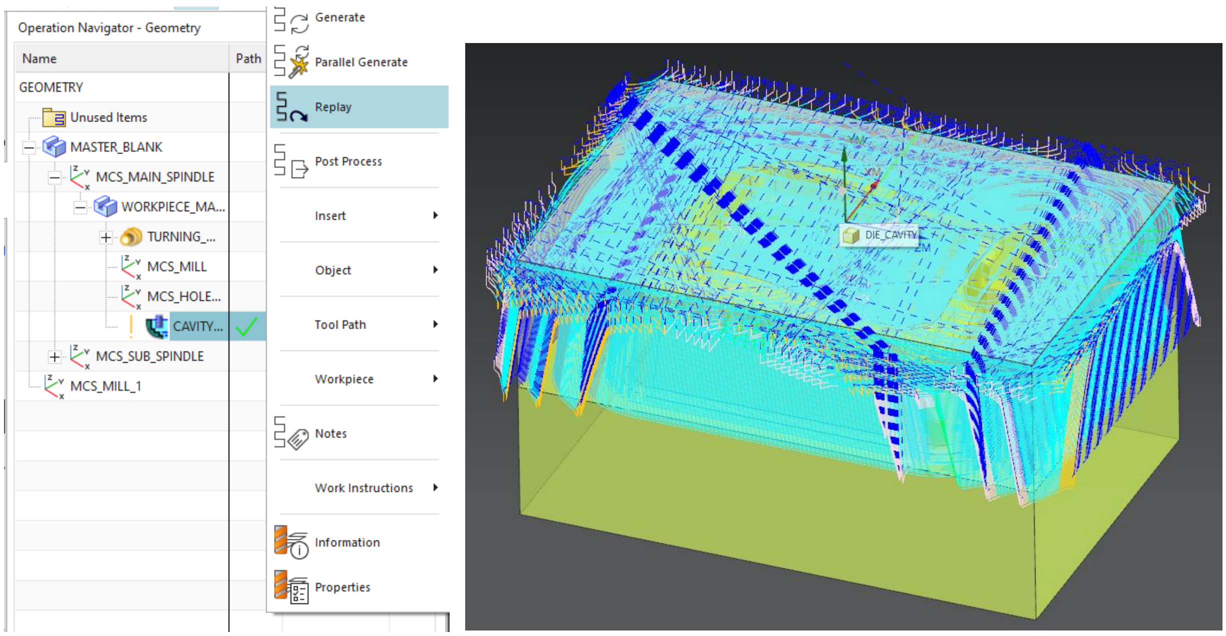
- Click on the **Generate** icon at the bottom of the window

You can now observe the software slicing the model into depths of cuts and creating tool-path at every level. You can find on the model cyan, blue, red and yellow lines as shown in the figure.



9.3.2 Tool Path Display

Whenever you want to view the entire tool-path of the program, right-click on the program in *Operation Navigator* and click *Replay*. It will give the display as shown in the Figure below.



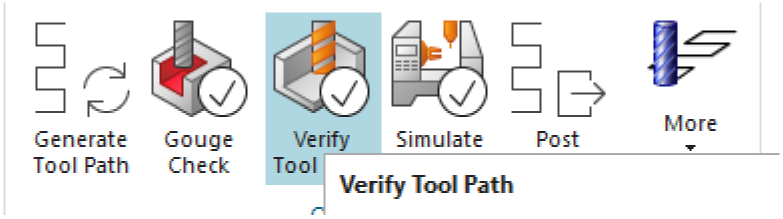
You can now observe that next to the program in the *Operation Navigator* is a yellow exclamation point instead of a red mark. This means that program has been generated successfully but has not been post-processed. If any change is made in the model, the program will again have a red mark next to it. This implies that the program has to be generated again. However, there is no need to change any parameters in the program.

9.3.3 Tool Path Simulation

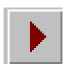
It is very important to check the programs you have created. This prevents any improper and dangerous motions from being made in the cutting path. It is possible that wrong parameters and settings will be given that cause costly damages to the work piece. To avoid such mistakes, NX provides a *Tool-path* verification and a *Gouge* check.

The *Tool-Path verification* can be used to view the cutter motion in the entire program. You can observe how the tool is engaged and how it retracts after cutting. It also shows the actual material being removed through graphical simulation. You can also view the specific zone of interest by moving the line of the program.

➢ Right-click on the program in the **Operation Navigator** and choose **Tool Path → Verify** or click on the **Verify Tool Path** button in the toolbar



This will allow you to set the parameters for visualization of the *Tool-Path*.

➢ On the **Tool Path Visualization** window, click on the **Play** icon  to view the motion

You can also view the visualization in different modes by changing the options in the drop-down menu next to *Display*.

➢ Click on the **3D Dynamic** tab on the same window

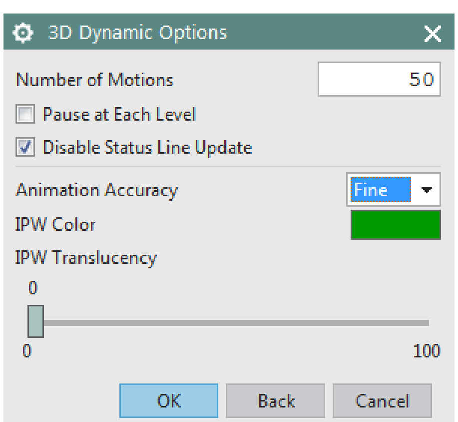
➢ Click on the **Display Options** button on the same window

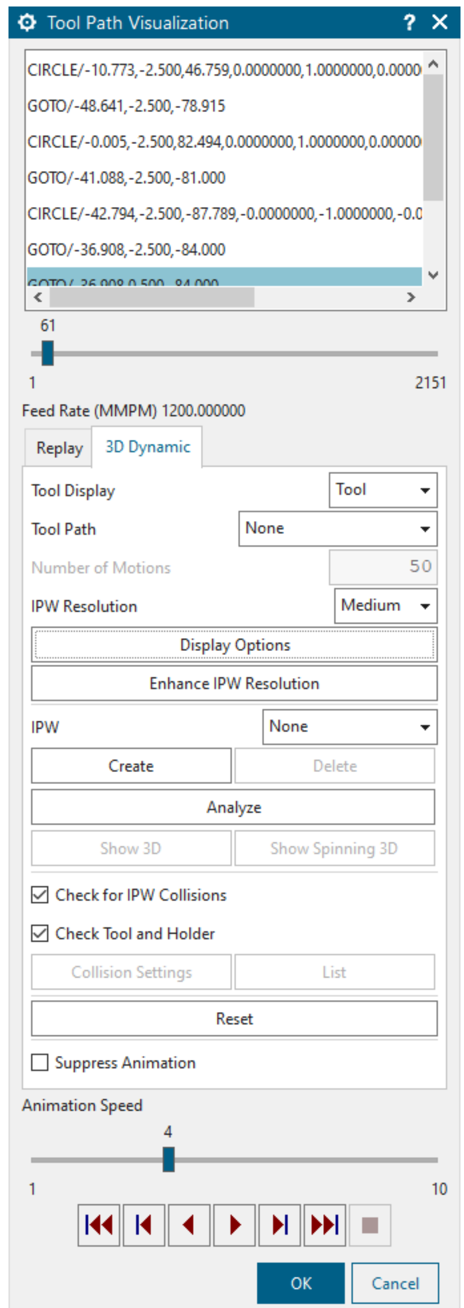
➢ Change the **Number of Motions** to **50**

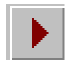
➢ Change the **Animation Accuracy** to **Fine**

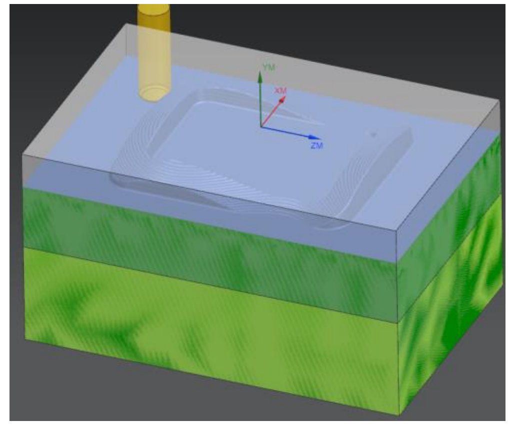
➢ Change the **IPW Color** to **Green**

➢ Click **OK**





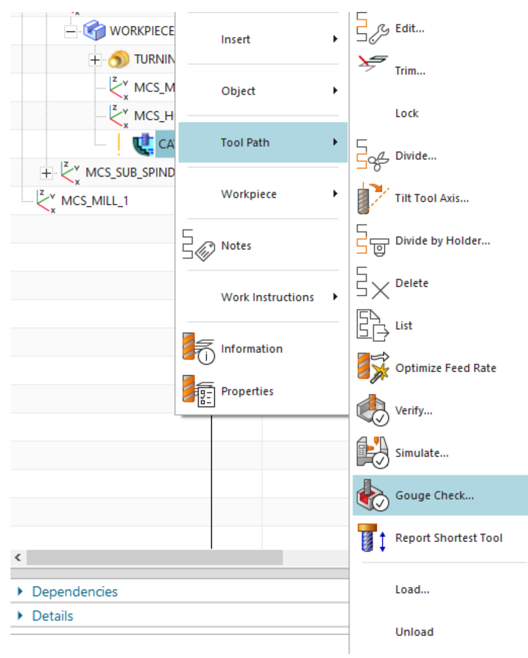
- Click on the **Play button**  again



The simulation will look as shown in the figure on the right. With this option, you will be able to view the actual cutting simulation and material removal through computer graphics. It is *3D Dynamic*, where you can rotate, pan and zoom the simulation when it is playing.

9.3.4 Gouge Check

Gouge Check is used to verify whether the tool is removing any excess material from the work-piece with respect to *Part Geometry*. Considering a *Design Tolerance*, any manufacturing process may produce defective parts by two ways. One is removing excess material, which is also called *Less Material Condition*. The other one is leaving materials that are supposed to be removed which is *More Material Condition*. In most cases, the former is more dangerous since it is impossible to rework the design part. The latter is safer since the leftover material can be removed by reworking the part. The gouge check option checks for the former case where the excess removal of material will be identified.



- Right Click the program in the **Operation Navigator**
- Choose **Tool Path → Gouge Check**
- Click **OK**

After the gouge check is completed, a **Tool Path Report** window will pop up. If in case there are any gouges found, it is necessary to correct the program. Otherwise,

- Click **OK** or directly close the popup window

9.4 OPERATION METHODS

9.4.1 Roughing

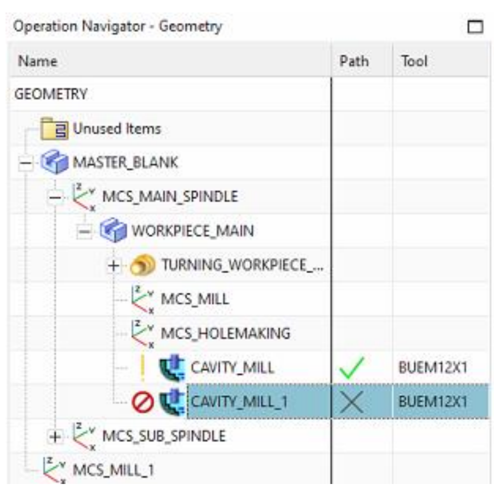
In case of milling operation, the first operation should be rough milling before finishing the job. The main purpose of roughing is to remove bulk material at a faster rate, without affecting the accuracy and finish of the job. Stock allowances are given to provide enough material for the finishing operation to get an accurate and good finish job. What we did in the previous part of this chapter is generating a roughing program. Now we have to moderately remove all the uneven material left over from the previous program.

9.4.2 Semi-Finishing

Semi-Finishing programs are intended to remove the unevenness due to the roughing operation and keep even part stock allowance for the *Finishing* operations. Once we are done with the first roughing program, semi-finishing is always easier and simpler to perform.

Now we will copy and paste the first program in the *Operation Navigator*. In the new program, you only have to change a few parameters and cutting tool dimensions and just regenerate the program.

- Right click **CAVITY_MILL** program in the **Operation Navigator** and click **Copy**
- Right-click **CAVITY_MILL** again and choose **Paste**
- Right-click the second **CAVITY_MILL_COPY** you just made and click **Rename**
- Rename the second program as **CAVITY_MILL_1**

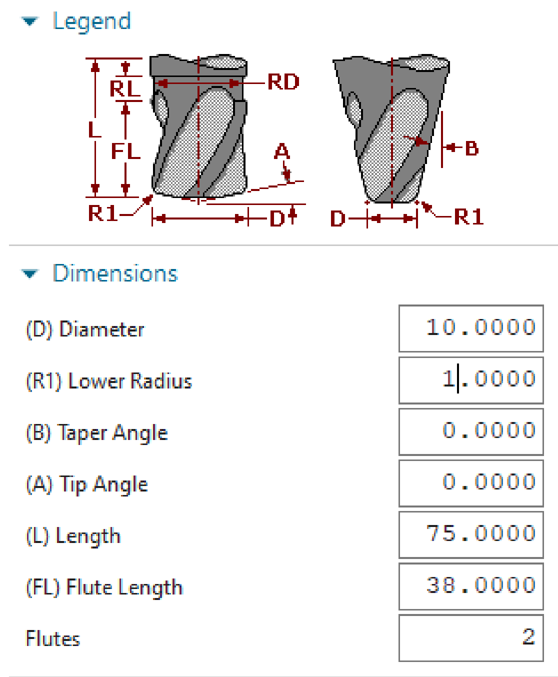


You can see that next to the newly created **CAVITY_MILL_1** is a red mark, which indicates that the program is not generated.

Let us now set the parameters that need to be changed for the second program. Before we start, we should analyze the part geometry to figure out the minimum corner radius for the cutter diameter. In our model, it is 5 mm and at the floor edges, it is 1 mm. Therefore, the cutter diameter can be anything less than 10 mm. For optimal output and rigidity, we will choose a *Bull Nose Cutter* with a diameter of 10 and a lower radius of 1.

➢ Double click **CAVITY_MILL_1** on **Operation Navigator** to open the parameters window

Just as we did in the previous program, we will create a new cutter. In the *Tool* tab, you will see the cutter you first chose. It will show *BUEM12X1* as the current tool.

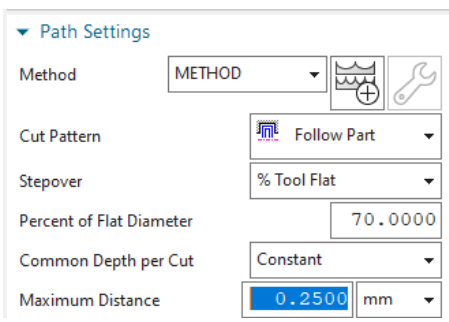


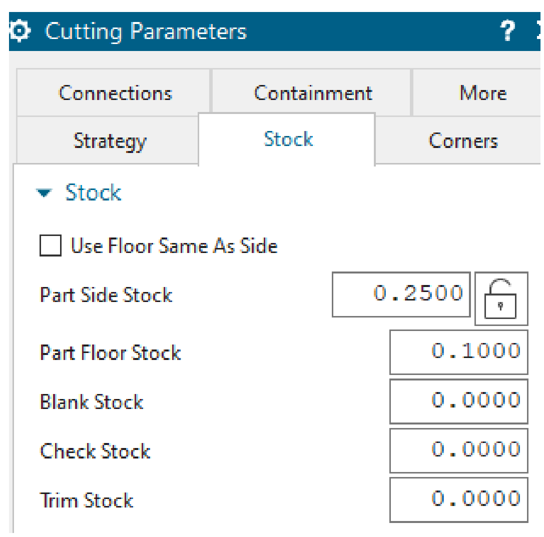
➢ Create a new **Mill** and name it **BUEM10X1**

➢ It should have a **Diameter** of **10**, a **Lower Radius** of **1** and a **Flute Length** of **38**

➢ Click **OK**

➢ Click the **Common Depth per Cut** as **0.25** in the **Path Settings**





➢ Then click on **Cutting Parameters** button

➢ Click on the **Stock** tab

➢ Uncheck the box next to **Use Floor Same As Side**

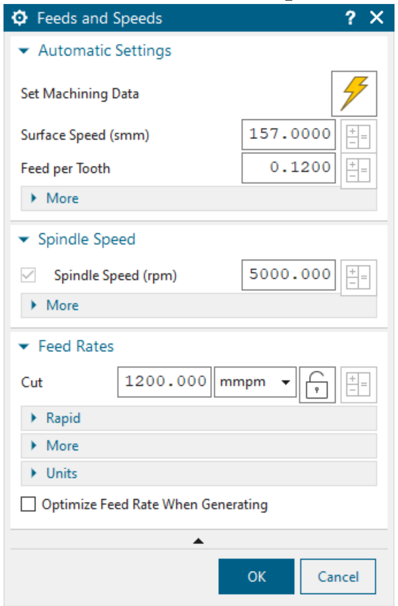
➢ Enter **0.25** for **Part Side Stock**

➢ Enter **0.1** for **Part Floor Stock**

➢ Click on the **Containment** tab button

➢ In the drop-down menu next to **In Process Workpiece**, choose **Use 3D**

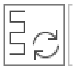
In Process Workpiece is a very useful option in NX. The software considers the previous program and generates the current program such that there is no unnecessary cutting motion in the *No-material* zone. This strategy reduces the cutting time and air cutting motion drastically. The algorithm will force the cutter to only remove that material, which is left from the previous program and maintain the current part stock allowance.



- ➢ Choose **OK** to return to the **Parameters** window
- ➢ Click **Feeds and Speeds**
- ➢ Enter the **Spindle Speed** as **5000**
- ➢ Then click **OK**

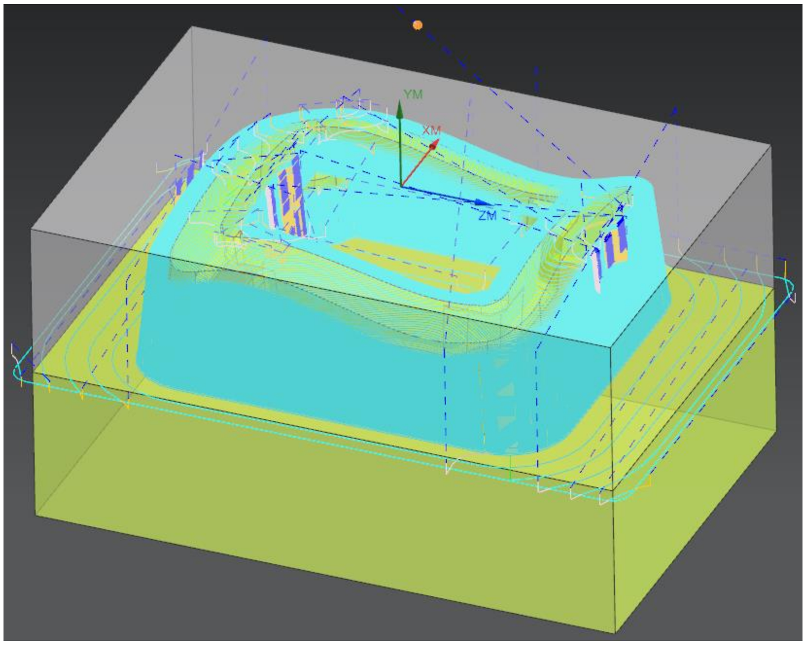
The parameters and settings are finished for the semi-finishing program.

- ➢ Regenerate the program by clicking on the **Generate** icon



- ➢ After the software finishes generating click **OK**

Then replay the *Tool Path Visualization*. The overall *Tool Path* generated in the second program will look like the following figure. You can replay it or check for the gouging in a similar way.



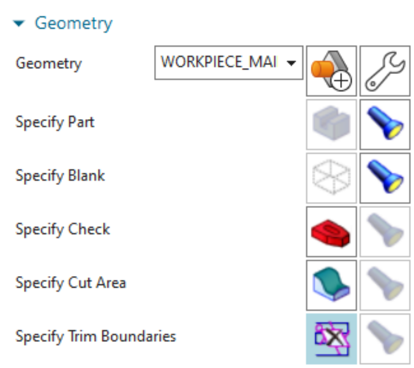
9.4.3 Finishing Profile

So far, we are done with the roughing and semi-finishing programs for the part. There is a small amount of material left in the *Workpiece* to be removed in the finishing programs to obtain the desired accurate part geometry. The finishing programs should be generated such that every surface in the part should be properly machined. Therefore, it is better to create more than one program to uniquely machine sets of surfaces with relevant cutting parameters and strategies rather than make one program for all the surfaces. The following illustrates how to group the profiles and surfaces and create the finishing programs.

9.4.3.1 Outer Profile

This program is intended to finish the outer inclined walls onto the bottom of the floor. Because the program should not touch the contour surface on the top, we will give *Check* and *Trim* boundaries in the program.

- Repeat the same procedure as before to copy and paste **CAVITY_MILL_1** on **Operation Navigator**
- Rename the program **CAVITY_MILL_2**
- Double click **CAVITY_MILL_2** to make parameter changes
- In the popup parameters window, change the **Cut Pattern** to **Profile** and the **Stepover percentage** to **40**
- Click on the **Specify Trim Boundaries** tab



The *Trim Boundaries* window will pop up. Make sure to carry out the following procedure in the right sequence. Keep the default setting of *Trim Side* to *Inside*. This tells the software that the cutter should not cut material anywhere inside the boundary. Trim allows you to specify boundaries that will further constrain the cut regions at each cut level.

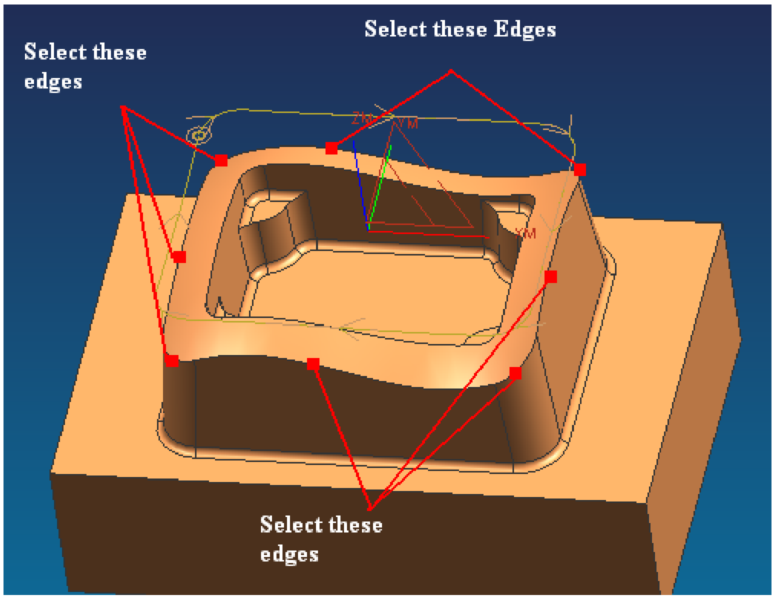
- Change the **Selection Method** to **Curves**
- Change the **Plane** from **Automatic** to **Specify** and click on the **Plane Dialog**

A new window will pop up. The window will ask for the mode of selection of the plane on which the curves should be projected. This should normally be over the topmost point of the part geometry. Precisely, it should be over the *MCS*.

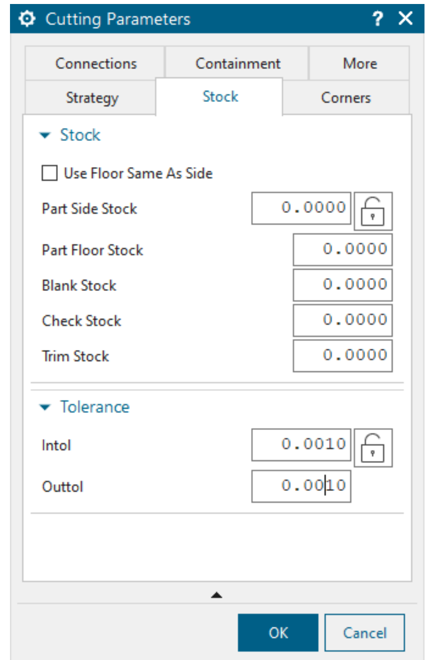
- Choose the **XC-YC Plane** from the drop-down menu under **Type**
- Enter a value of **3** next to **Distance**
- Click **OK**

Now we will start selecting edges from the part. These selected edges will be projected on the *Z = 3* plane as curves and used as the boundary.

- Select all the top outer edges on the wall along the contour surface as shown in the figure. Make sure to select all **8 edges** and in a **continuous order**
- Choose **OK**

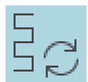


- Enter the **Common Depth per Cut** as **0.2**
- Click **Cutting Parameters**
- In the popup dialog box, click on **Stock** tab
- Enter the **Part Side Stock** and **Part Floor Stock** values to be **0.00**

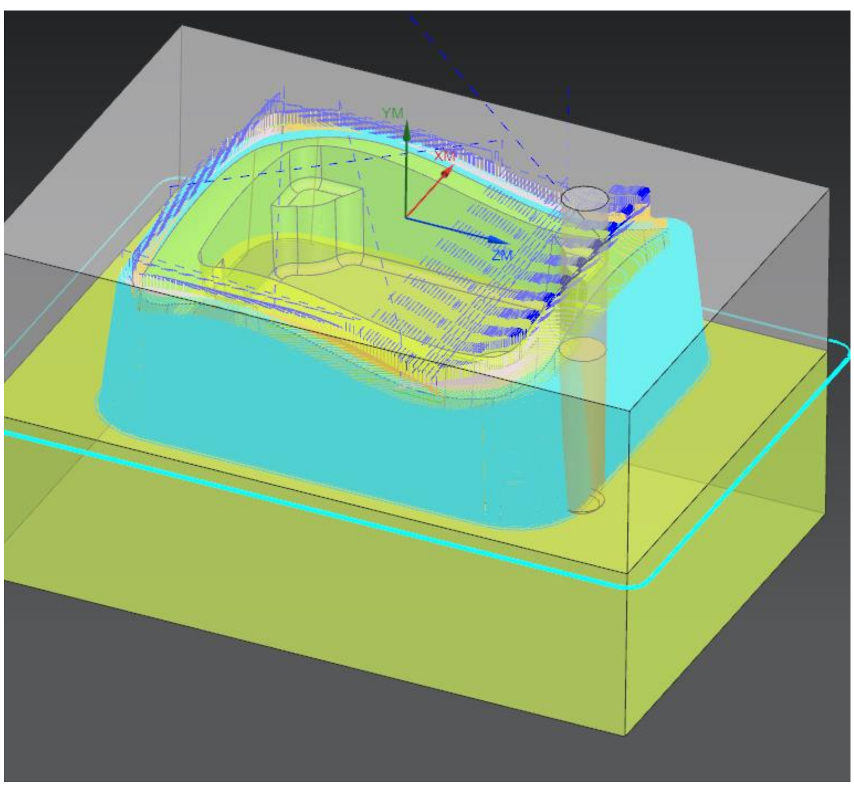


Intol allows you to specify the maximum distance that a cutter can deviate from the intended path into the workpiece.

Outtol allows you to specify the maximum distance that a cutter can deviate from the intended path away from the workpiece.

- Enter the **Intol** and **Outtol** values to be **0.001** as shown in the figure
- Click **OK**
- Click on the **Generate** icon  to generate the program in the **Main Parameters** window
- Click **OK** on the parameters window when the program generation is completed

The finishing program for the outer profile is now ready. You can observe while replaying the tool path that the cutter never crosses the boundary that has been given for trim and check. The cutter retracts to the Z=3 plane for relocation.

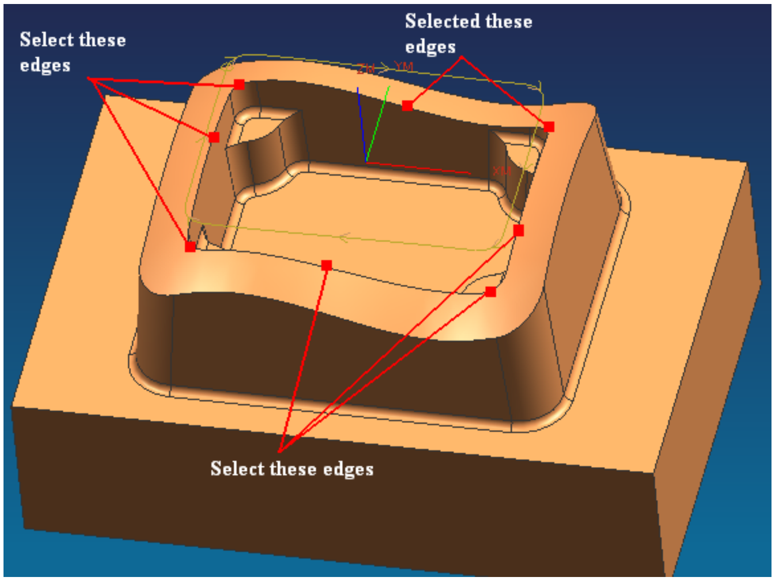


9.4.3.2 Inner profile

- Repeat the same procedure as before to copy and paste **CAVITY_MILL_2** on **Operation Navigator** and rename it as **CAVITY_MILL_3**
- Double click **CAVITY_MILL_3** to edit the parameters or right click on it and choose **Edit**
- Select the **Specify Trim Boundaries** tab and choose **Trim Side** to be **Outside** in the popup dialog box

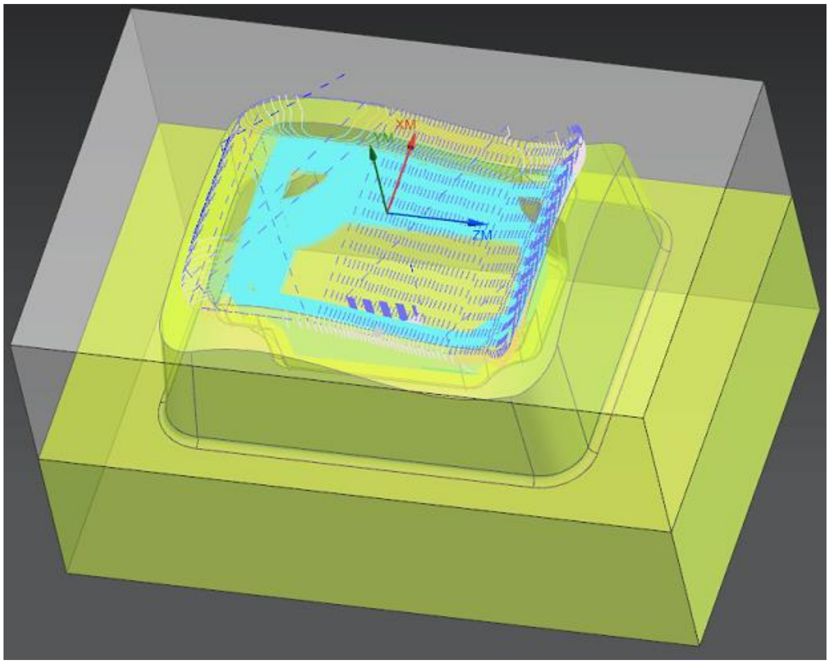
This will prevent the cutter from passing outside the boundary.

- Select all the top inner edges along the contour surface as shown in the figure. Again, make sure all 8 edges are selected in a continuous order **(**using Shift+click to de-select a curve**).**



- Change the plane manually to be the **XC-YC** plane and enter the offset distance as **3**
- Click **OK**
- Choose **OK** to return to the parameters window
- Generate the program
- Click **OK** when the generation is finished

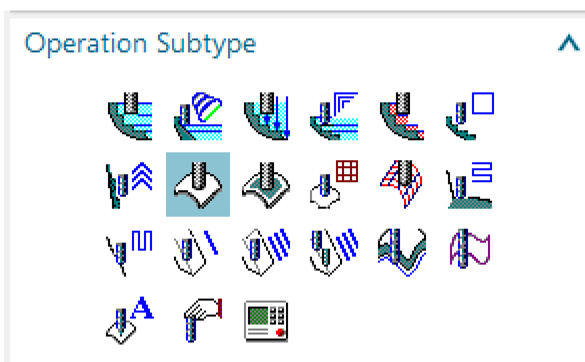
The finishing program for the inner profile is now ready. By replaying the tool path, you can observe that the cutter never crosses the boundary that has been given for trim and check.



9.4.4 Finishing Contour Surface

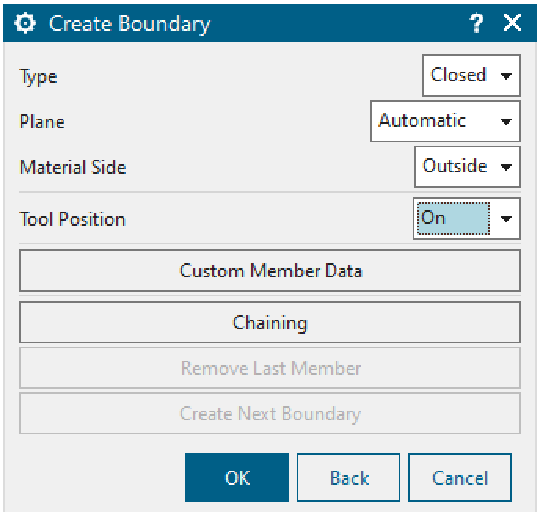
Now we will use a different type of strategy to finish the top freeform surface.

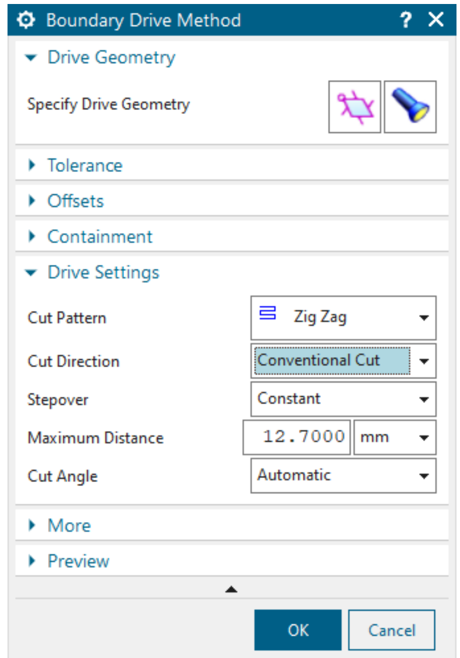
- Click on the **Create Operation** icon  in the Toolbar
- Click on the **Fixed Contour** icon as shown in the figure



- Choose **1234** for Program
- Choose **WORKPIECE_MAIN** for **Geometry**
- Keep the default name of program
- Click **OK**
- On the **Parameters** window, under **Drive Method**, make sure that **Boundary** is selected
- Click  icon, which will pop up the **Boundary Drive Method** window, click on the  icon as shown in the figure to open the **Boundary Geometry** menu
- Change the **Mode** to **Curves/Edges**

- Select the **Material Side** to be **Outside**
- Select the **Tool Position** to be **On**

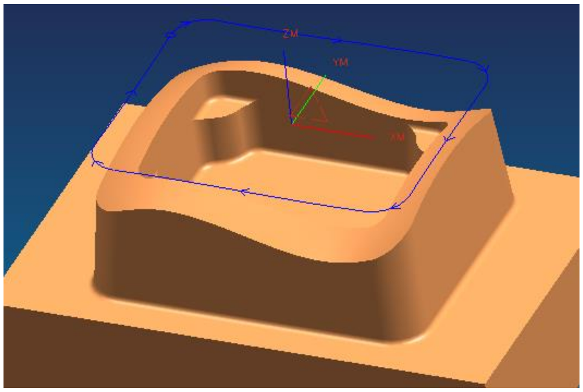




The *Tool Position* determines how the tool will position itself when it approaches the *Boundary Member*. *Boundary Members* may be assigned one of three tool positions: *On*, *Tanto*, or *Contact*.

- In **On** position, the center point of the tool aligns with the boundary along the tool axis or projection vector.
- In **Tanto** position, the side of the tool aligns with the boundary.
- In **Contact** position, the tool contacts the boundary.

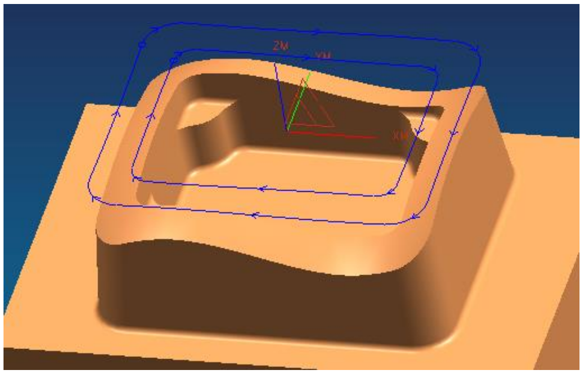
- For the **Plane**, choose **User-Defined**
- Again, set the plane to be **XC-YC** with a **Distance** of **3**
- Click **OK**
- Select the outer loop of the top contour surface as shown in the figure. Remember to select the edges in a continuous order



- Click **OK**

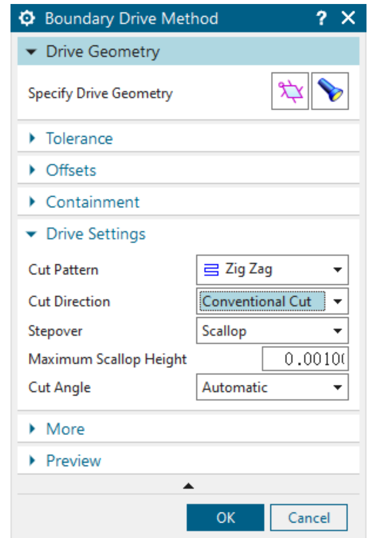
We have trimmed the geometry outside the loop. Now we have to trim the geometry inside the inner loop so that the only geometry left will be the area between the two loops.

- Choose the **Mode** to be **Curves/Edges**
- Choose the **Material Side** to be **Inside** and **Tool Position** to be **On**
- Choose the plane to be user-defined at **XC-YC** with a **Distance** of **3**
- Select the inner edges of the contour surface as shown
- Click **OK** to return to the **Boundary Drive Method** window



- Change the **Stepover** method to **Scallop** and enter the height to be **0.001** and click **OK**
- Click on **Cutting Parameters**
- Change the **Tolerance** values in the **Stock** tab so that the **Part Intol** and **Part Outtol** is **0.001**
- Click on the **More** tab button and enter the value of **Max Step** as **1.0**

➢ Click **OK**



In the main parameters window,

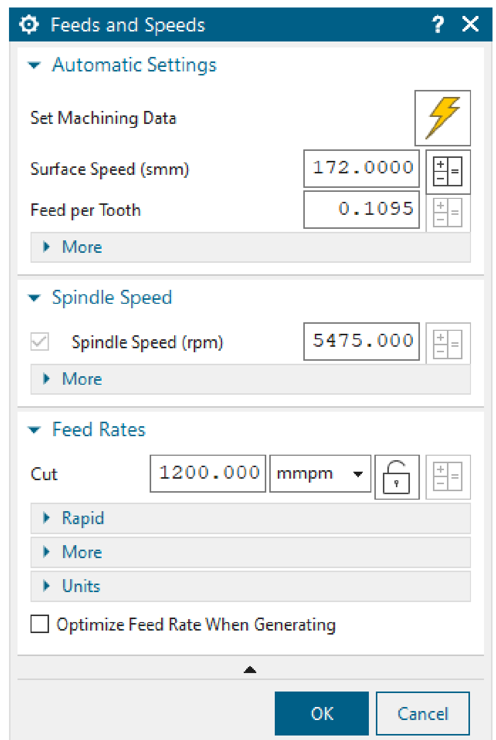
➢ Create a new tool and name it **BEM10**

➢ Change the diameter to be **10 mm** and the lower radius to be **5 mm**.

➢ Click **OK**

➢ Click on the **Feeds and Speeds** icon on the parameters window

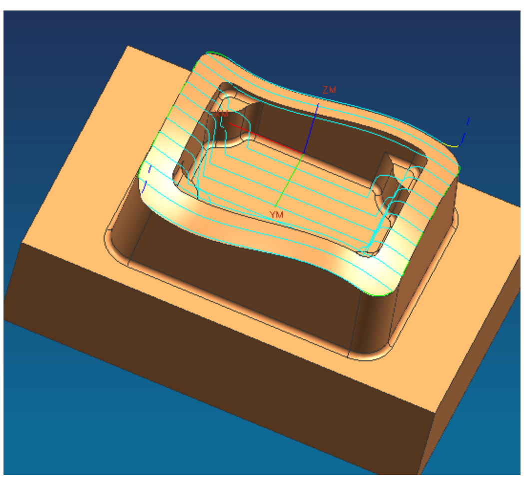
➢ Enter the **Spindle Speed, Feed Rates (Cut)** and calculate the others as shown in the figure on right



➢ Click **OK**

➢ **Generate** the program

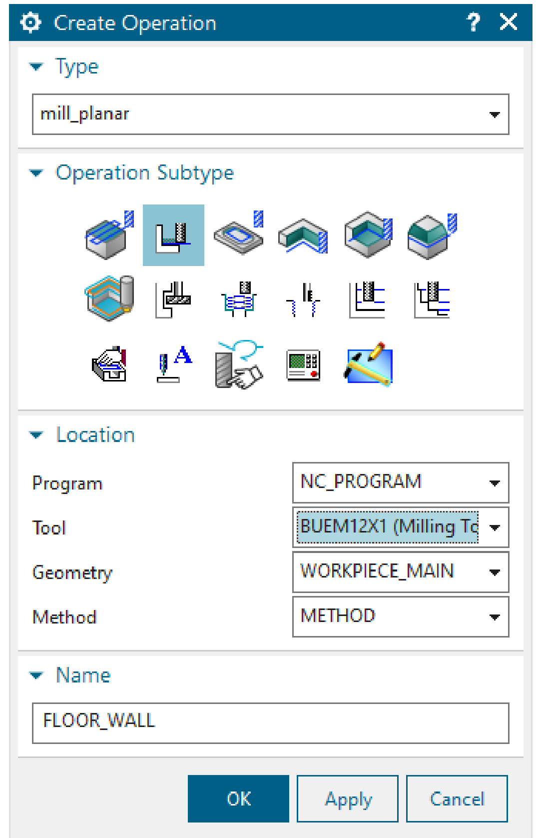
The contour surface is now finished and you can view the simulation by *Tool Path Verification*.



9.4.5 Flooring

Flooring is the finishing operation performed on the horizontal flat surfaces (*Floors*) of the part. In most of the milling processes, flooring will be the final operation of them. All the horizontal surfaces will be finished. This planar operation runs the cutter in a single pass on every face.

- Click on the **Create Operation** icon 
- Change the **Type** to be **mill_planar** at the top of the window
- Change all the options as shown in the figure
- Click **OK**
- In the parameters window, change the **Cut Pattern** to be **Follow Part**
- Change the percent of the tool diameter for **Stepover** to be **40**



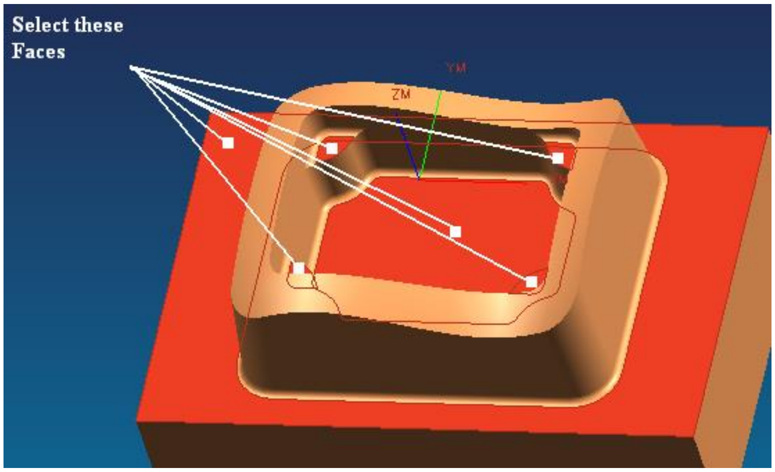
In flooring operations, it is always better to keep the *Stepover* value to be less than half of the diameter of the cutter in order to achieve more flatness on the planar surfaces.

Unlike previous programs, we have to select a cut area.

- Click on the **Specify Cut Area Floor**

- Select the highlighted surfaces shown in the figure below

In case you are not able to select the surfaces as shown go to *Part Navigator* and *Hide* the *Blank* (to choose the blank, you can right click on the blank, use *Select from List…)*, select the surfaces and *Unhide* the *Blank* again.



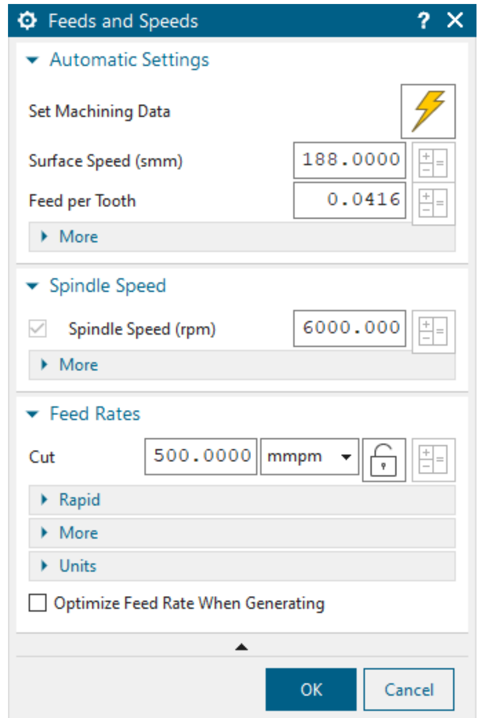
- Click **OK**
- Click on **Cutting Parameters** in the main parameter window
- Choose the **Stock** tab button and enter the **Intol** and **Outtol** values as **0.001**
- Click **OK**
- Click on **Feeds And Speeds**

In the main Parameters window,

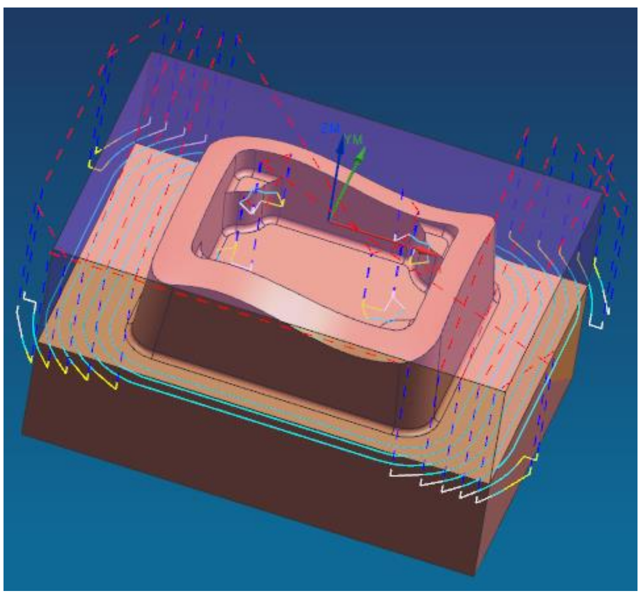
- Create a new tool and name it **BEF105**
- Change the diameter to be **10 mm** and the lower radius to be **5 mm**
- Click **OK**

Because this is a *Flooring* operation, it is better to make the spindle speed high and the feed rates low compared to the previous operations.

- Enter the values for **Spindle Speed**, **Feed Rates (Cut)** and calculate the others as shown in the figure
- Choose **OK**
- Generate the program. Then replay and verify the cutter path



The following figure shows the *Tool Path* display for the flooring.



9.5 POST PROCESSING

The primary use of the *Manufacturing Application* is to generate tool paths for manufacturing parts. Generally, we cannot just send an unmodified tool path file to a machine and start cutting because there are many different types of machines. Each type of machine has unique hardware capabilities, requirements and control systems. For instance, the machine may have a vertical or a horizontal spindle; it can cut while moving several axes simultaneously, etc. The controller accepts a tool path file and directs tool motion and other machine activity (such as turning the coolant or air on and off).

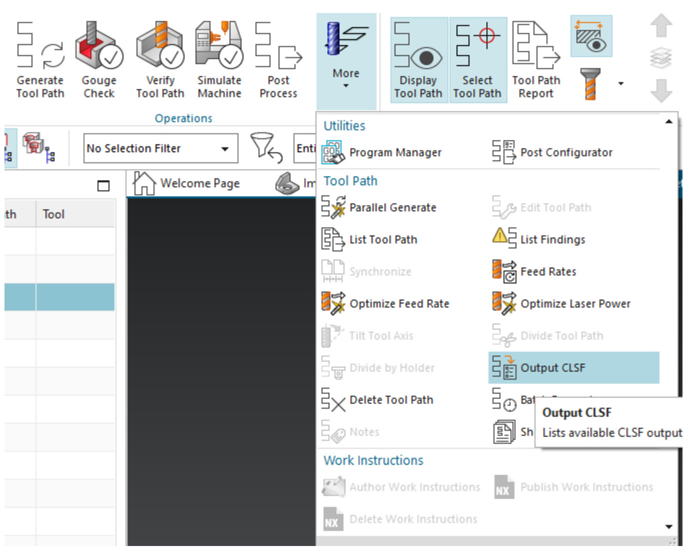
Naturally, as each type of machine has unique hardware characteristics; controllers also differ in software characteristics. For example, most controllers require that the instruction for turning the coolant on be given in a particular code. Some controllers also restrict the number of M codes that are allowed in one line of output. This kind of information is not in the initial NX tool path. Therefore, the tool path must be modified to suit the unique parameters of each different machine/controller combination. The modification process is called *Post Processing*. The result is a *Post Processed* tool path.

There are two steps involved in generating the final post-processed tool path.

1. Create the tool path data file or called *CLSF (Cutter Location Source File)*.
2. Post process the *CLSF* into machine CNC code (*Post Processed* file). This program reads the tool path data and reformats it for use with a particular machine and its accompanying controller.

9.5.1 Creating CLSF

After an operation is generated and saved, the resulting tool path is stored as part of the operation within the part file. *CLSF (Cutter Location Source File)* provides methods to copy these internal paths from the operations in the part file to tool paths within the *CLSF*, which is a text file. The *GOTO* values are a "snapshot" of the current tool path. The values exported are referenced from the MCS stored in the operation. The CLS file is the required input for some subsequent programs, such as postprocessors.

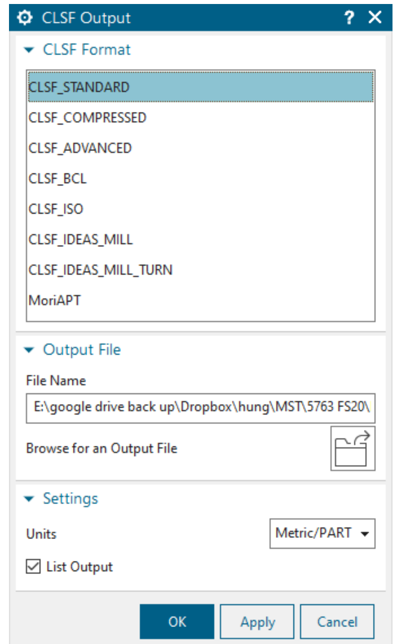


- ➢ Click on one of the programs that you want to post process in the **Operation Navigator**
- ➢ Click on **Output CLSF** in the **Operations** toolbar

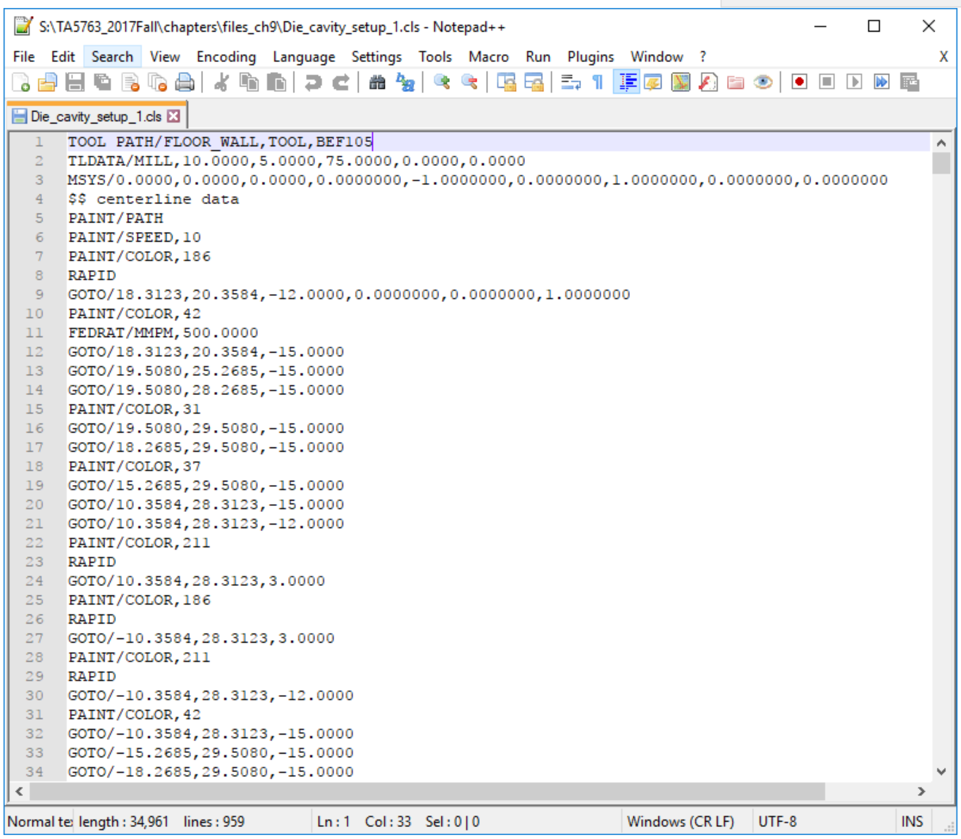
A window will pop up to select the *CLSF Format.*

- ➢ Choose **CLSF_STANDARD** and enter a location for the file

➢ Choose **OK**



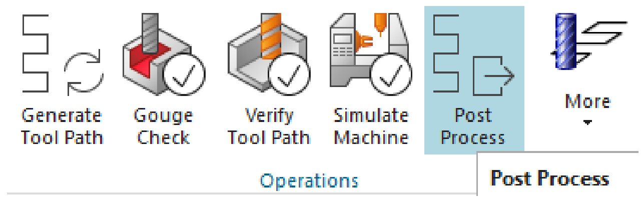
The *CLSF* file will be created. It will be similar to the figure below. The contents of the file contain the basic code of the cutter motion without any information about machine codes and control systems. This file can be used for post-processing any machine control. The extension of the file is **.cls**.



Any program that has been output to *CLSF* or *Post Processed* will have a green checkmark next to it in the *Operation Navigator*.

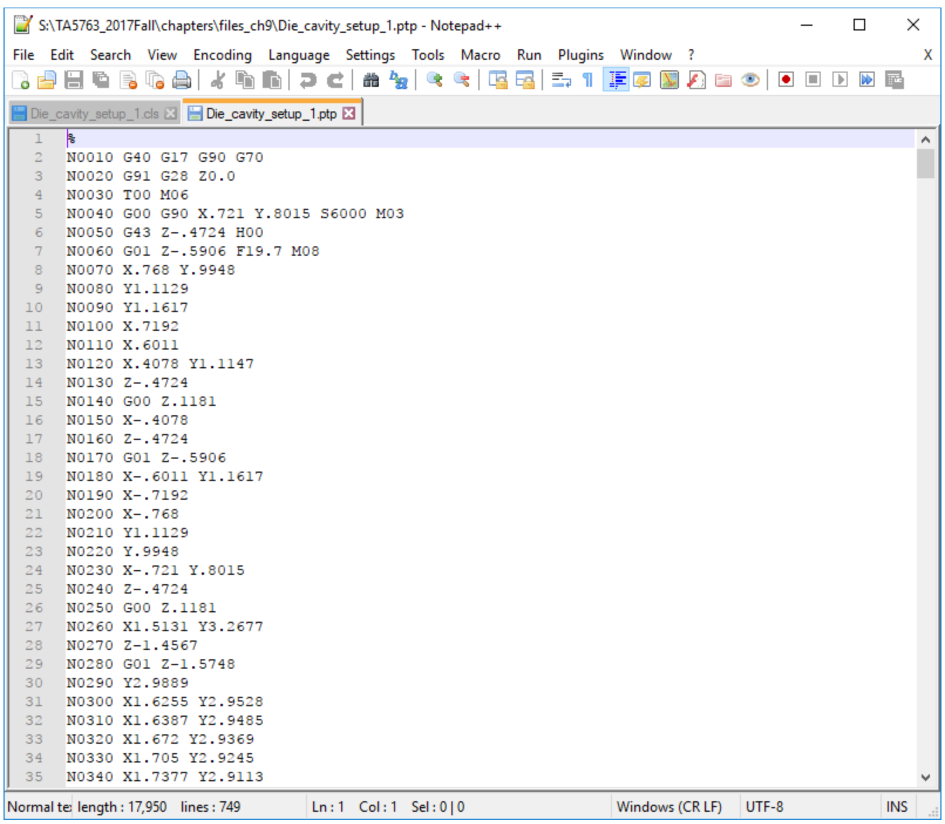
9.5.2 Post Processing

- Click on a program in the **Operation Navigator** that you want to post process
- Click **Menu** → **Tools** → **Operation Navigator** → **Output** → **Postprocess** or from the **Home** tab as shown



- Select the **MILL_3_AXIS** machine and enter a location for the file
- Select **OK**

This will create the *Post Processed* file for the target machine. You can find the block numbers with *G and M codes* concerning the machine controller type. The extension of the file is **.ptp**.




```
%
N0010 G40 G17 G90 G70
N0020 G91 G28 Z0.0
N0030 T00 M06
N0040 G00 G90 X.721 Y.8015 S6000 M03
N0050 G43 Z-.4724 H00
N0060 G01 Z-.5906 F19.7 M08
N0070 X.768 Y.9948
N0080 Y1.1129
N0090 Y1.1617
N0100 X.7192
N0110 X.6011
N0120 X.4078 Y1.1147
N0130 Z-.4724
N0140 G00 Z.1181
N0150 X-.4078
N0160 Z-.4724
N0170 G01 Z-.5906
N0180 X-.6011 Y1.1617
N0190 X-.7192
N0200 X-.768
N0210 Y1.1129
N0220 Y.9948
N0230 X-.721 Y.8015
N0240 Z-.4724
N0250 G00 Z.1181
N0260 X1.5131 Y3.2677
N0270 Z-1.4567
N0280 G01 Z-1.5748
N0290 Y2.9889
N0300 X1.6255 Y2.9528
N0310 X1.6387 Y2.9485
N0320 X1.672 Y2.9369
N0330 X1.705 Y2.9245
N0340 X1.7377 Y2.9113
```

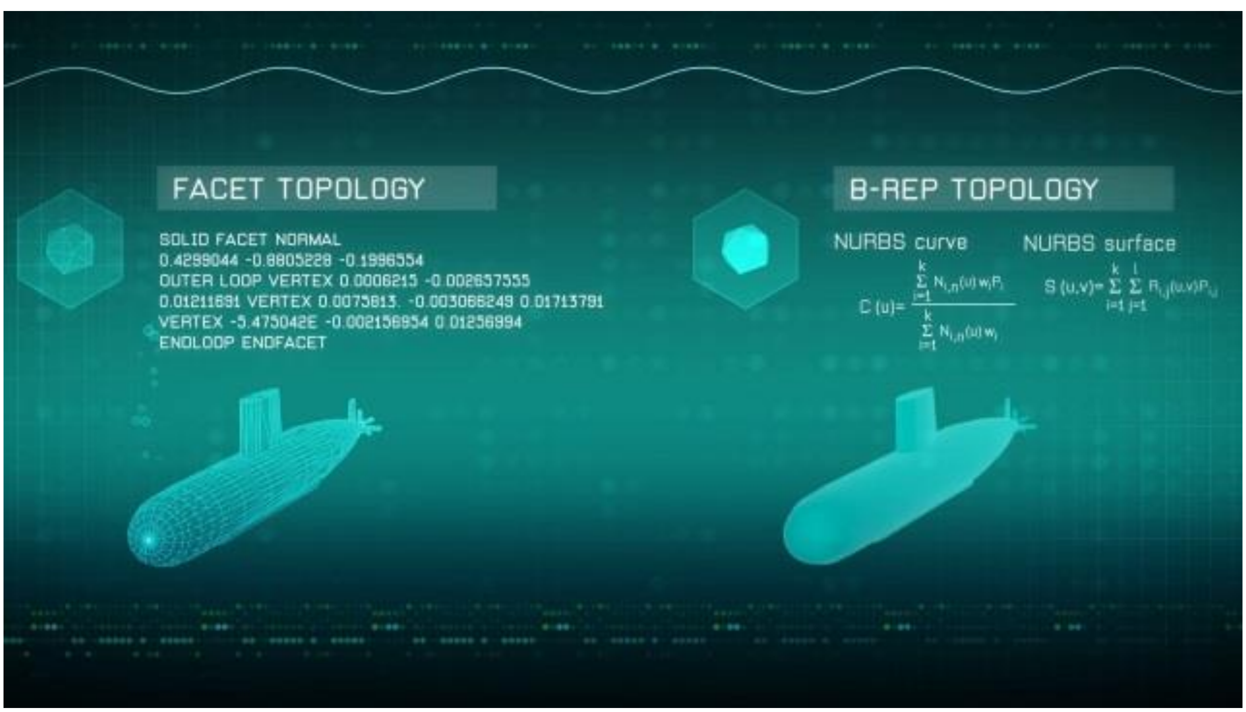
The final output (*filename.ptp*) file can be transferred to a machine for conducting an actual milling operation. The entire sequence starting from the transfer of the model into the *Manufacturing* module to the transfer of the files to the machine and fabricating the raw piece into the final part is called *Computer Aided Manufacturing*.

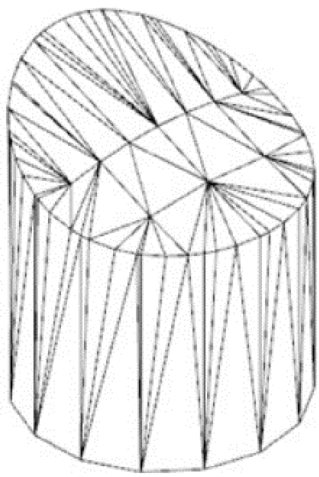
CHAPTER 10 – ADDENDUM TOPICS

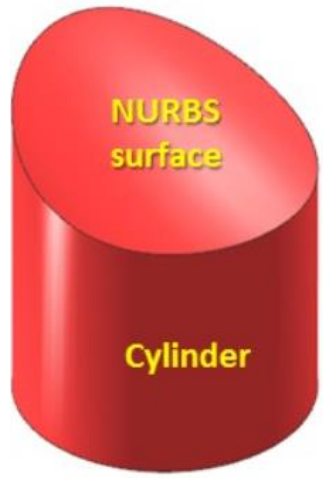
10.1 CONVERGENT MODELING

Convergent Modeling technology provides a unified 3D modeling capability on combinations of facet and B-Rep data. These two data formats have different geometric characteristics and as a result, one format is better suited to some applications than the other. Facet data is characterized by triangular meshes to approximate surfaces and tends to generate larger data sets. Applications that use facet data include digital mock-up, gaming and animation. B-Rep data is characterized by mathematically defined surfaces that represent accurate, closed (solid) volumes using smaller data sets. Applications that use B-Rep data include 3D modeling for engineering and manufacturing.

A proliferation of facet data in high growth areas like 3D scanning, topology optimization, and 3D printing means that design engineers often need to move facet data into 3D modeling systems that were designed for B-Rep data. Convergent Modeling technology enables 3D product modeling on both facet and B-Rep data sources in a single environment, while eliminating the complexity, errors, and delay of conversion between the two types of data. It enables the user to create 3D models based on import facet data (e.g., 3D part scanning profile file or an STL file), which significantly saves the modeling time.







Facet Topology B-Rep Topology

There are many products which are often developed based on the reverse engineering process starting with physical objects. A few common examples are:

- Hand-held objects such as computer mice and the handles of guns and power tools, whose "feel" needs to be assessed during development.
- Objects like toys and jewelry, which are often modeled as physical prototypes by craftsmen who have no CAD skills.
- Designs that have been modified during physical try-out, such as molds or stamping dies.
- Objects for which no CAD data is available because it has been lost or is owned by someone who is not willing to share.

In each of the above cases, the design starts out in physical form, and the physical part is scanned to produce digital data. The 3D scan data typically consists of a "point cloud" or an STL file. Next comes a "surfacing" step in which the scan data is converted to a traditional CAD model. This is a tedious manual process, and it requires a CAD modeler with surfacing skills. The surfacing step is often a serious bottleneck that delays the start of downstream work. NX Convergent Modeling enables the surfacing step to be skipped in the part modeling process.

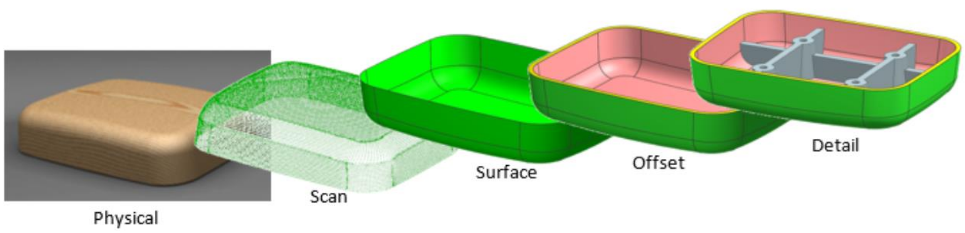
The following example illustrates the difference between the traditional modeling approach and the new approach using [NX Convergent Modeling] with facet data taken by scanning a physical object as the input for reverse engineering. As shown below, the following steps are used in the traditional modeling process to create a shell model:

(1) Use a scanning device such as a coordinate measuring machine to obtain point data from the shell and import the data to NX or some other CAD software.

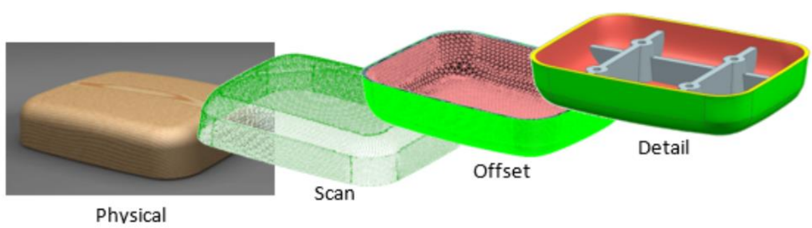
(2) Create an analogous surface that matches the imported point data.

(3) Offset the surface created in Step 2 to create an inner wall.

(4) Add features such as ribs, support cylinders, threaded holes, etc., for increasing the shell's strength and assembling it with other components.

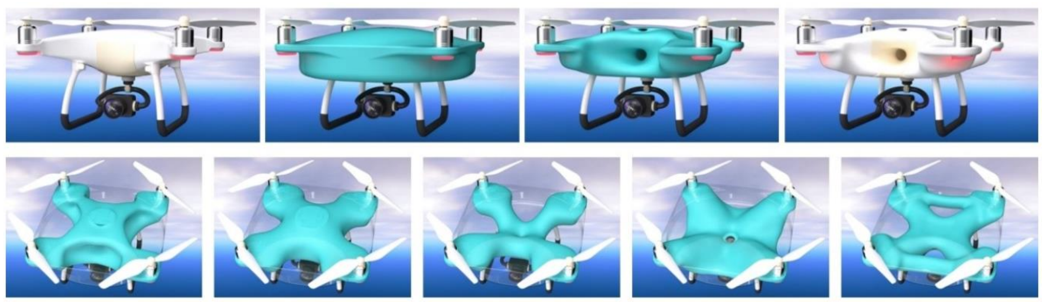


By using NX Convergent Modeling, you do not need the surfacing step for reverse engineering as shown above, i.e., you can skip Step 2 used in the traditional modeling approach. You could import the scanned point data to directly create an offset surface for the inner wall without having to create an analogous surface first. The steps of using NX Convergent Modeling for reverse engineering is illustrated below.

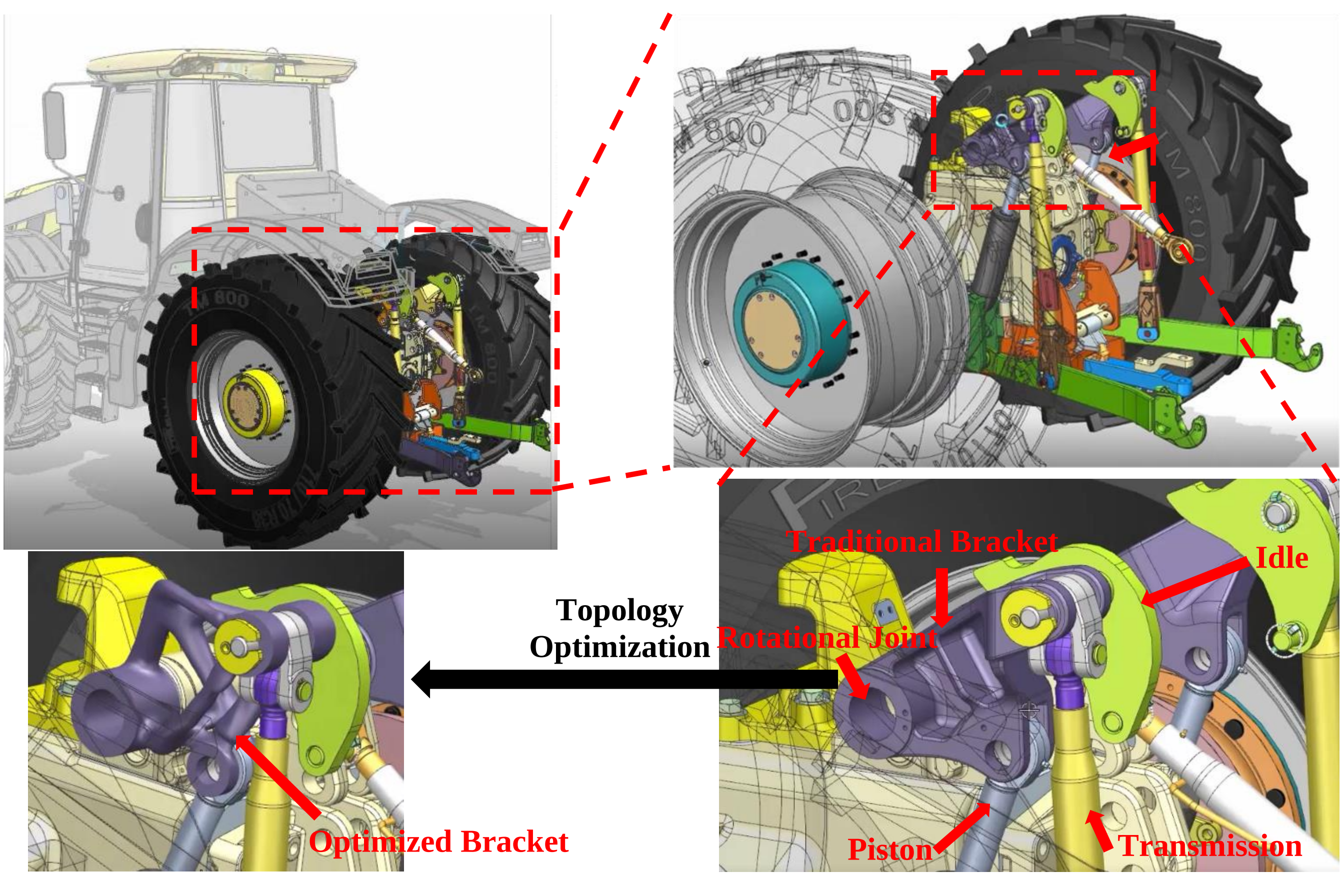


10.1.1 Topology Optimization (A.K.A. Topology Optimizer)

Topology Optimization in NX provides you with a tool to design and optimize models with specific targets (objectives) and constraints. It uses finite element analysis to conduct structural optimization based on your design constraints, such as weight or volume allowance, to achieve desired targets. NX Topology Optimization can minimize material and fabrication costs while satisfy the desired performance. The drone figures below show an example with different results obtained from changing the Topology Optimization settings to optimize its volume in NX.

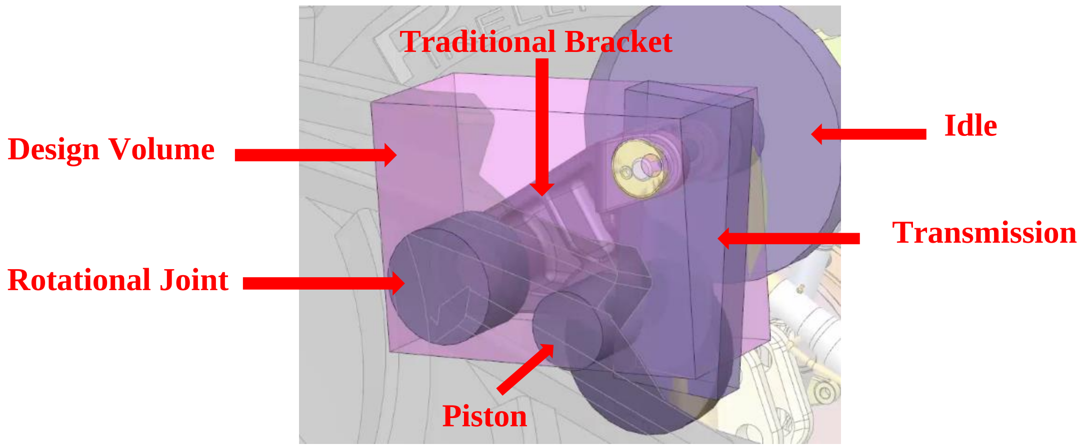


In another example (shown below), we use a bracket of the front-end loader, which joins different components, to show how NX Topology Optimization can be used to reduce its volume and weight.

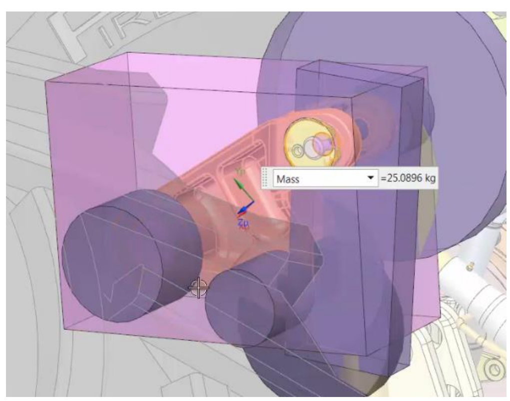


To conduct the topology optimization of this traditional bracket in NX, there are several steps shown below.

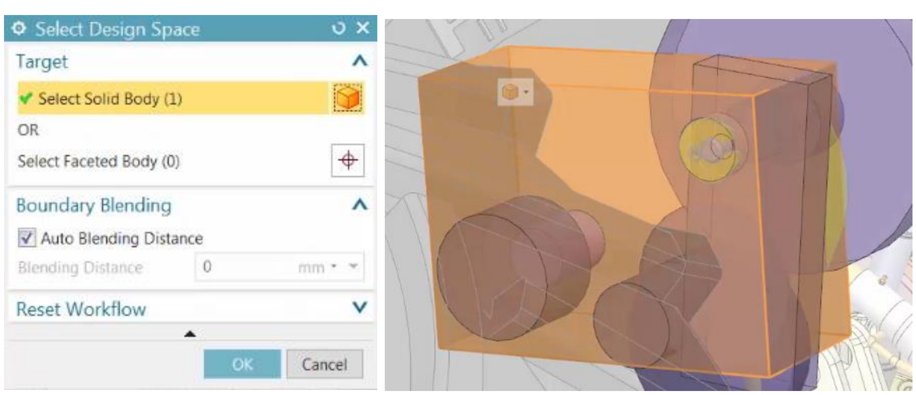
- First, we create the **Design Volume** envelope to define the bracket's size and **cylinders** to represent the joined components



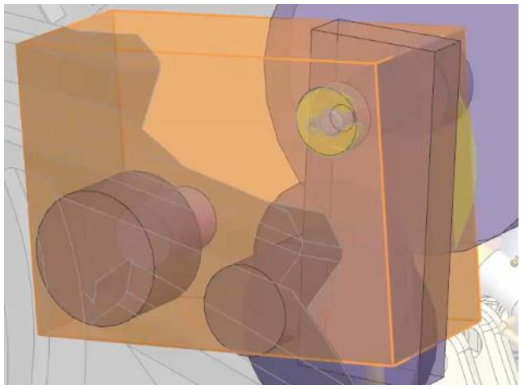
- Use **Measure** to measure the mass of the traditional bracket (25.0896 kg) to let us use this mass as the reference when we apply the mass constraint.

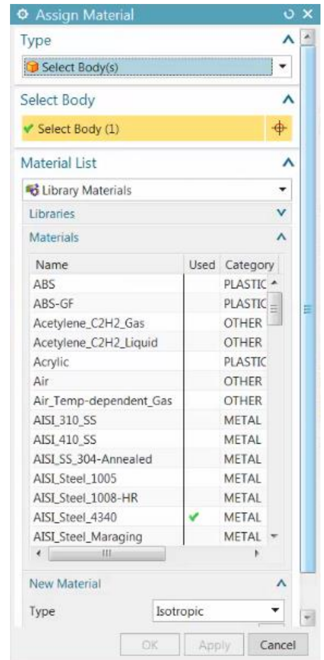


- **Select Design Space** to select **Design Volume**

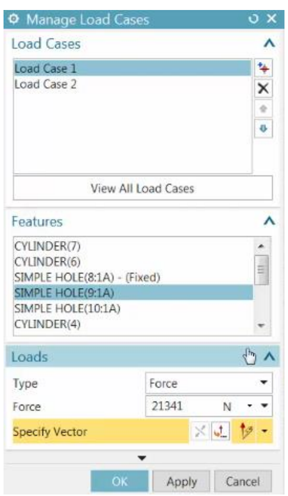


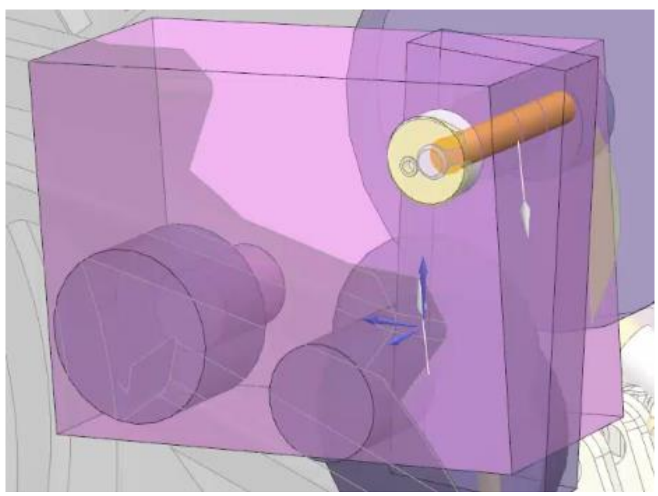
➢ **Assign Material** (AISI_Steel_4340) to **Design Volume**

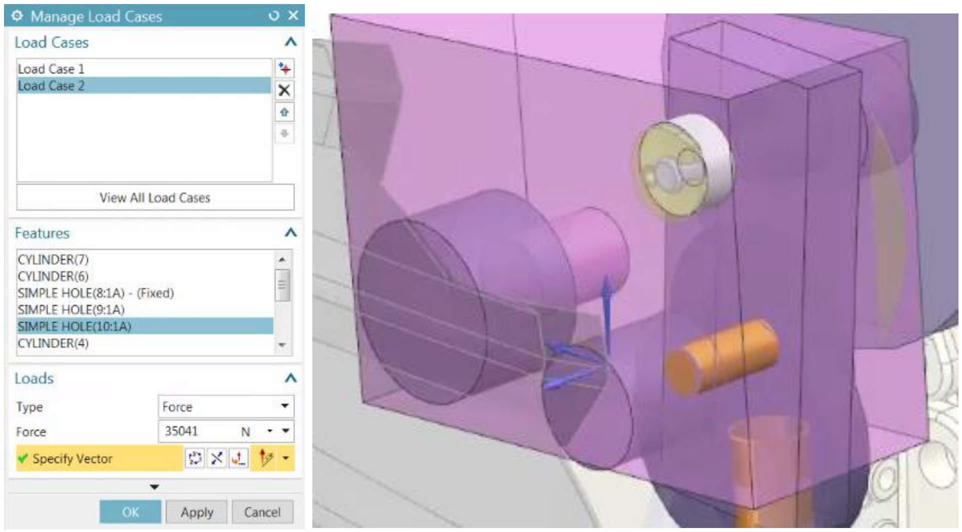


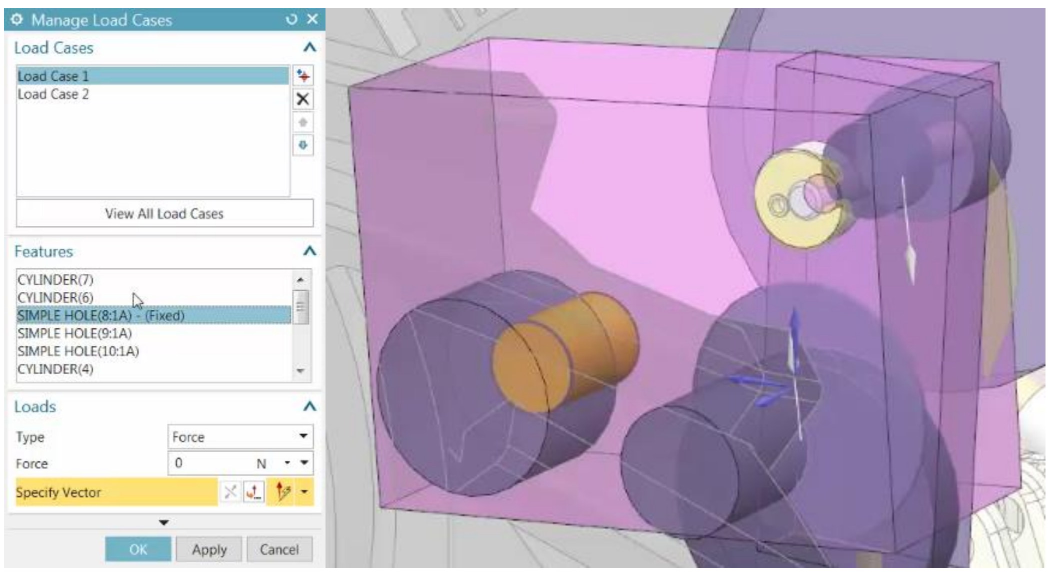


➢ **Manage Optimization Features** to manage global loads (Apply 21341 N to SIMPLE HOLE (9:1A) and 35041 N to SIMPLE HOLE (10:1A)) and Fixed constraint to SIMPLE HOLE (8:1A).

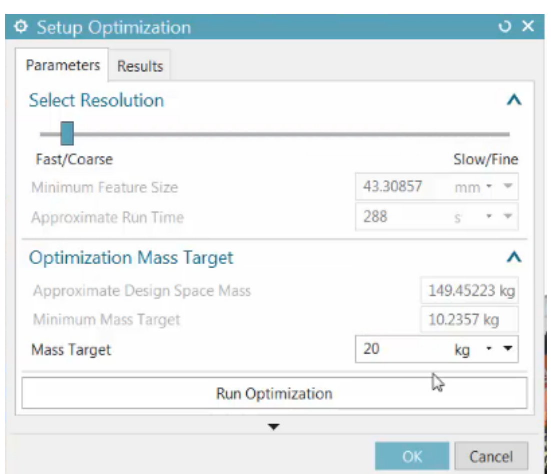




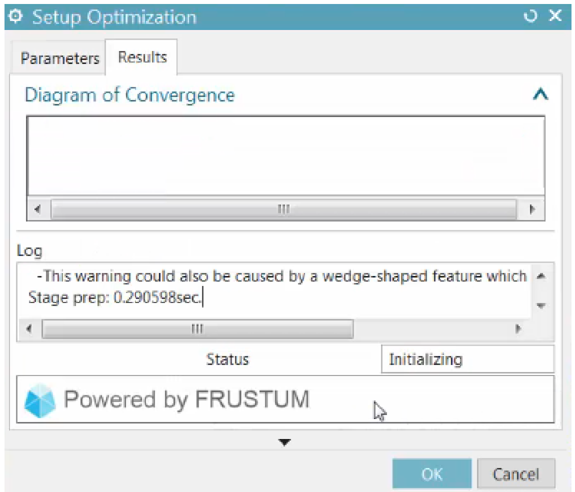




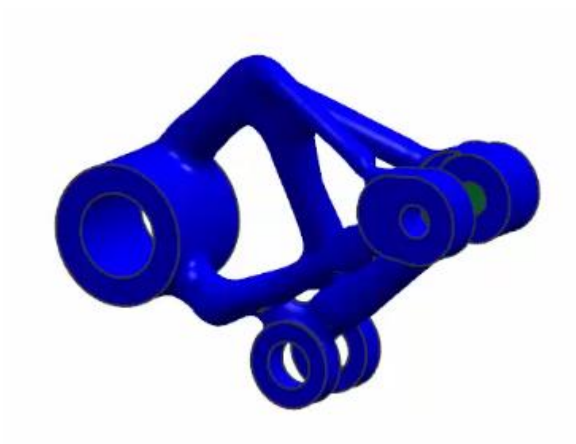
➢ **Setup Optimization** to set the *Mass Target* to 20 kg



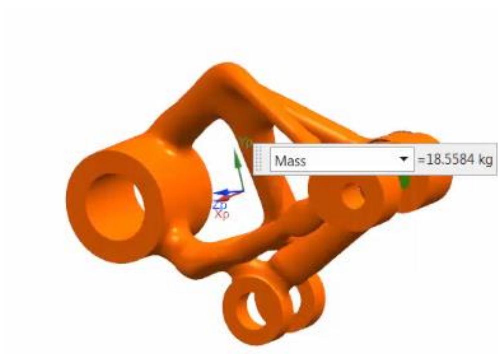
➢ **Run Optimization** to run the topology optimization calculation



➢ After the optimization calculation is done (end result with organic look), use **Convert Facet Body** to get the optimized bracket structure



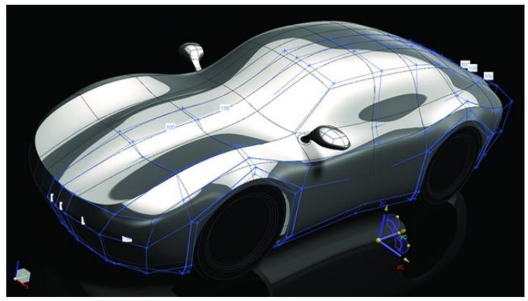
➢ **Assign Materials** and **Measure** to get the bracket mass of 18.558 kg



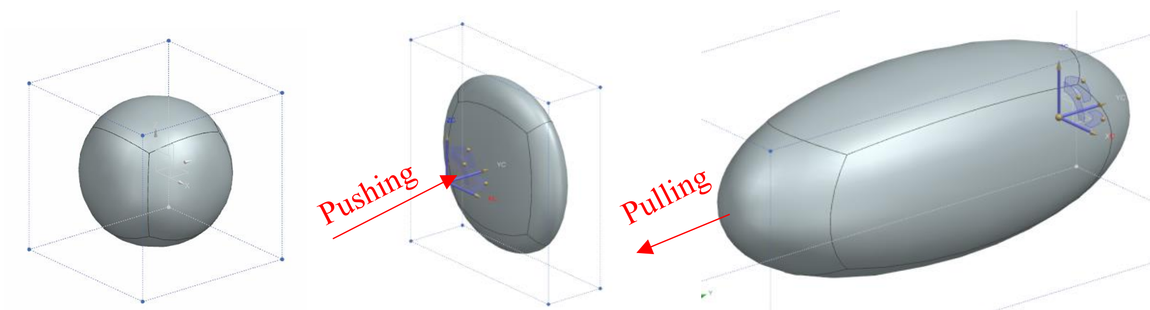
From this bracket design example, from the topology optimization result you can see that NX Topology Optimization can be used to significantly reduce the weight of a design part (i.e., the mass of bracket was reduced 35.2% from 25.086 kg to 18.558 kg) compared to the traditional bracket, thus the Topology Optimization not only minimizes the material usage but also saves the manufacture time and cost.

10.1.2 NX Realize Shape

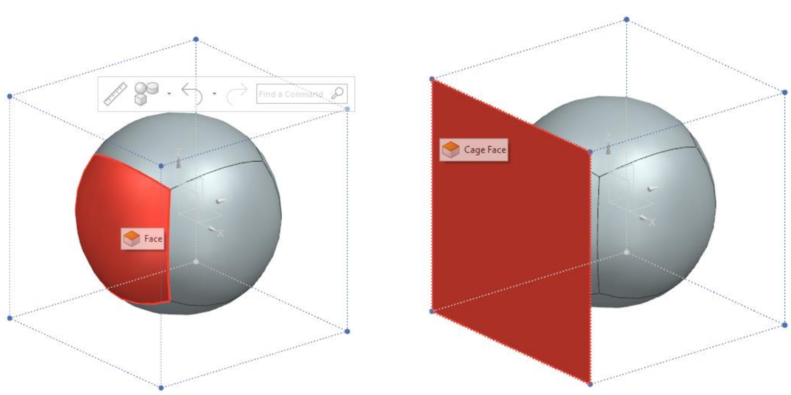
[NX Realize Shape]() is a powerful surface feature modeling tool. It allows you to create a high-quality shape geometry like a car shell model as shown in the figure below relatively easily by using an intuitive freeform modeling method.



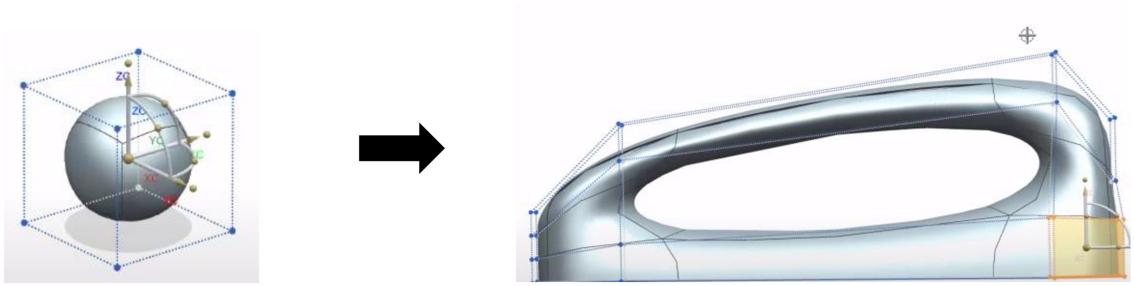
In the NX Realize Shape modeling environment, you start with the object selection of a primitive shape (sphere, cylinder, rectangle, etc.). After creating the primitive-shape object (shown in the left figure below), you can manipulate the shape of this object by pushing a Cage Face (middle figure) or pulling a Cage Face (right figure).



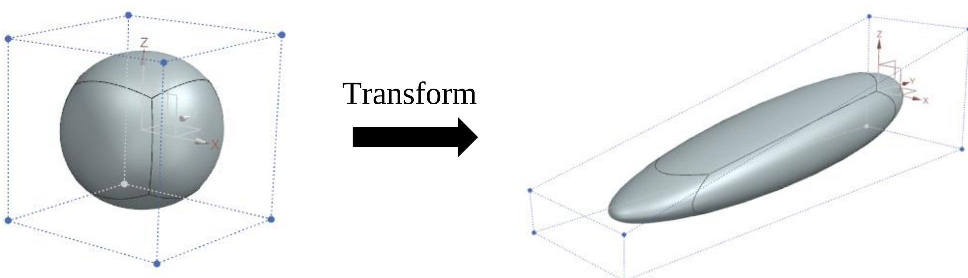
Note that an object's Face is defined by controlling a Cage Face as shown below. Transforming the Cage Face will change the Face shape of the object.



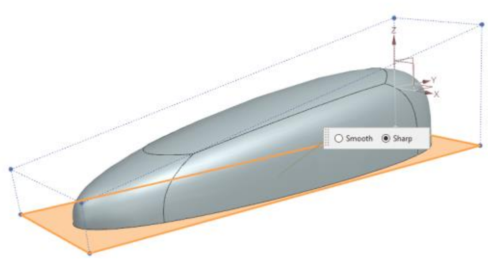
For example, some of the steps are given below to illustrate how to shape an Iron model from a sphere primitive using NX Realize Shape.



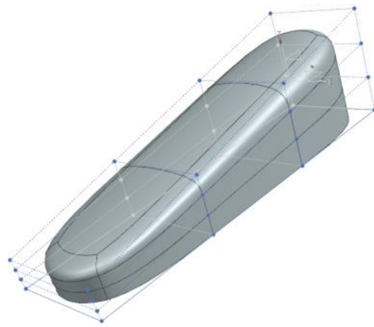
- ➢ We first create a primitive shape "sphere"
- ➢ **Transform Cage** to transform an edge of a Cage Face as the first step of several steps to finally shape the sphere to an oval shown below



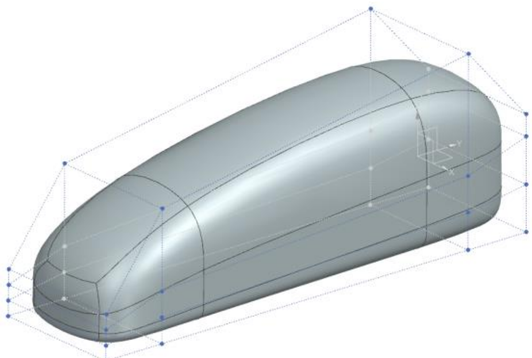
- ➢ **Set Continuity** to flatten the bottom face



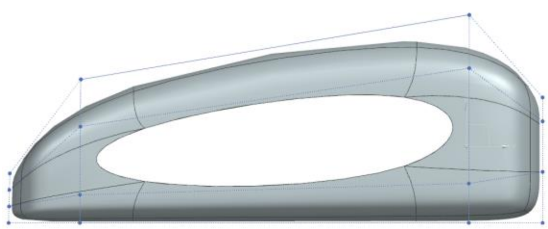
- ➢ **Split Face** to split each of the two cage faces on its right side and left side into nine cage faces and also to split each of the cage faces on the top side, front side, and rear side into three cage faces



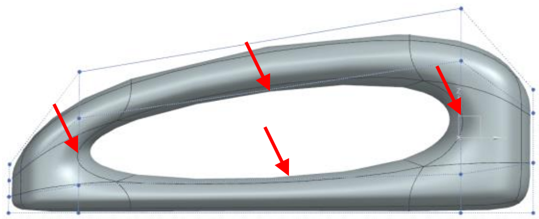
- **Transform cage** to transform some of the cage edges to form the shape below



- **Delete** the middle cage face of the nine cage faces on the right side and also the middle cage face of the nine cage faces on the left side



- **Fill** edges

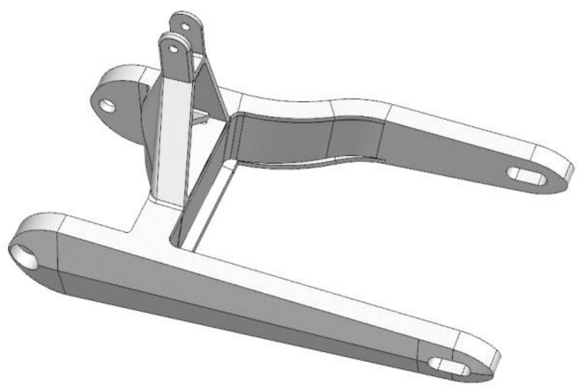


As you can see in the above example, an iron can be modeled starting from a sphere by using NX Realize Shape. This makes freeform surface modeling easier for CAD users.

10.1.3 Design for Additive Manufacturing

Additive manufacturing (AM), popularly known as 3D printing, has been used to fabricate parts with complex geometries layer by layer using various techniques such as Stereolithography (SLA), Fused Deposition Modeling (FDM), Powder Bed Fusion (PBF), etc. The NX Design for Additive Manufacturing provides helpful features for designing and verifying your AM models to prevent printability issues, e.g., checking if support structures are needed in some regions, finding out the best printing orientation, etc.

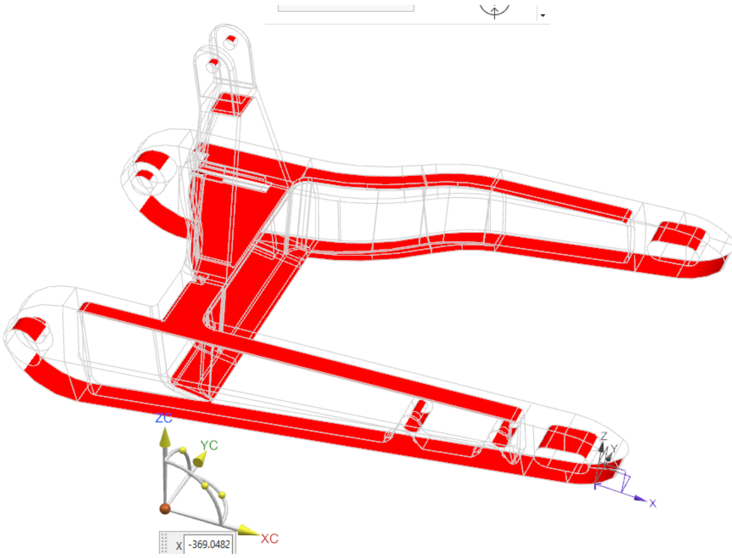
Here we use an object model with complex geometry, which is difficult to fabricate using the traditional manufacturing method of machining, to illustrate some features in the NX Design for Additive Manufacturing.



Overhang

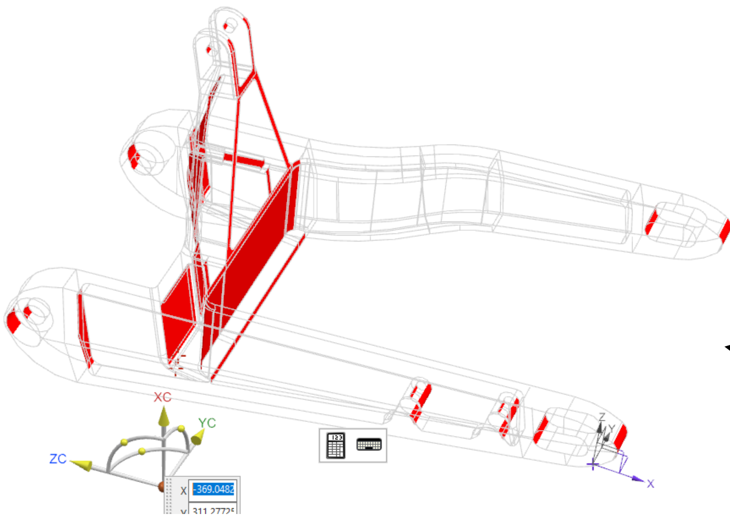
In the Design for Additive Manufacturing, the **Overhang** is an essential feature in NX to visualize as it checks the Overhang Area that requires support structure when fabricating a three-dimensional part using additive manufacturing. The Overhang Area varies with part building direction. The Maximum Overhang Angle depends on the additive manufacturing technique and needs to be input by the NX user.

Here we use some figures to illustrate the Overhang Area with the Maximum Overhang Angle of 45° and different building directions below. The red regions represent the Overhang Area.



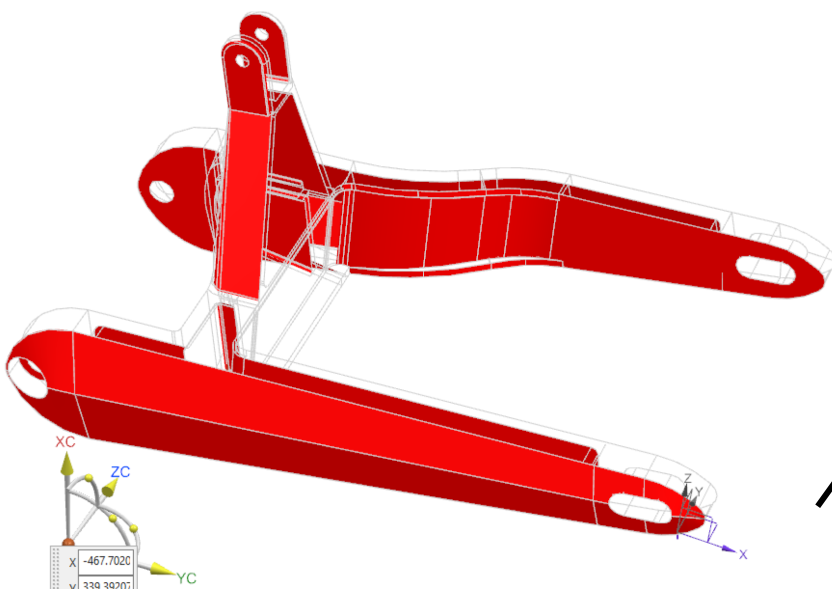
Building direction

Overhang Area: 712.7 cm^2



Building direction

Overhang Area: 475.3 cm^2



Building direction

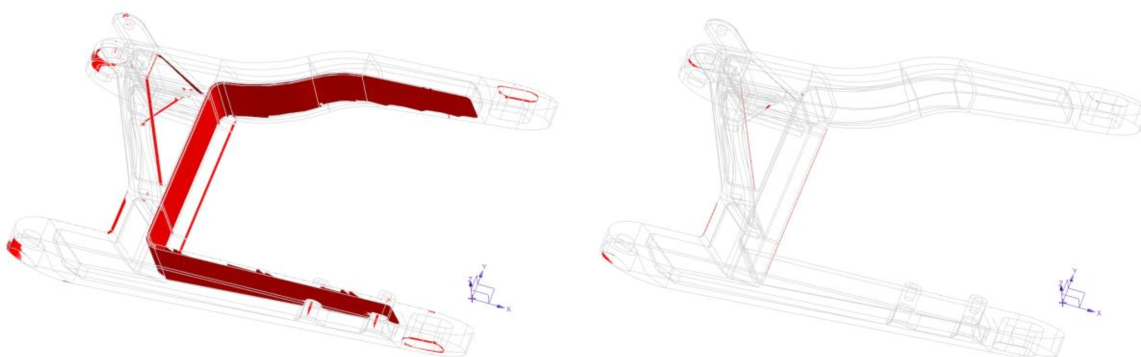
Overhang Area: 1337 cm^2

From the above figures we can see that the Overhang Area ranges from 475.3 cm^2 to 1337 cm^2 for the three building directions. The **Overhang** feature allows you to evaluate the Overhang Area for each building direction with a specified Maximum Overhang Angle.

Besides the evaluation of Overhang Area, the NX Design for Additive Manufacturing also provides the optimization feature to help you find out the part orientation that results in the minimum Overhang Area for your part build. This will be demonstrated when we introduce the feature of Optimize Part Orientation.

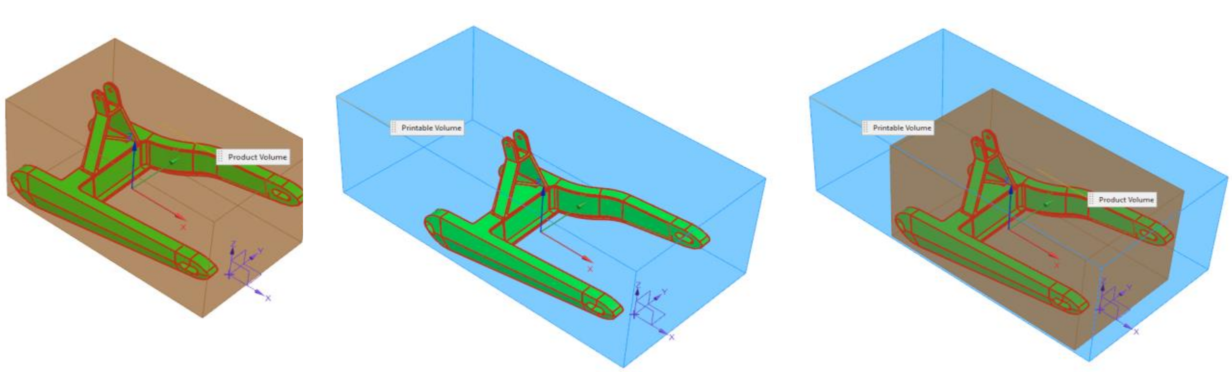
Wall Thickness

The Wall Thickness feature allows you to identify those regions that will have printing difficulty because the wall thickness of the design part is smaller than the minimum wall thickness that the AM machine is capable of. As an example, the figures below display regions (in red) that have wall thickness larger than the capability of the AM machine.

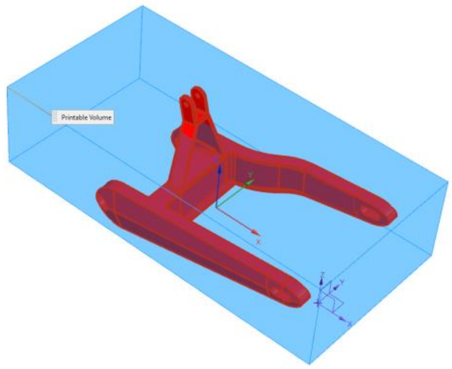


Printable Volume

The **Printable Volume** feature helps you evaluate whether your 3D printer has the printable volume that is big enough to fabricate your part, as illustrated below.

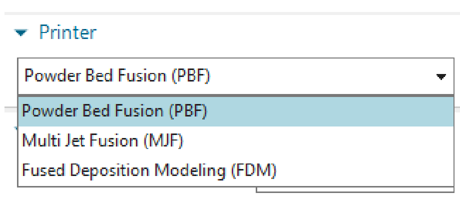


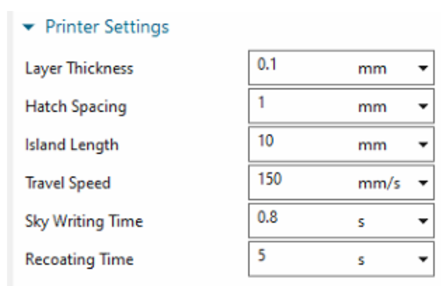
The above figure displays your part and its range (left), printer volume (middle), and both (right). It indicates that the printable volume is larger than your part volume. If your part volume is larger than the printable volume, the part will be shown in red, as seen in the figure below.



Print Time

When you want to fabricate a part using an AM machine, the part fabrication time needs to be considered in order to evaluate whether the part building time will be too much. In the NX Design for Additive Manufacturing, the **Print Time** feature can help you estimate the time taken to print a part. You can estimate the print time by selecting the type of Printer and then entering the values of process parameters (printer settings) you will use for part fabrication.

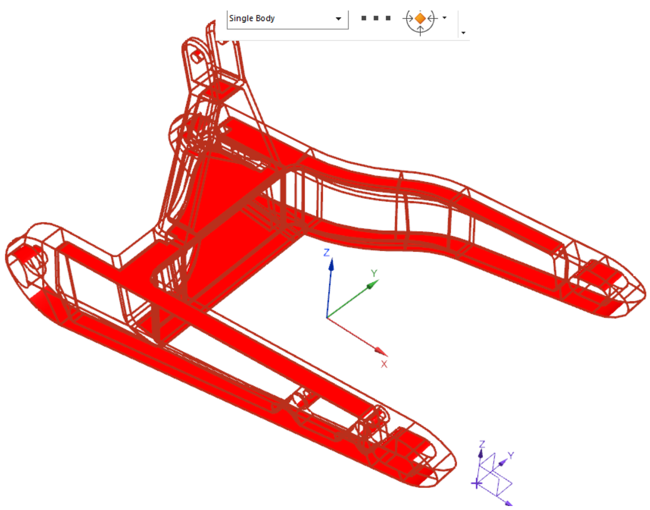




Besides evaluating the **Print Time**, the NX Design for Additive Manufacturing provides the optimization feature to help you find out what part orientation will result in the minimum **Print Time** for your part build. This will be demonstrated when we introduce the feature of **Optimize Part Orientation**.

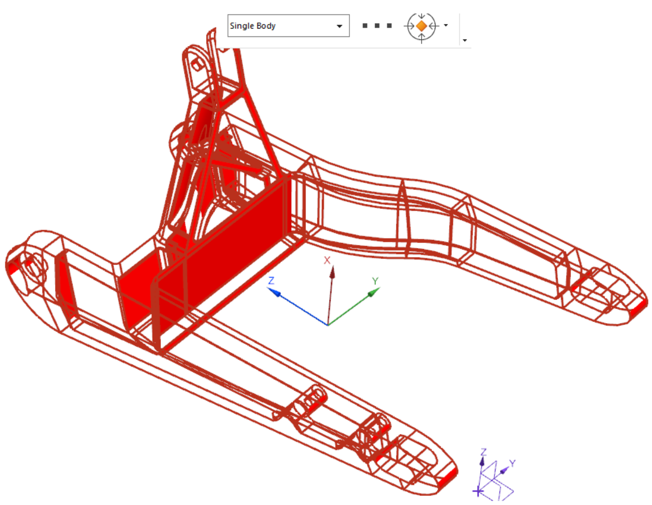
Overheating

In the laser powder bed fusion processes, the **Overheating** feature in NX could be very useful because it helps you identify the risk of overheating caused by laser melting of metal powder. Similar to the Overhang Area described above, the Overheating Area depends on the part building direction. We show the Overheating Areas in three different part building directions below.



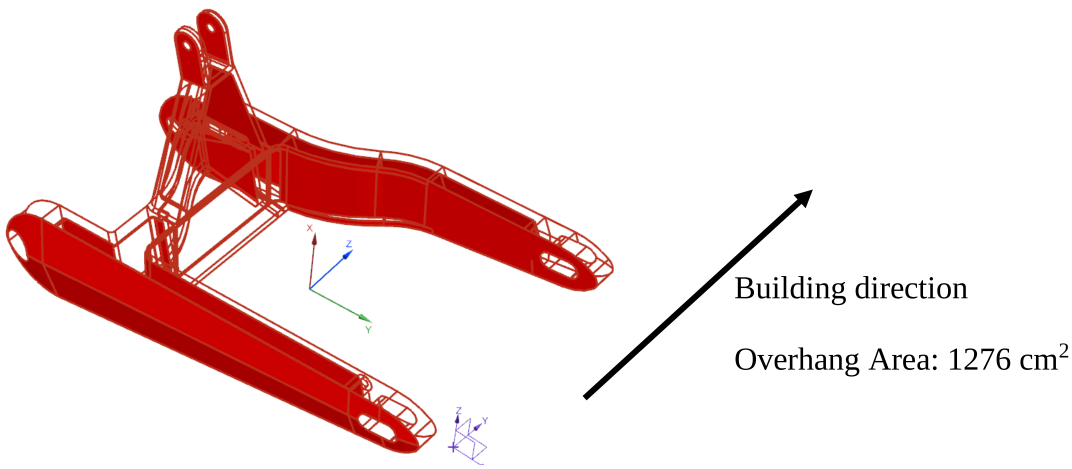
Building direction

Overheating Area: 640.9 cm^2



Building direction

Overhang Area: 394.6 cm^2

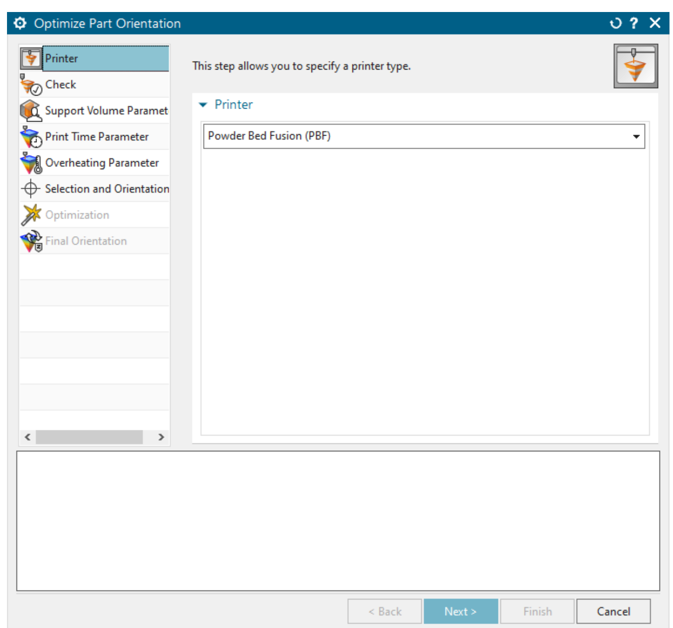


From the above figures, we can see that the Overheating Area ranges from 394.6 cm^2 to 1276 cm^2 for the three building directions. Therefore, this Overheating feature allows you to estimate the Overheating Area.

Besides the evaluation of Overheating Area, the NX Design for Additive Manufacturing also provides the **Optimization** feature to help you find out what part orientation would have the minimum Overheating Area for your part build. It will be demonstrated in the feature of **Optimize Part Orientation** below.

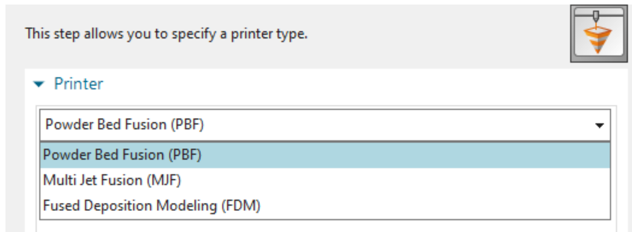
Optimize Part Orientation

The **Optimize Part Orientation** is the most important feature in NX Design for Additive Manufacturing because it could optimize the part orientation to minimize the **Surface Area** (the area needed for the build plate (substrate) for the current part orientation), **Support Volume** (the overhang area), **Print Time**, and **Overheating Area**.

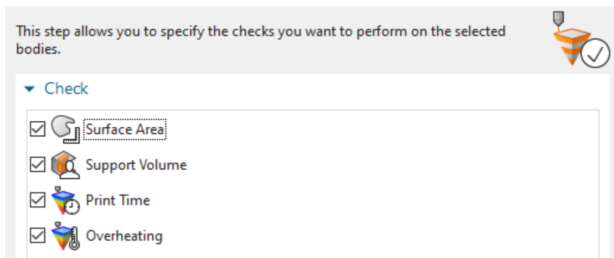


An example of using the **Optimize Part Orientation** feature is given below.

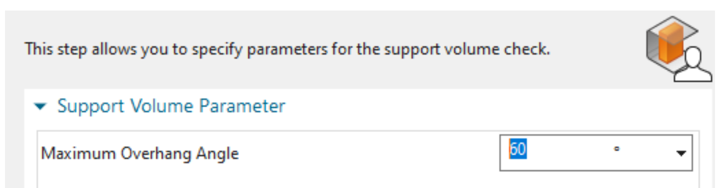
- Select the type of 3D printer you will use



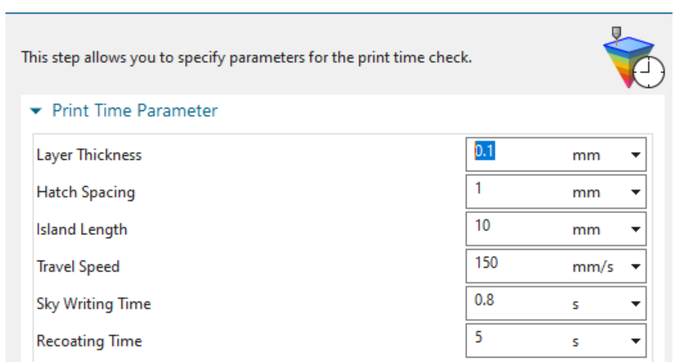
- Select the features that you would like to optimize



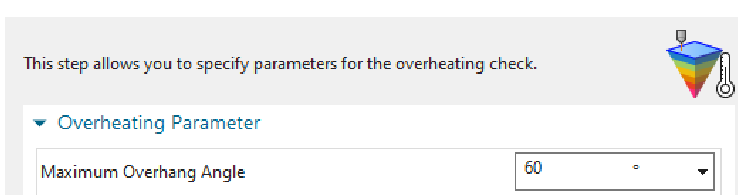
- Specify the **Maximum Overhang Angle**



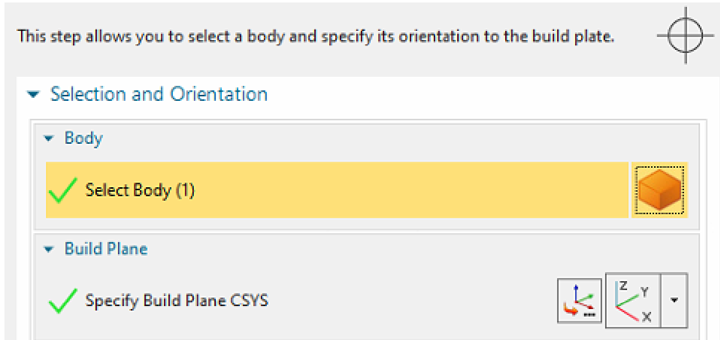
- Specify Printing Parameters



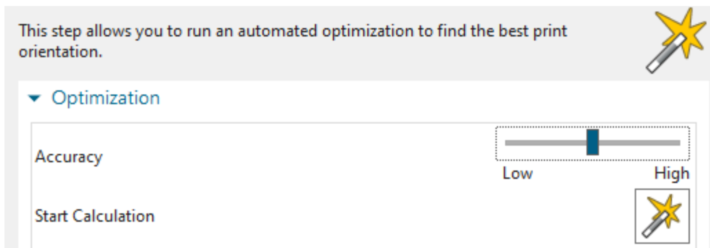
- Specify Overheating Parameter



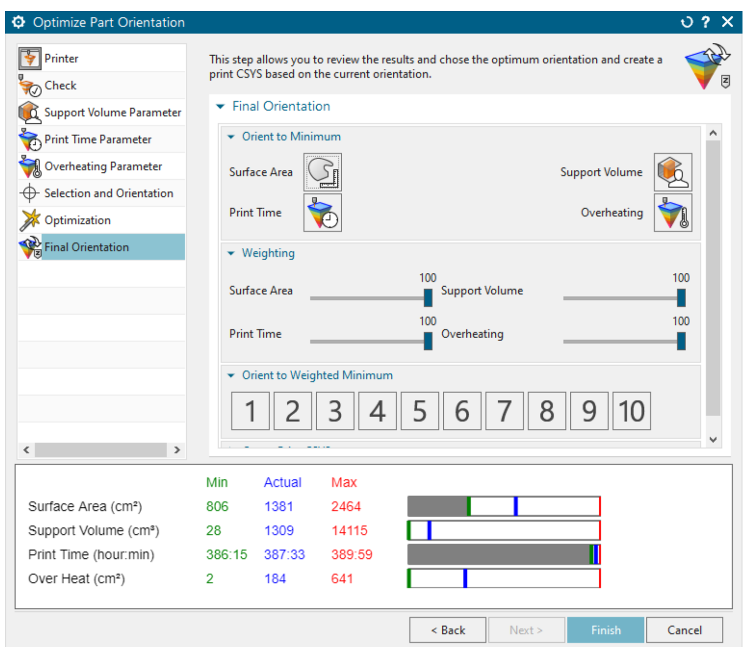
➢ Select Body and specify Building Plane



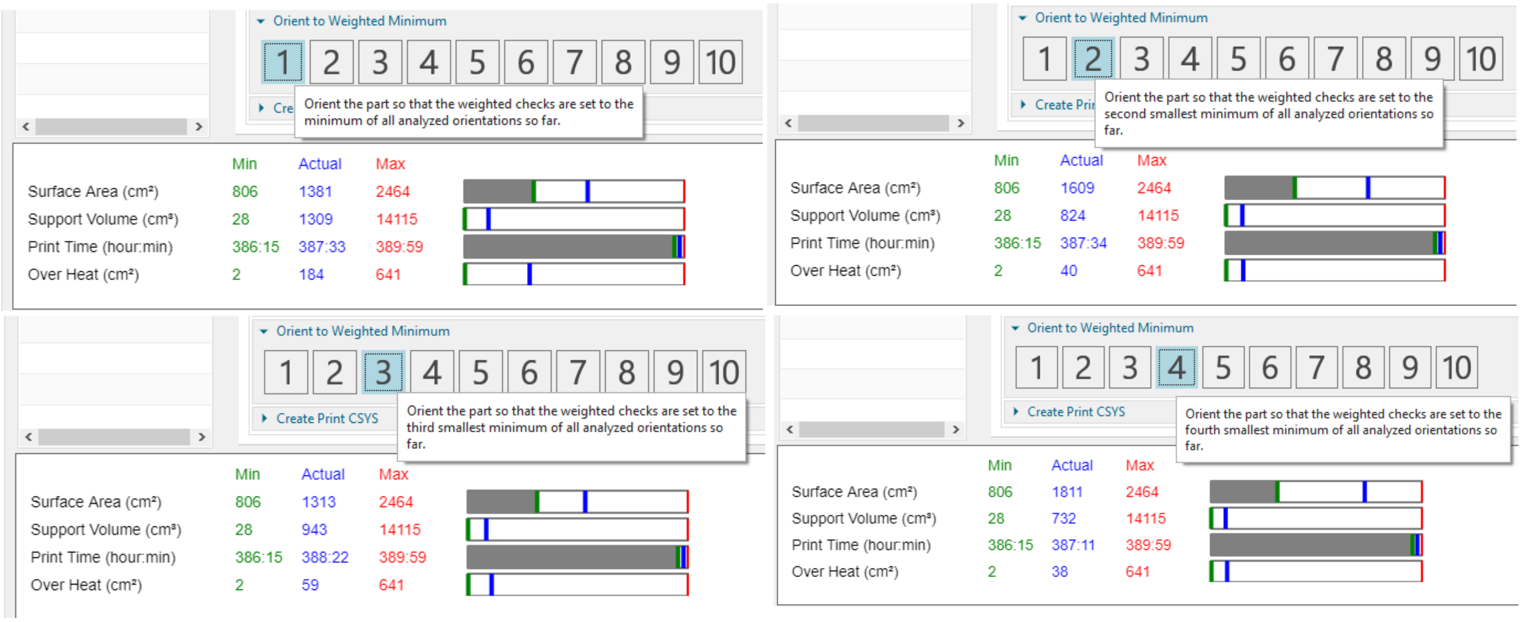
➢ Adjust **Accuracy** and **Start Calculation** to find the optimal part orientation



➢ After the optimization is done, click icons in the Orient to Minimum window to select the orientation for minimizing Surface Area, Support Volume, Print Time, and Overheating. Depending on your preference, the weighting of each check (parameter) could be changed accordingly as shown below.



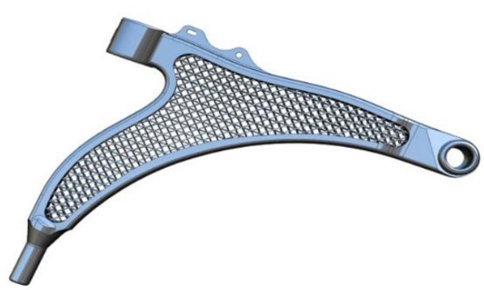
➢ You can also choose 10 part orientations to find different weighted checks.



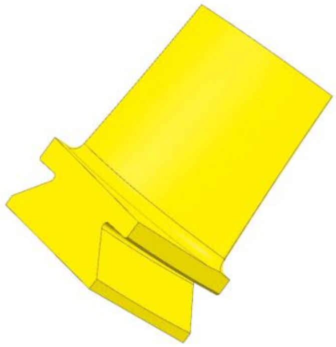
With the **Optimize Part Orientation**, the values of Surface Area, Support Volume, Print Time, and Overheating could be found in different part orientations. Therefore, we could efficiently choose a desired part building direction to satisfy our additive manufacturing needs.

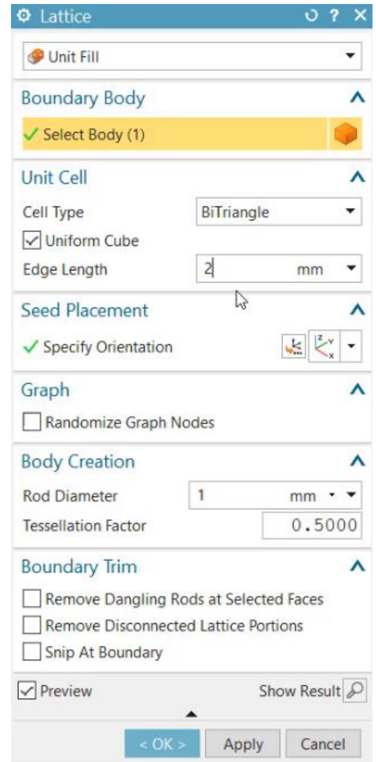
Lattice

Besides the above features, the NX Design for Additive Manufacturing can also allow you to modify your design to reduce weight and cost but still have structural integrity by using the **Lattice** feature. The built-in **lattice** feature makes it easy to embed lattice structures in a solid part. The density, shape, and angle of lattice structure can be modified easily using the NX Convergent Modeling Technology.

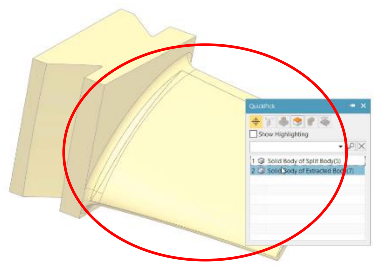


Here, we demonstrate how to use **Lattice** feature inside a part below.





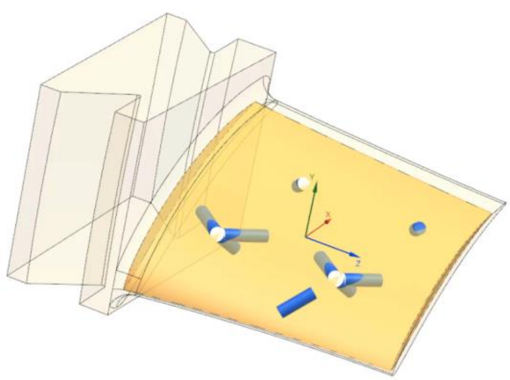
- Click **Lattice** feature as shown in the right figure
- Select this part in *Select Body*



- Change the lattice parameters (Edge Length and Rod Diameter) based on your need as given in the examples below.

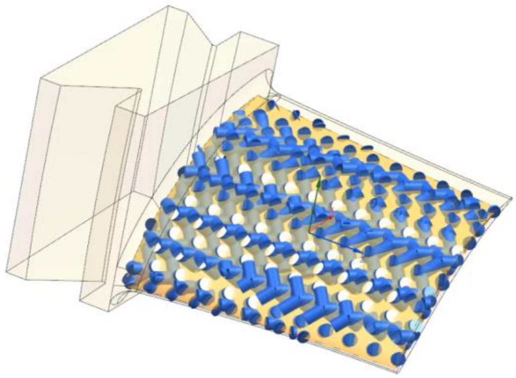
Edge Length: 10 mm

Rod Diameter: 1 mm



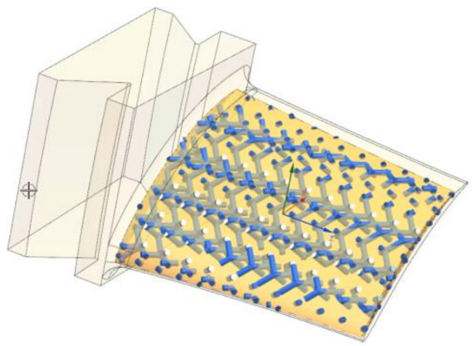
Edge Length: 2 mm

Rod Diameter: 1 mm



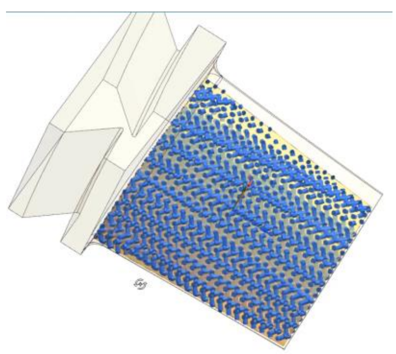
Edge Length: 2 mm

Rod Diameter: 0.5 mm

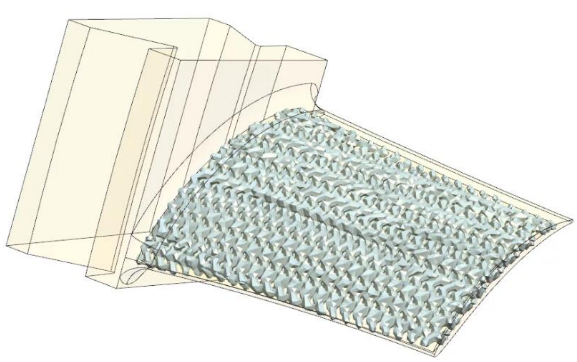


Edge Length: 1 mm

Rod Diameter: 0.5 mm

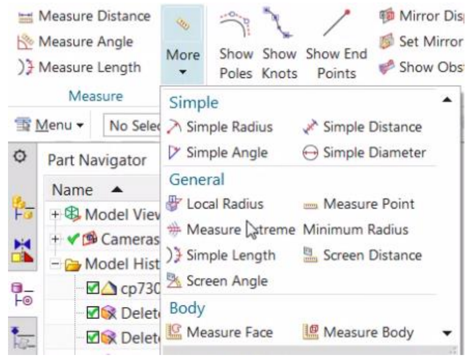


Therefore, with Edge Length of 1 mm and Rod Diameter of 0.5 mm, the lattice structure can be easily formed for the selected body as shown in the figure below using the **Lattice** feature in the NX Design for Additive Manufacturing.

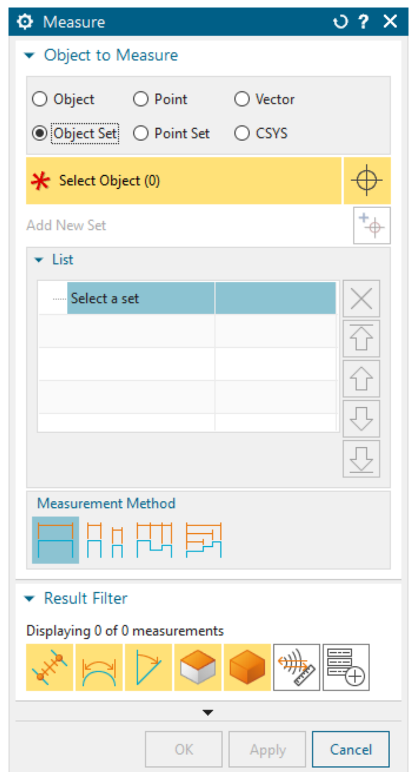


10.2 PROCESS AUTOMATION

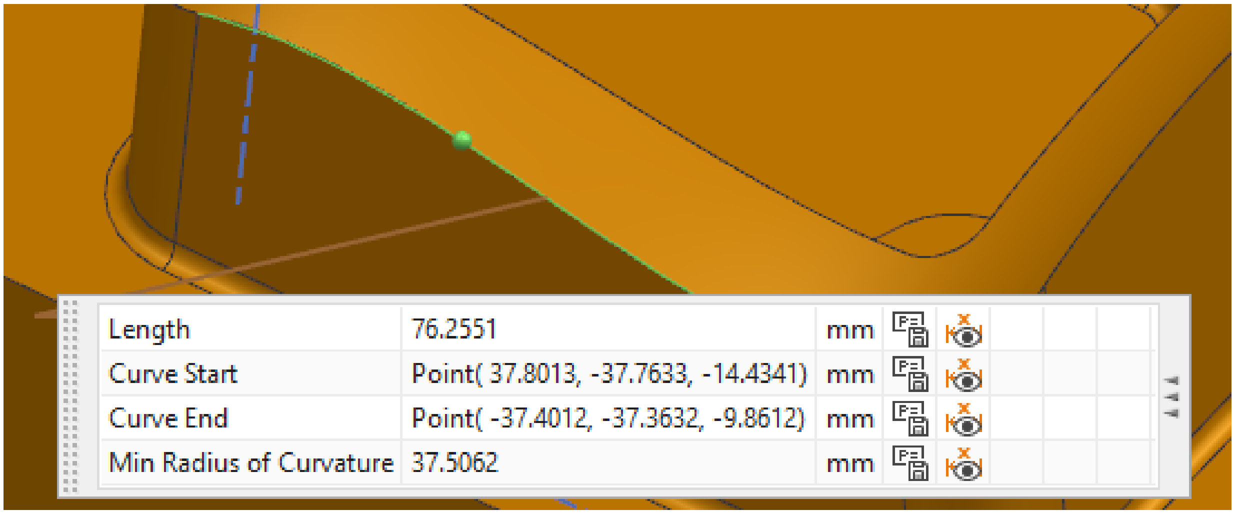
10.2.1 Measurement



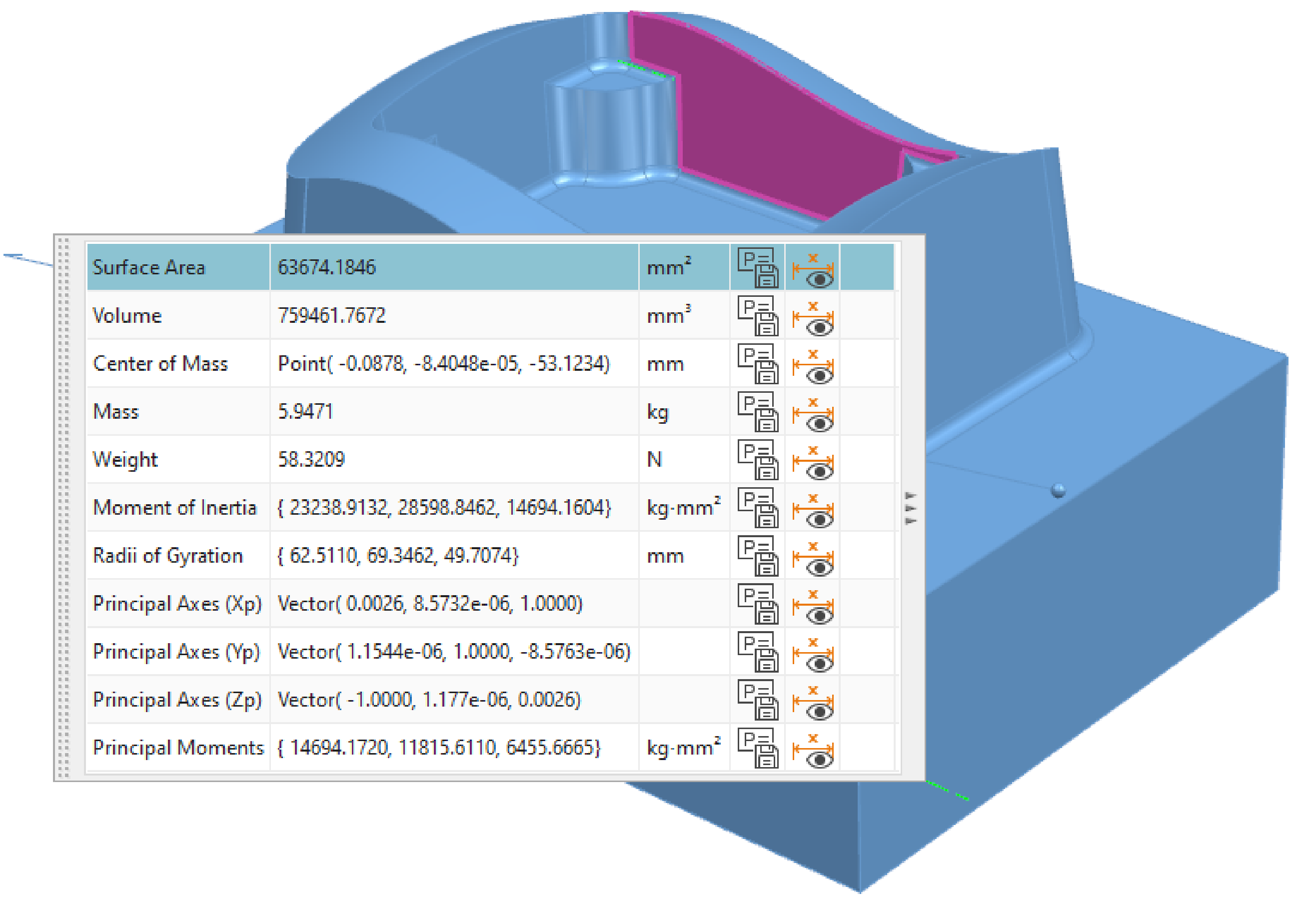
In old versions of NX, depending on the measured objects (face or body) and dimensions (length or angle), there were many commands and options (e.g., 16 menu commands and more than 50 options in one NX version) available for dimensional measurement. You would need to spend a lot of time to select the proper commands and options for measuring your CAD model.



With the latest version of NX, the various dimensional measurement commands have been combined into one single command "**Measure**" under the **Analysis** tab to measure distance, angle, length, body, etc. The [user interface of **Measure**] is shown in the right figure. The measurement result types (distance, curve/edge, and angle) and objects (face and body) can be customized in the **Result Filter** to add or remove. This **Measure** command will allow you to directly select entities (e.g., object, face, curve, point, vector) you want to measure. Then, NX software displays every possible dimension related to your selection. It saves your time in measuring objects and their dimensions. For example, the figure below measures a curve in the CAD model using the latest measurement command **Measure**, which will pop up a window with all possible dimensions (Length, Curve Start Point, Curve End Point, and Min Radius of Curvature). This contrast old versions of NX, where different commands (curvature length, minimum radius, and point measurement commands) were needed to measure these dimensions.

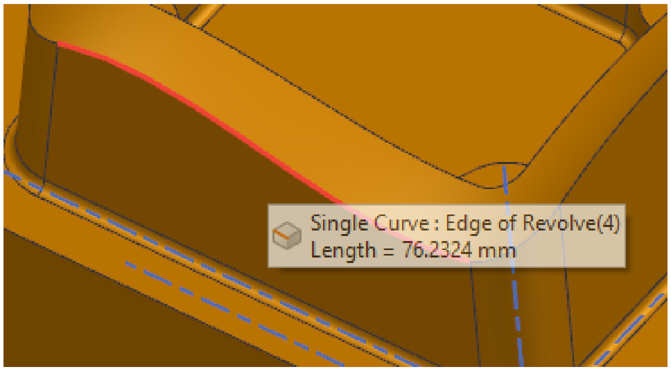


Selecting an object with **Measure** command will display comprehensive dimensional details.

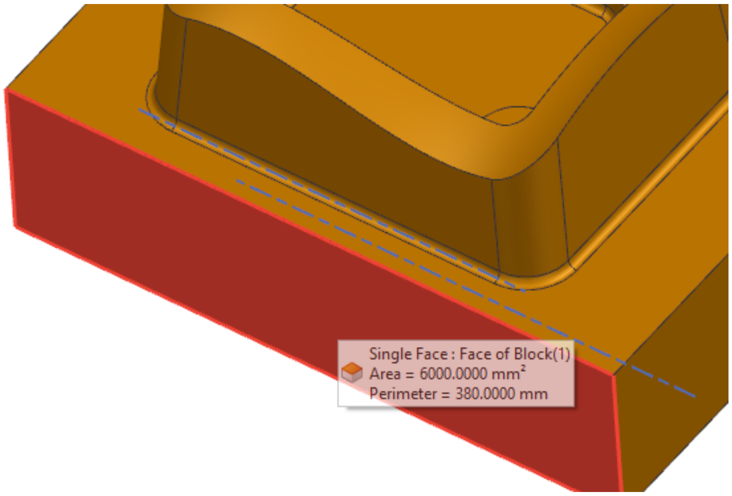


With use of **Measure** command NX will automatically show you the commonly interested dimensions for an object (face or curve) when you move your mouse to the object without clicking. For example, the details of an object are displayed for three different objects blow.

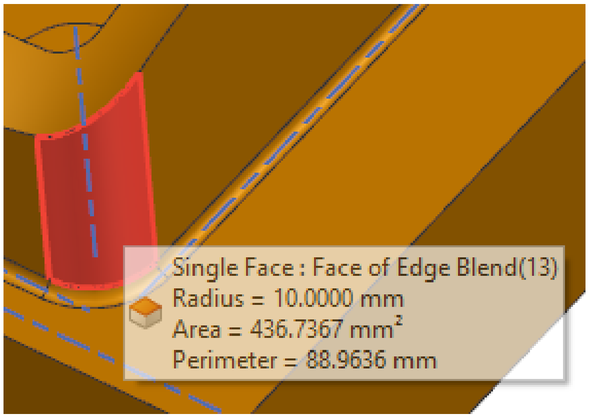
Curve with Length



Face with Area and Perimeter



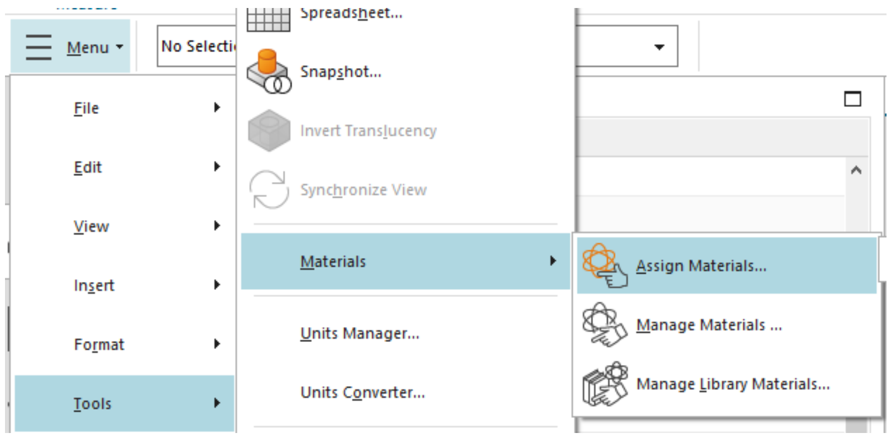
Edge Blend Face with Radius, Area, and Perimeter



10.2.2 Weight Analysis

When designing a 3D model, weight evaluation is often an essential step to help you evaluate whether the material amount satisfies the customer's weight specification. Here we will show you how to easily get the Weight and Mass of your model automatically.

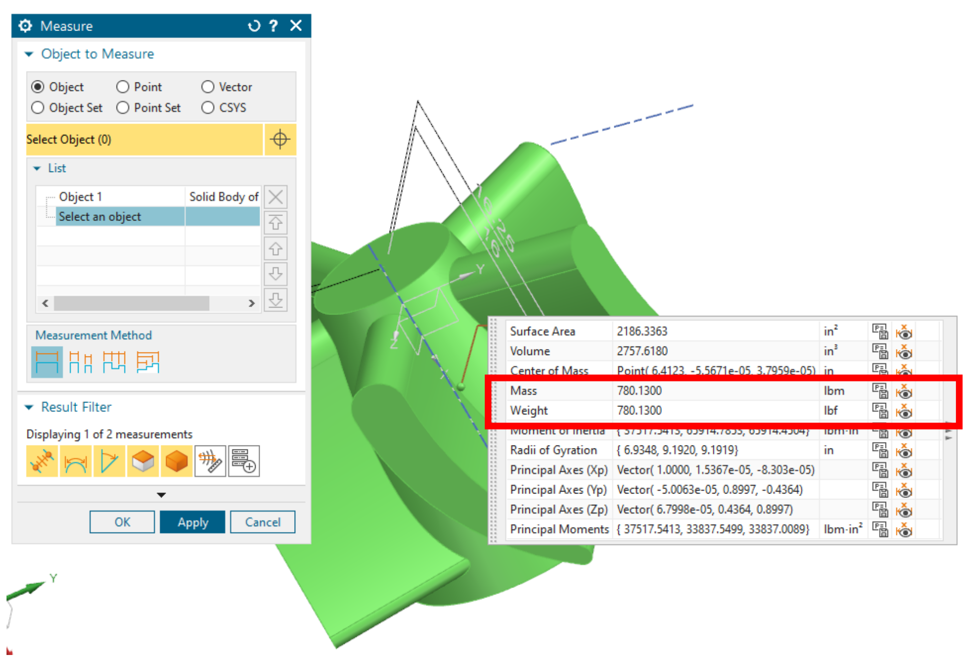
- Open the file "Impeller_impeller" or a file with the part you want to measure
- Assign Material: **Menu → Tools → Materials → Assign Material**



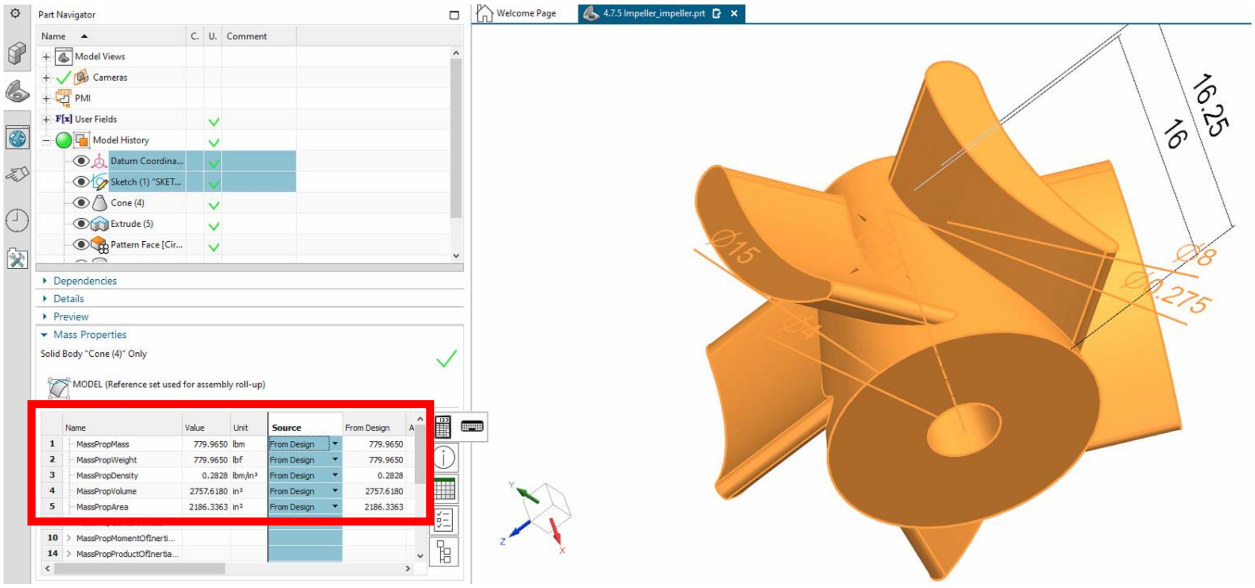
Assign the material you want to use. In this example, Steel is assigned as the material.

There are two different ways you can get the weight value.

1. Use **Measure** Command to get the Mass and Weight of the part



2. Select your model and expand the window of **Mass Properties** under **Part Navigator**

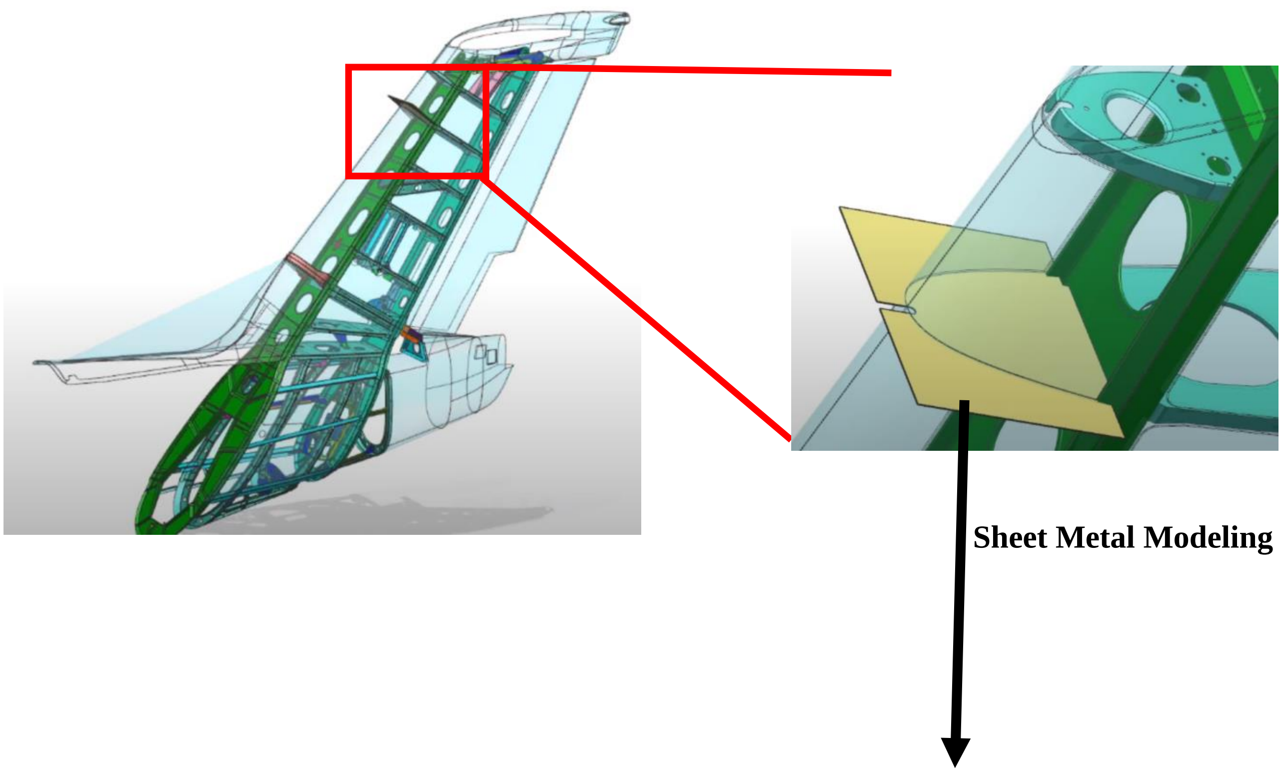


The Mass, Weight, Density, Volume and (surface) Area will be shown in the Mass Properties.

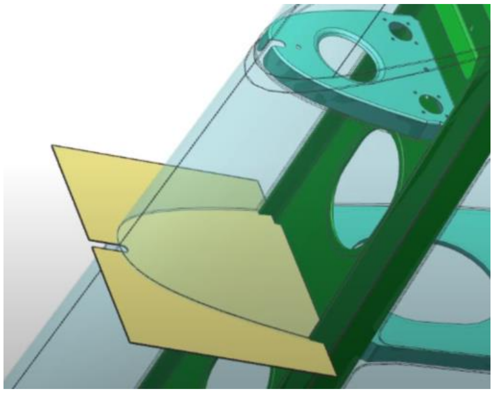
10.3 SHEET METAL

The Sheet Metal modeling of NX is a fast, accurate method to model thin frames and shelling parts for automotive, aerospace and other products. The Sheet Metal feature provides a host of modeling tools in the NX environment, such as bend, unbend, and re-bend a metal sheet. It allows you to create sheet metal parts following the industrial workflow but with better bending calculations, tooling, and sheet metal features compared to part modeling by the extrude feature. Providing an accurate sheet metal pattern not only enhances the quality of part modeling (e.g., thickness uniformity) but also reduces the cost of part manufacture. To have uniform thickness in complex parts, NX Sheet Metal either uses surfacing features and then thickening the surface or starts modeling a solid with blending and then shelling, these two methods are just examples how users can create sheet metal geometries then using the Convert To Sheet Metal.

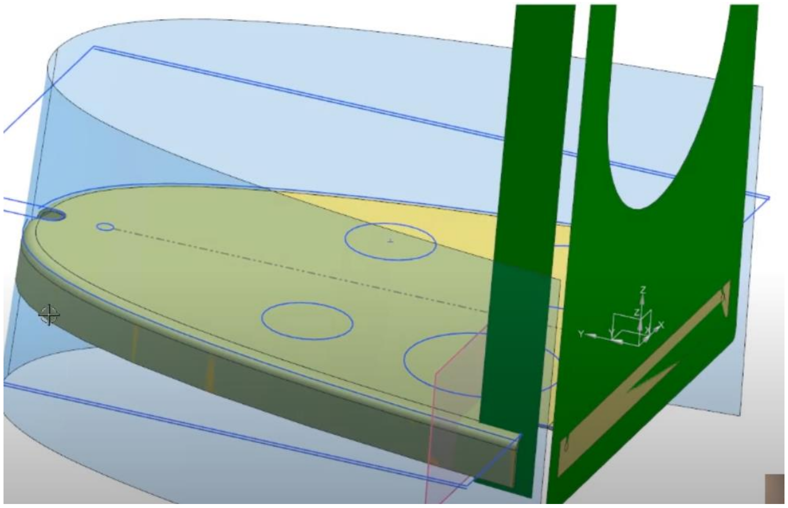
For example, NX Sheet Metal could model a component in the tail section of an aircraft using the following steps.



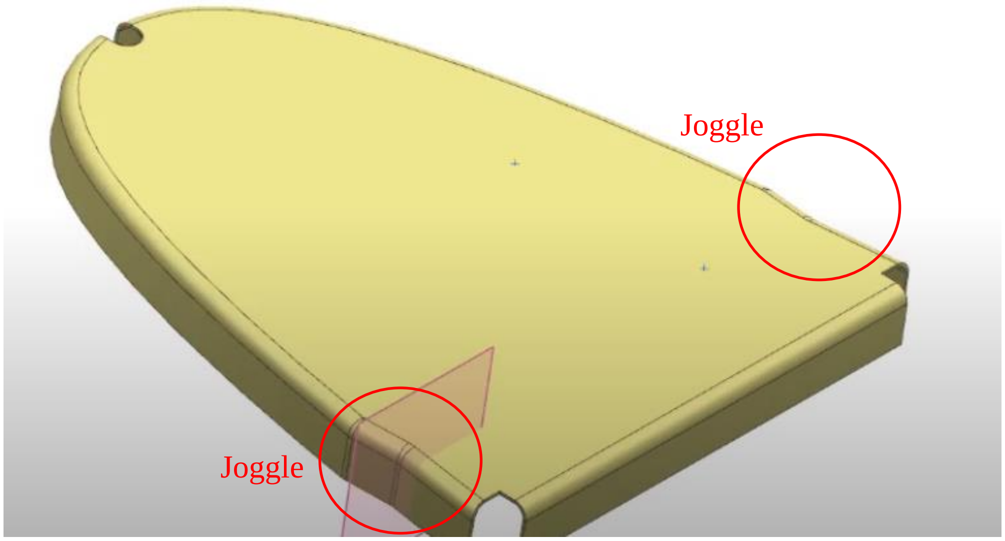
- First, create a flat sheet in the model of aircraft’s tail section in NX



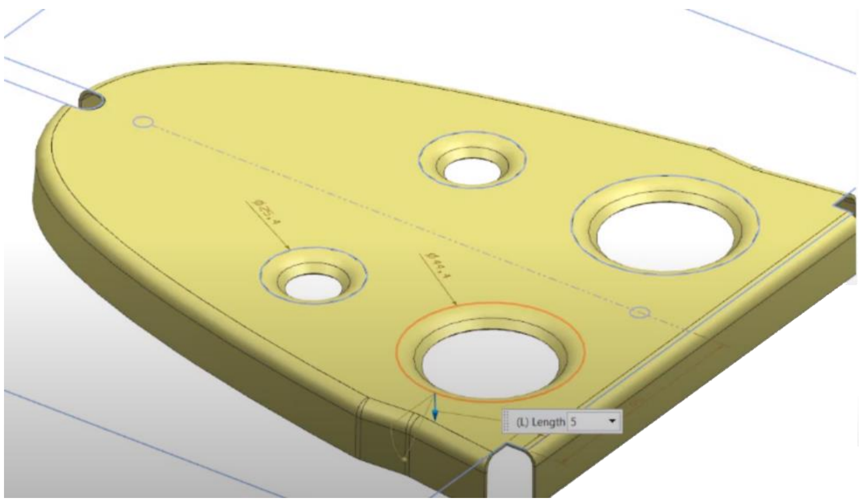
- Use **Advanced Flange** to trim and bend the edges



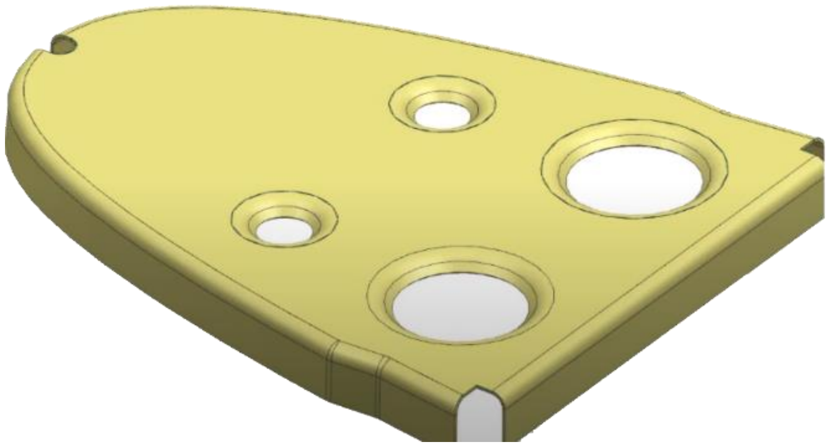
- Use **Joggle** to create joggle features on both side



- Use **Lightening Cutout** to create lightening holes to reduce structural weight while maintain strength and rigidity



Finally, the modeling of this aircraft component using NX Sheet Metal is completed as shown below.



10.4 CONTINUOUS RELEASE

Siemens NX uses a Continuous Release methodology starting from January 2019 to release new versions of NX, which enables the software user to have:

(1) Quicker functional enhancements to increase productivity

(2) Constant updating to get better adaptation for the utilization of latest technologies

(3) Efficient response from NX development team

(4) Lower deployment cost

(5) Faster response to new ideas and trends in technologies

(6) Continued full NX-Simcenter 3D integration and interoperability

Before using the Continuous Release methodology, typically, it would take several years to deploy PLM-specific software updates on an average, which means the NX users were not able to benefit from software enhancements and fixes rapidly. With Continuous Release the users are able to realize Siemens Digital Industry Software's latest advancements in the production environment without any delay. The Users in the Continuous Release mode will see monthly update releases, with major enhancement releases approximately every 6 months. The monthly updates will be for fixes and a small number of high-priority items. Continuous Release will incorporate new development practices that will remove the need to upgrade currently supported versions of Teamcenter and Active Workspace when updating NX. Customers will still have the option to use their existing Siemens Support Center (used to be known as GTAC) process of downloading updates and enhancements to NX, NX CAM, and Simcenter 3D. As soon as an update is released under this new Continuous Release process, it will be available on Support Center for the user to download

Siemens will continue to improve the upgrade process through quality product improvements with a focus on protecting user's data, and ease the upgrading cycle, so the end users can stay ahead of competition and innovate faster.