NX 9.0 for Engineering Design

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FOREWORD

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

The NX software integrates knowledge-based principles, industrial design, geometric modeling, advanced 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 free-form shapes such as airfoils and manifolds. It also merges solid and surface modeling techniques into one powerful tool set.

This self-guiding tutorial provides a step-by-step approach for users to learn NX9.0. 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 NX9.0 session to creating models and designs that have various applications. Each chapter has components explained with the help of various dialog boxes and screen images. These components are later used in the assembly modeling, machining and finite element analysis. These models of components are available online to download and use. We first released the tutorial for Unigraphics 18 and later updated for NX2 followed by the updates for NX3, NX5 and NX7. This write-up further updates to NX9.0.

Our previous efforts to prepare the NX self-guiding 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 Albin Thomas at atqk3@mst.edu. The models and all the versions of the tutorial are available at http://web.mst.edu/~mleu/.

CHAPTER 1 - INTRODUCTION

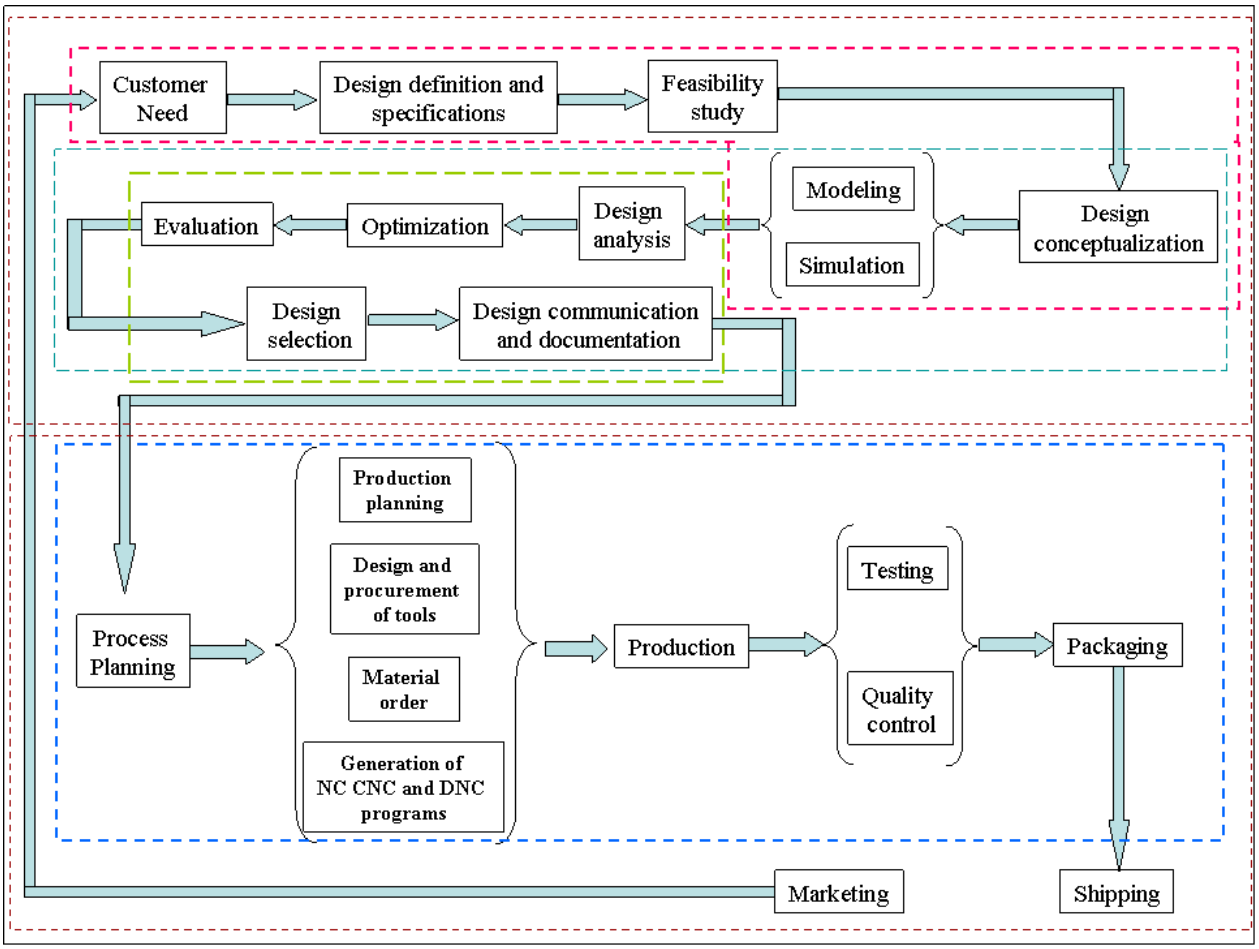
The modern manufacturing environment can be characterized by the paradigm of delivering products of increasing variety, smaller batches and higher quality in the context of increasing global competition. Industries cannot survive worldwide competition unless they introduce new products with better quality, 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 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 1800's. 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 1950's 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 1950's 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 1970's, 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 1970's 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 1980's, new theories and algorithms evolved and integration of various elements of design and manufacturing was developed. The major research and development focus was 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.

- AutoCAD and Mechanical Desktop are some low-end CAD software systems, which are mainly used for 2D modeling and drawing.

- NX, Pro-E, CATIA and I-DEAS are high-end modeling and designing software systems that are costlier but more powerful. These software systems also have computer aided manufacturing and engineering analysis capabilities.

- ANSYS, ABAQUS, NASTRAN, Fluent and CFX are packages mainly used for analysis of structures and fluids. Different software are used for different proposes. For example, Fluent is used for fluids and ANSYS is used for structures.

- Geomagic and CollabCAD are some of the latest CAD systems that focus on collaborative design, enabling multiple users of the software to collaborate on computer-aided design over the Internet.

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, 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 NX9.0 for designing mechanical components and assemblies. Learning to use this software will also be valuable for learning how to use other CAD systems such as PRO-E and CATIA.

This tutorial provides a step-by-step approach for learning NX9.0. The topics include Getting Started with NX9.0, Form Features, Feature Operations, Drafting, Sketching, Free Form Features, Assembly Modeling, and Manufacturing.

Chapter 1 gives the overview of CAD/CAM/CAE. The product realization cycle is discussed along with the history of CAD/CAM/CAE and the definitions of each.

Chapter 2 includes the NX9.0 essentials from starting a session with Windows to getting familiar with the NX9.0 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.

The actual designing and modeling of parts begins with **chapter 3**. It describes different features such as reference features, swept features and primitive features and how these features are used to create designs.

Chapter 4 is a continuation of chapter 3 where various kinds of feature operations are performed on features. The different kinds of operations include Trim, Blend, Boolean operations and many more.

You will learn how to create a drawing from a part model in **chapter 5**. In this chapter, we demonstrate 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 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 7 introduces free-form modeling. The method of modeling curves and smooth surfaces will be demonstrated.

Chapter 8 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 9 is capsulated into a brief introduction to Structures Module available in NX9.0 for the Finite Element Modeling and Analysis.

Chapter 10 will be 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 Codes) to produce the designed parts from Vertical Machining Centers.

The examples and exercise problems used in each chapter are so designed that they will be finally assembled in the chapter. Due to this distinctive feature, you should save all the models that you have generated in each chapter.

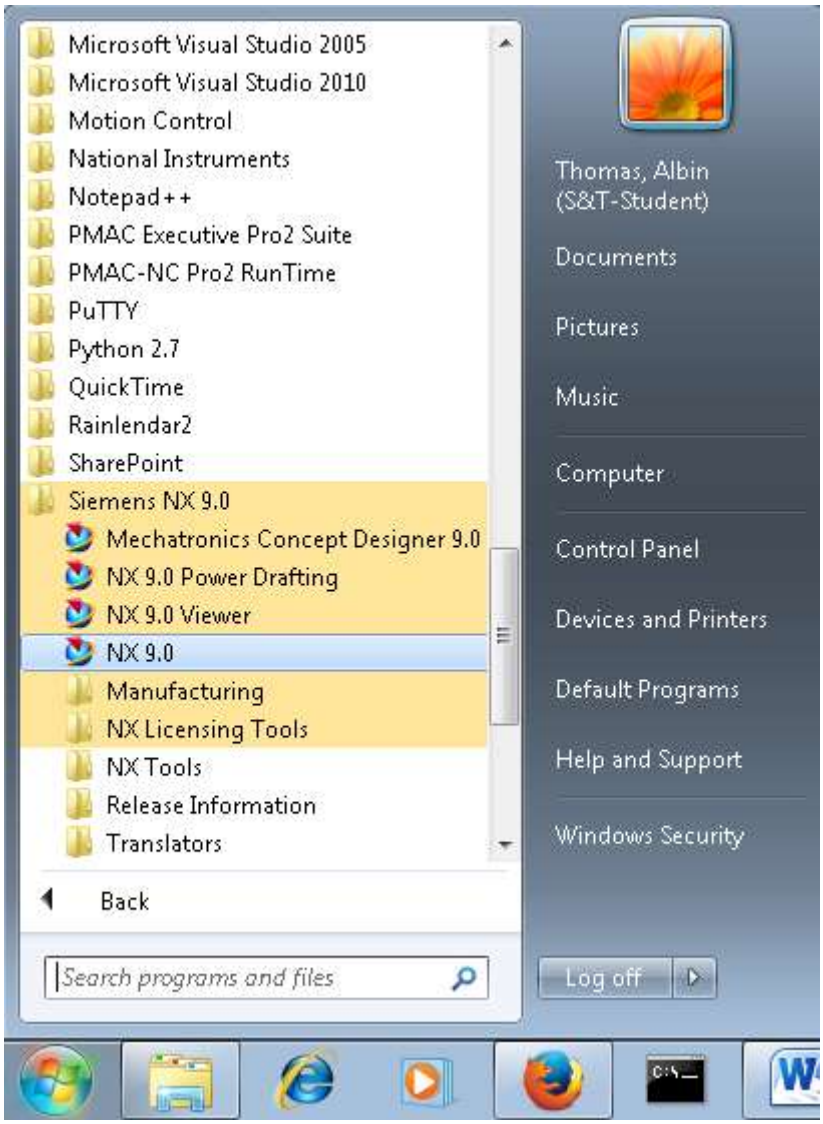
CHAPTER 2 - GETTING STARTED

We begin with starting of an NX9.0 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 NX9.0 package for modeling, drafting, etc. It contains five sub-sections a) Opening an NX9.0 session, b) Printing, saving, and closing part files, c) getting acquainted with the NX9.0 user interface d) Using layers and e) Understanding important commands & dialogs.

2.1 STARTING NX9.0 SESSION AND OPENING FILES

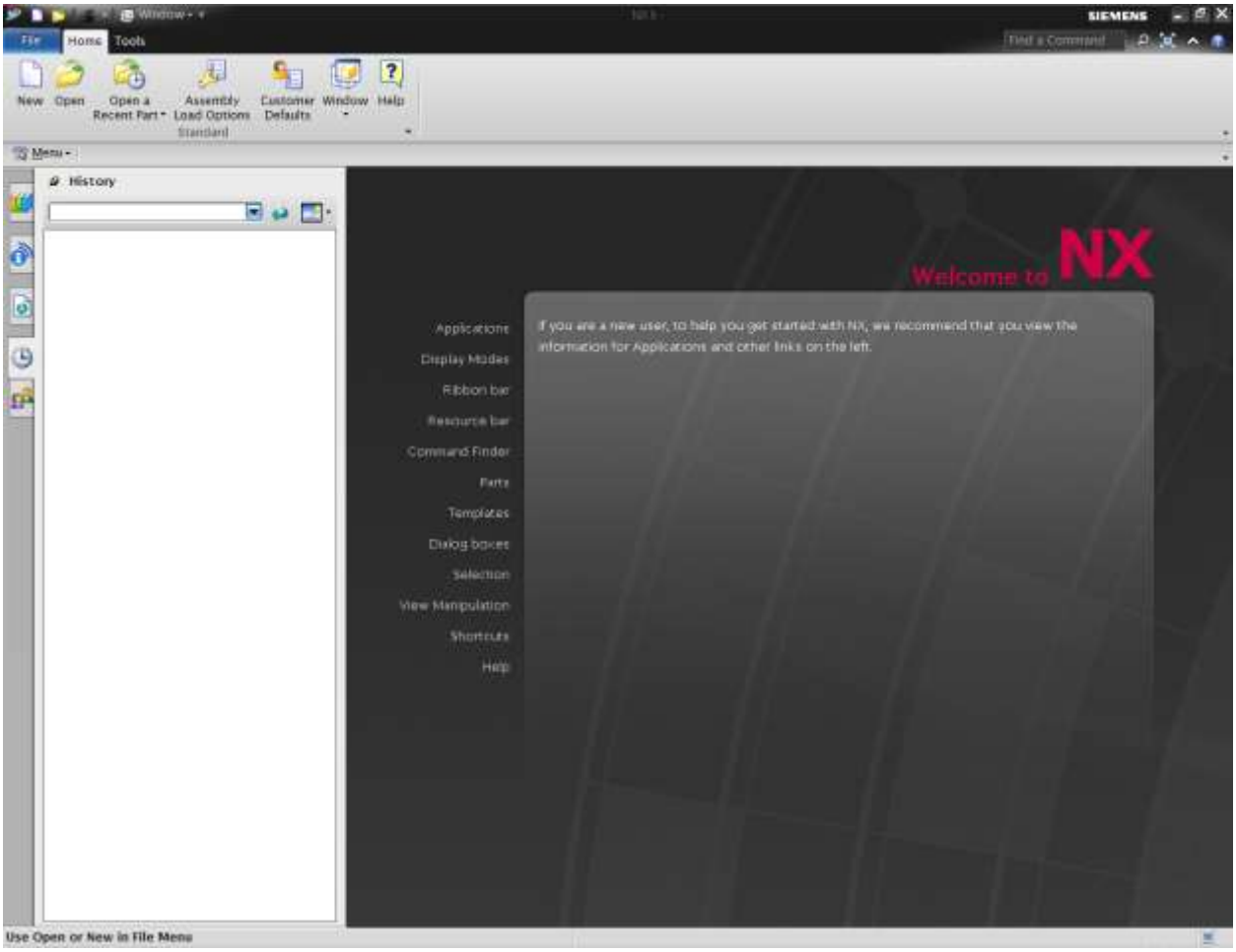
2.1.1 Open NX9.0 Session

- From the Windows desktop screen, click on **Start → All Programs → Siemens NX 9.0 → NX 9.0**



The main NX9.0 Screen will open. This is the Gateway for the NX9.0 software. The NX9.0 blank screen looks like the figure shown below. There will be different tips displayed on the screen about the special features of the current version. The Gateway also has a **Ribbon Bar** that will allow you to create a *new* file or *open* an existing file (under home tab). On the left side of the Gateway screen, there is a Toolbar called as **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. Let's begin by learning how to open a part file in NX9.0.

To create a new file there are two options. You can click on the '**New**' command on top of the screen or go through the '**File**' drop-down tab.



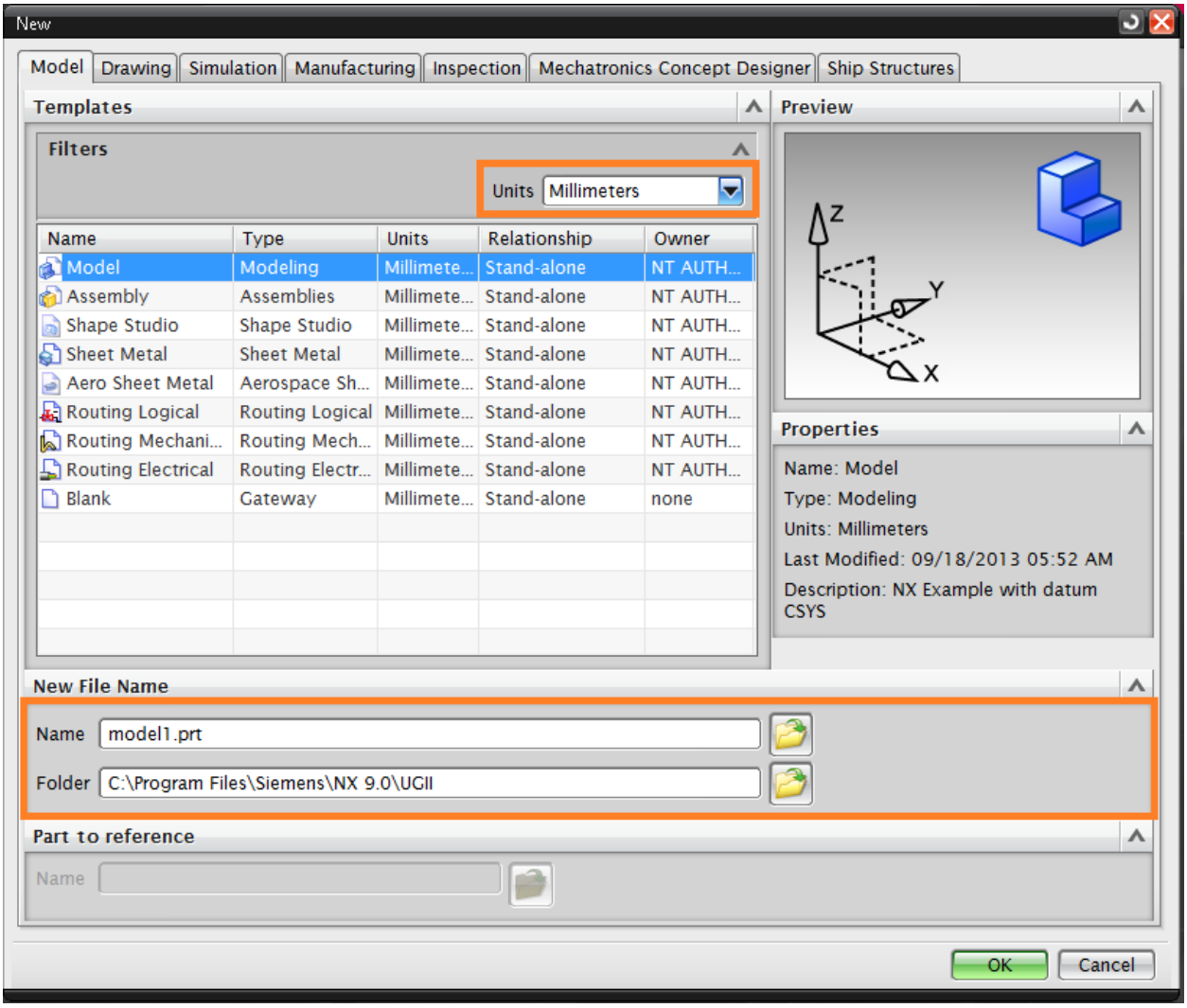
2.1.2 Open a New File

- On the Ribbon Bar, under the **file** tab found at the top-left of the screen, click **FILE → NEW**

This will open a new session, asking for the name and location of the new file to be created as shown at the bottom left. You need to select the units (inches or millimeters) of the working

environment by clicking on the drop-down menu on the top right corner. The default is ***millimeters***. However, most of the material in the tutorials is modeled in ***inches***. So always, be sure to select ***inches*** before creating a new .**prt** file unless otherwise specified. You can also select the type of the file you want to create – either a part file or an assembly file or sheet-metal file – by selecting the file type as shown in ***Templates*** dialogue box located at the center of the window. The properties of the selected file are displayed below the ***Preview*** on the middle right corner.

- Enter the location of the file and then and click **OK**



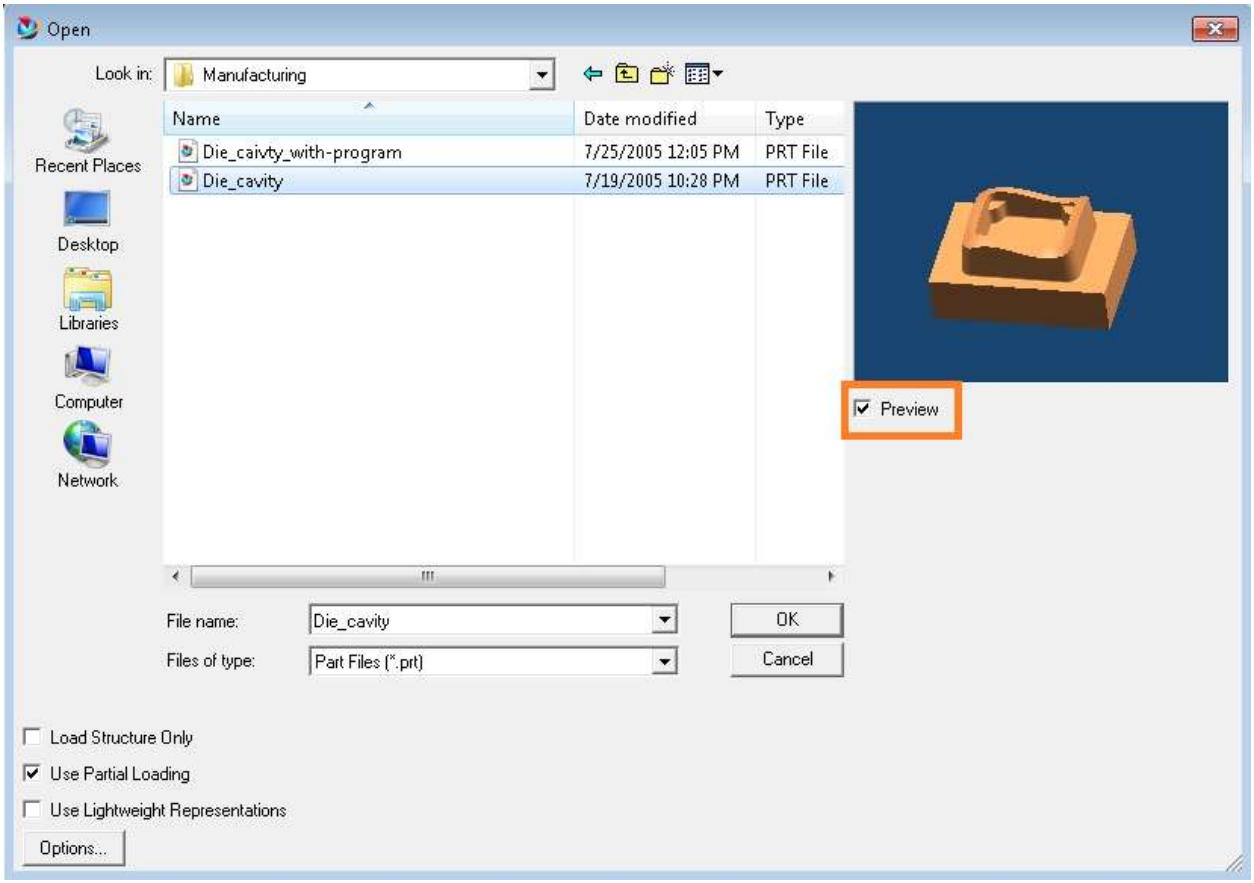
2.1.3 Open a Part File

Click **FILE → OPEN**

You can also click the ***Open*** icon from the Home tab at the top of the screen.

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



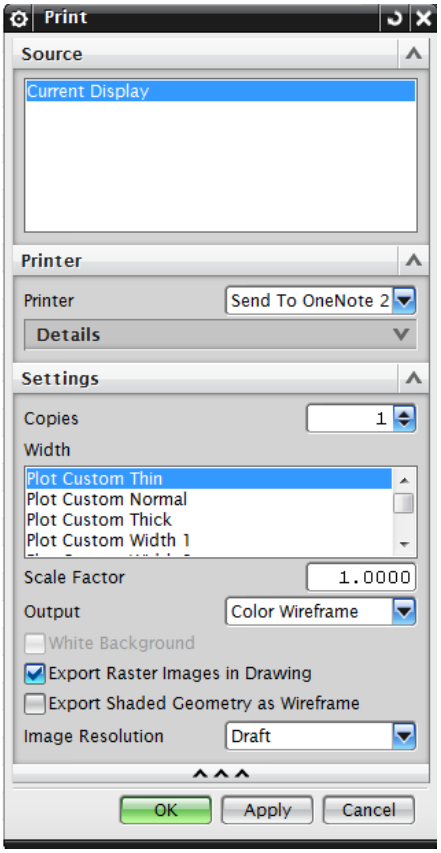
2.2 PRINTING, SAVING AND CLOSING PART FILES

2.2.1 Print a NX9.0 Image

- 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



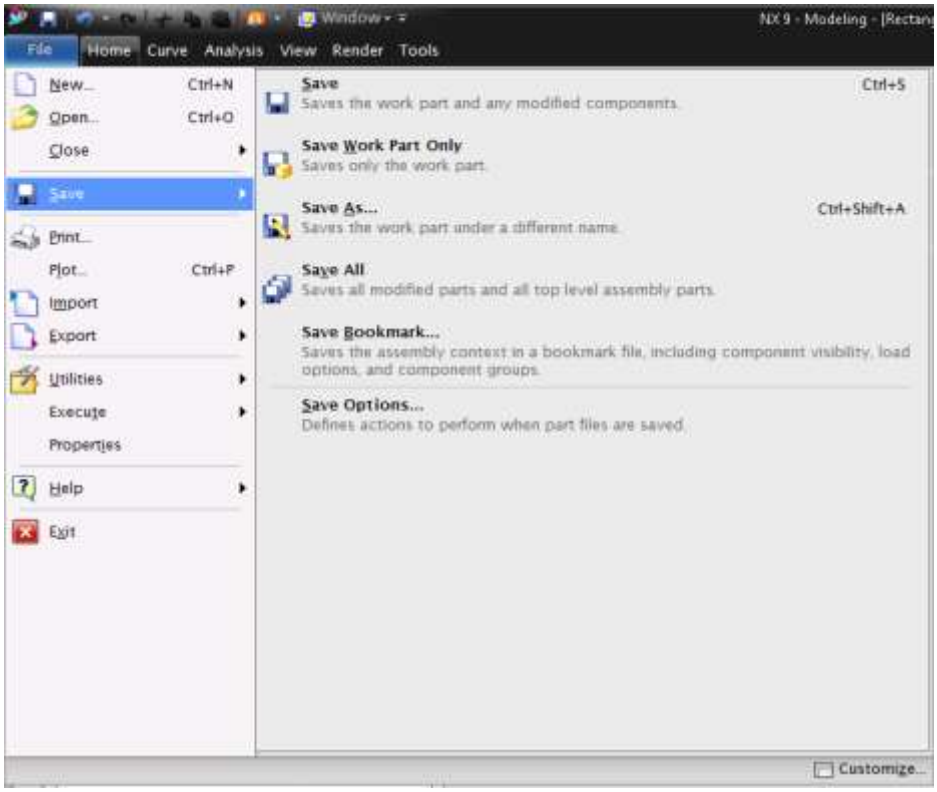
2.2.2 Save Part Files

It is imperative that you save your work **very frequently**. If for some reasons, NX9.0 shuts down and the part is not saved, all the work will be lost. To save the part files

➢ Click **FILE**

On the File drop-down tab hover over the save option , there are five different 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:*** option will only save the active part on the screen.
- ***SAVE AS:*** option allows you to save the part on screen using a different name.



- ***SAVE ALL:*** This option will save all the opened part files with their existing names.
- ***SAVE BOOKMARK:*** This option will save a screenshot of the current model on the screen as a .JPEG file and bookmarks.

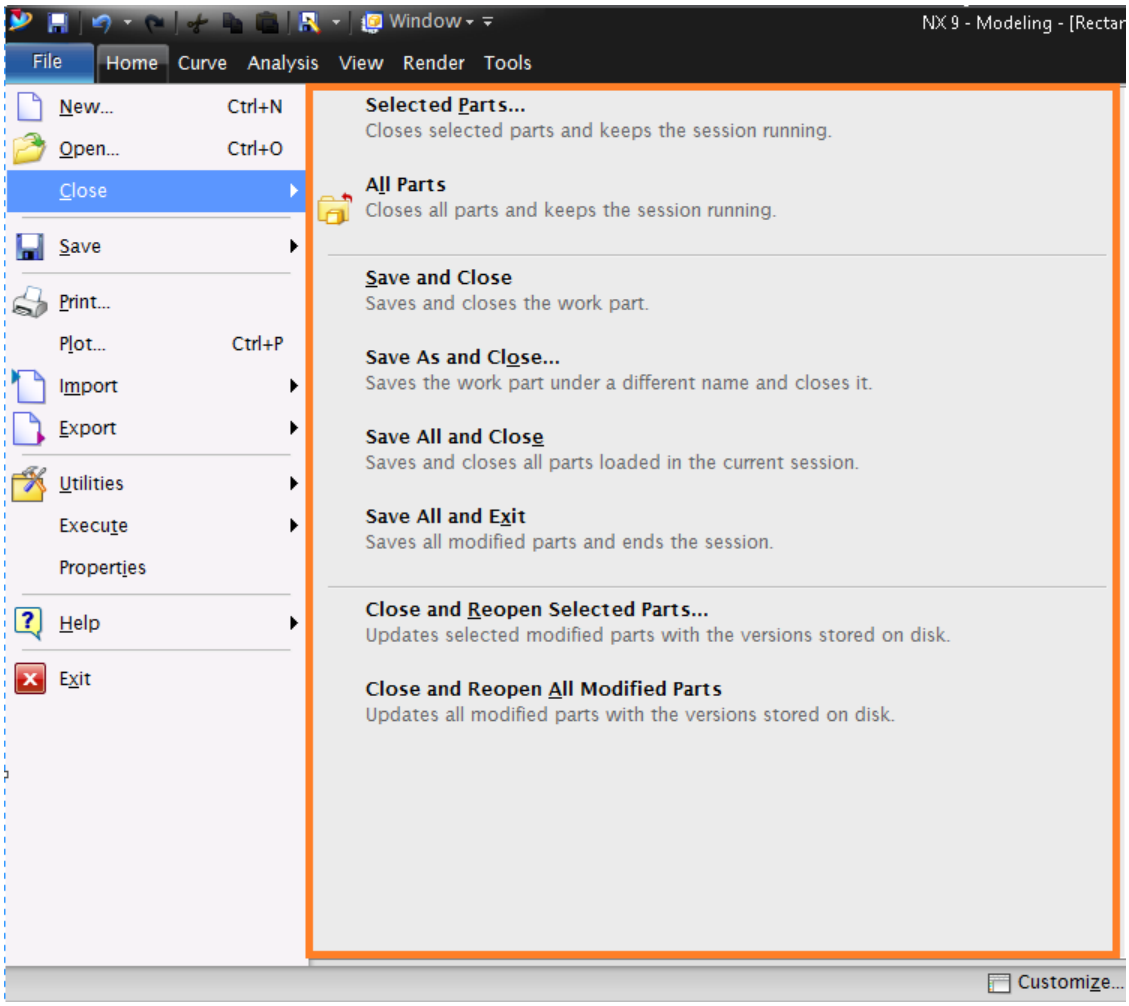
Remember as in previous versions all the parts are saved with a ***.prt*** extension in NX9.0.

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*** or ***SAVE ALL AND CLOSE*** or ***SAVE ALL AND EXIT***.



In case of the first two options, the parts that are ***selected*** or the ***all parts*** the files will be closed but the NX9.0 session keeps on running.

2.2.4 Exit an NX9.0 Session

- Click **FILE → EXIT**

- Since we are not ready to exit NX9.0, click **NO**

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

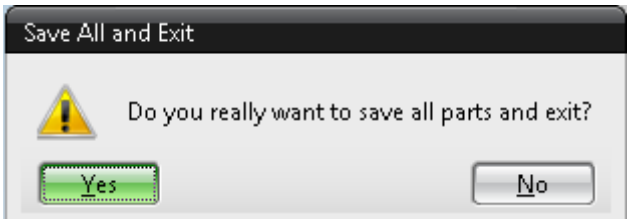
- Select **NO**, save the files and then **Exit**

2.2.5 Simultaneously Saving All Parts and Exiting

A second way to exit NX9.0 session at the same time save all the files and exit the program is

- Click **FILE → CLOSE → SAVE ALL and EXIT**

The Save and Exit warning dialog window is shown below.



- Choose **NO** or **CANCEL**

2.3 NX9.0 INTERFACE

The user interface of NX9.0 is made very simple through the use of different icons. 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 for entering values and naming files.

2.3.1 Mouse Functionality

It is highly recommended to use a three-button mouse or a scroll-mouse while working with NX9.0. The power of mouse buttons and their primary functions are discussed below.

2.3.1.1 Left Mouse Button (MB1):

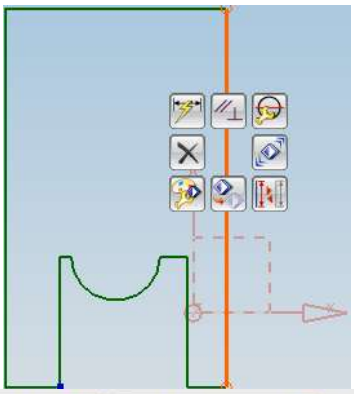
The MB1 or left mouse button is used for ***Selection*** of icons, menus, and other entities on the graphic screen. Double clicking MB1 on any feature will automatically open the ***Edit*** Dialog box.

2.3.1.2 Middle Mouse Button (MB2):

The MB2 or middle mouse button or the scroll button is used to ***Rotate*** the object by pressing, holding and dragging. It can be used for ***Pan*** and ***Zoom*** options in combination with other mouse buttons or key buttons. If it is a scroll button, the object can be zoomed in and out by scrolling. Just clicking the MB2 will execute the **OK** command if any pop-up window or dialog box is open.

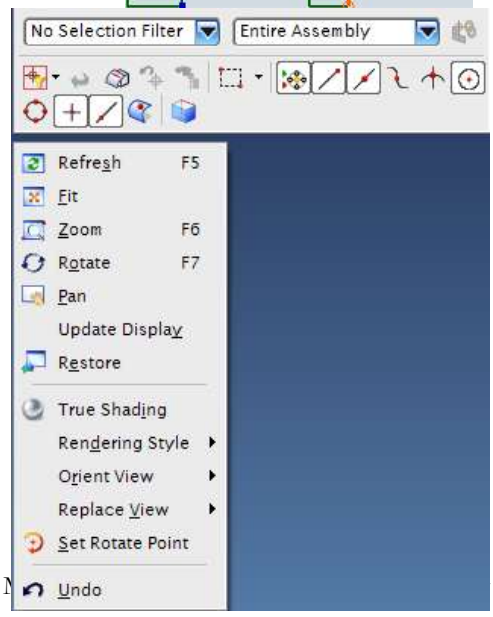
2.3.1.3 Right Mouse Button (MB3):

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 figures shown on the right are 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.

Clicking MB3 on graphics screen will pop up the ***View*** menu options as shown below.



*** Note: The functionality of the mouse buttons depends on the *Application* used. For instance, the**

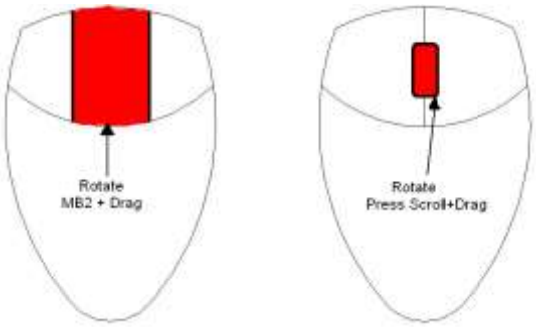
menus that pop-up in *Modeling* are different from those in *Simulation.*

2.3.1.4 Mouse Functionality

The following is the illustration of the mouse buttons used for rotating, panning and zooming in or out on the graphic screen. Besides using these different combinations of mouse buttons, the following commands can also be performed by icons in the Toolbar.

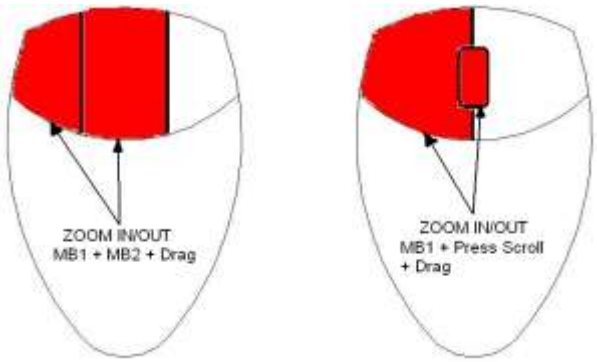
Rotate:

- Press and hold the middle mouse button (or scroll button) and drag around the screen to view the model in the direction you want. 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.



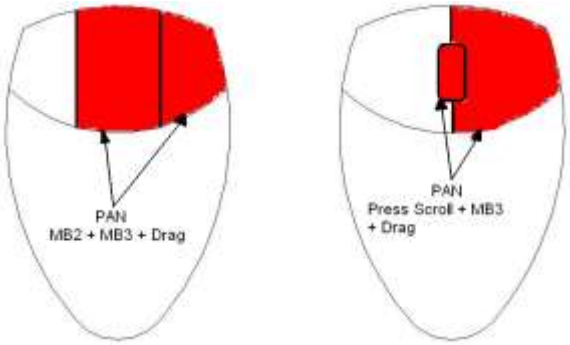
Zoom In /Out:

- Press and hold both the left mouse button and middle button (or scroll button) simultaneously and drag OR
- Press and hold <Ctrl> button on the keyboard and then press and drag the middle mouse button. OR
- Scroll up and down if the mouse has a scroll wheel.



Pan:

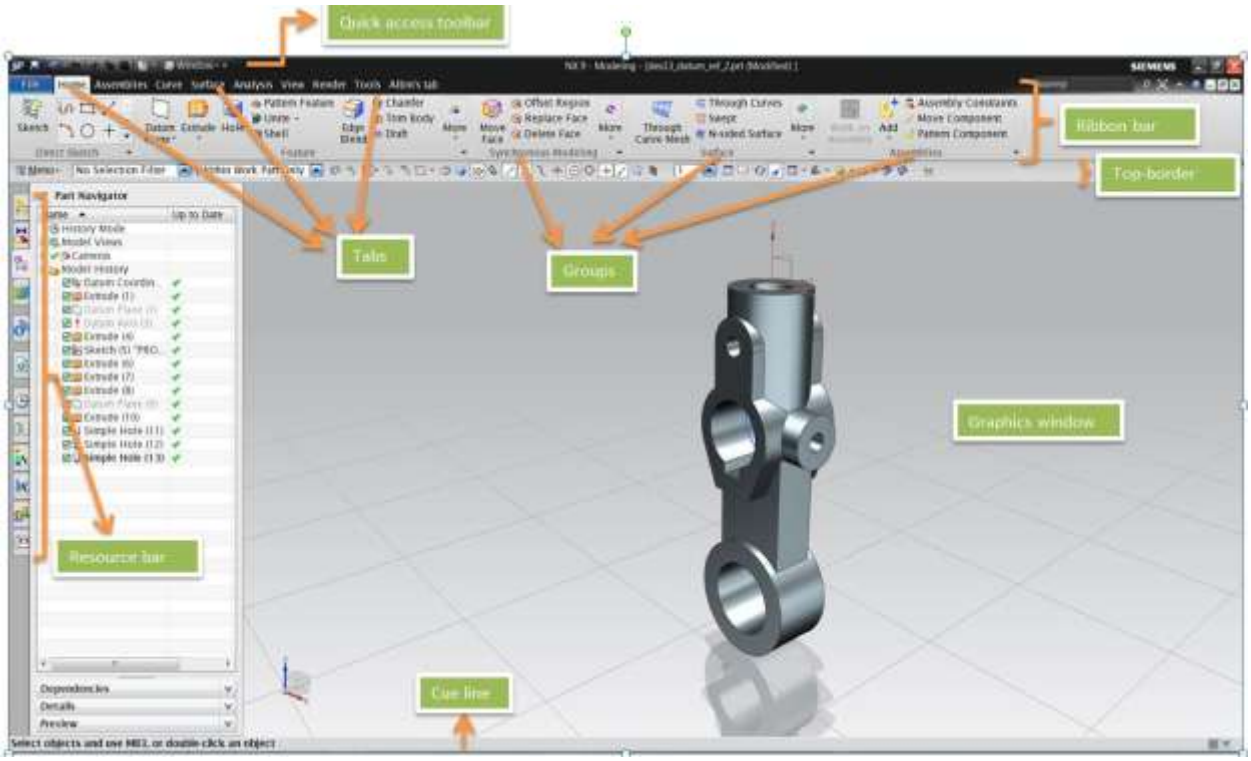
- Press and hold both the middle button and right mouse button simultaneously and drag OR
- Press and hold <Shift> button on the keyboard and press and drag the middle mouse button.



2.3.2 NX9.0 Gateway

The following figure shows the typical layout of the NX9.0 window when a file is opened. This is the Gateway of NX9.0 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.

Shown below is the NX9.0 program in the modeling application.



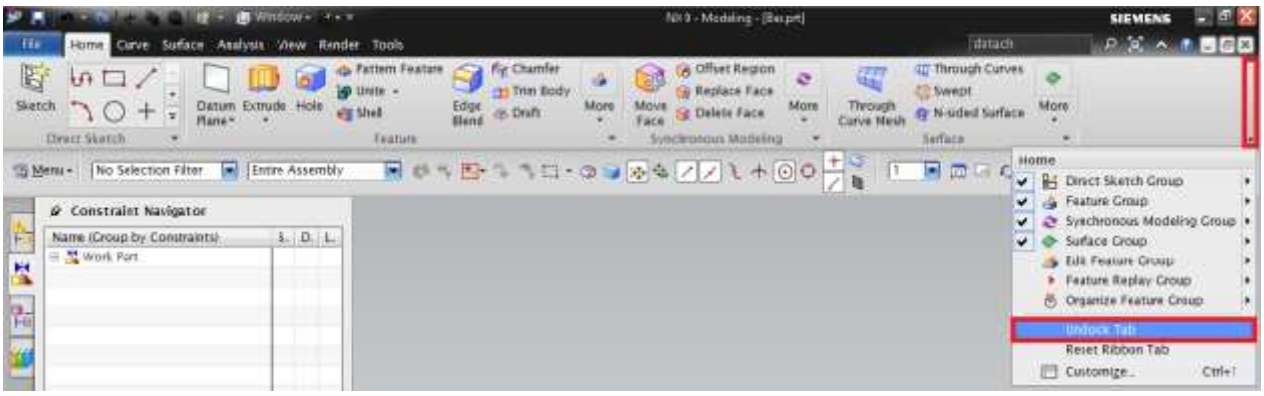
2.3.2.1 Functions of Gateway Zones

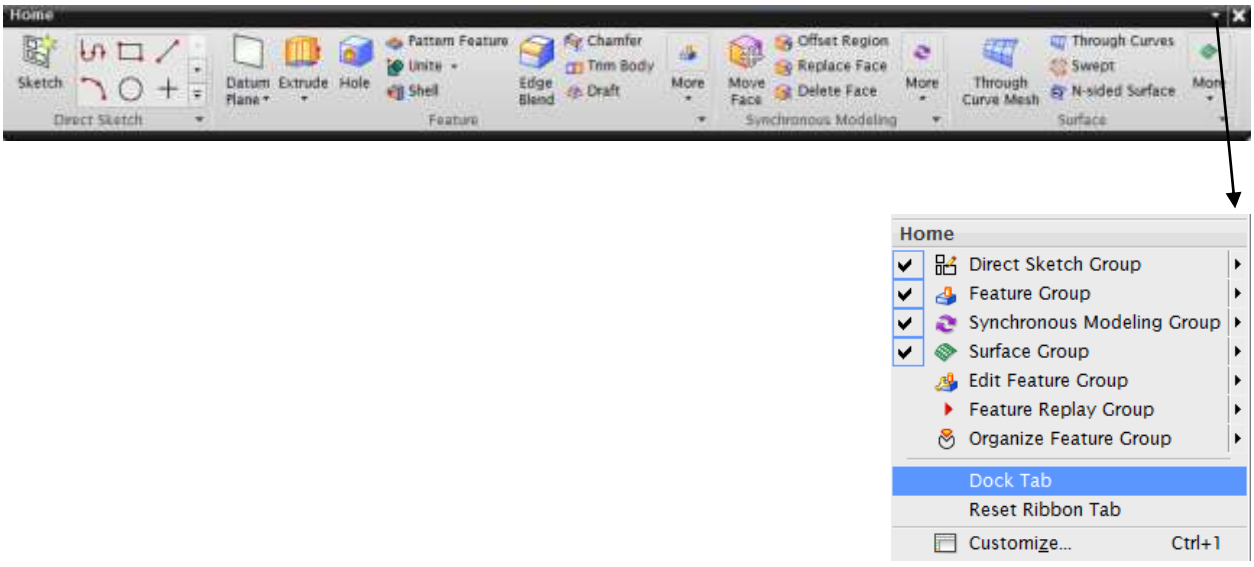
Ribbon Bar:

The ribbon bar interface gives the user with 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, surface tabs. In the home tab, we have direct sketch, feature, synchronous modeling and surface groups. And in each group, we have the set of featured commands.

Unlike previous versions of NX, this is a new feature developed in NX 9.0 with a view for touchscreen systems.





For docking back on to the main toolbar, click the down-arrow on the undocked tab, located towards the top right of the tab and click dock tab.

Selection Bar:

The ***Selection Bar*** is located below the active ***Toolbars*** and displays the selection options. These options include the ***Filters, Components/Assembly,*** and ***Snap Points*** for selecting features.

Resource Bar:

The ***Resource Bar*** features icons for a number of pages in one place using very little user interface space. NX9.0 places all navigator windows in the ***Resource Bar***, as well as the History Palette, Assembly navigator, Part navigator, Animation navigator, Simulation navigator, Roles and the Web Browser. By Default, the ***Resource Bar*** is located on the left side of the NX9.0 window. You can dock and undock the resource bars by clicking on the pin icon on the top left of the resource window.

- UNDOCKED - DOCKED

Cue Line:

The ***Cue Line*** is shown at the bottom of the main NX window below all the Toolbars. The ***Cue Line*** displays prompt messages that indicate the next action that needs to be taken.

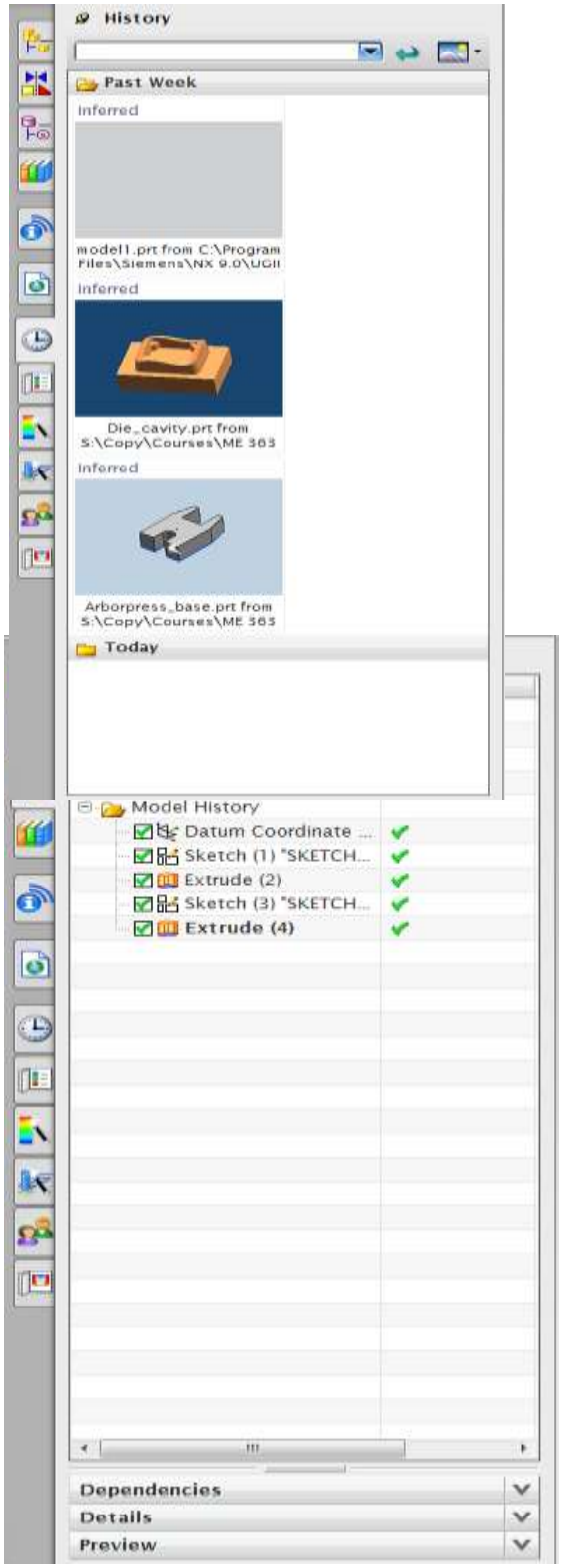
Status Line:

The ***Status Line***, located to the right of the Cue area, displays information messages about the current options or the most recently completed function.

Progress Meter:

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.

2.3.2.2 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 un-suppress 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.

2.3.2.3 History

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

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 to the Graphics Window. To reload a part, click on a saved session bookmark.

2.3.3 Geometry Selection

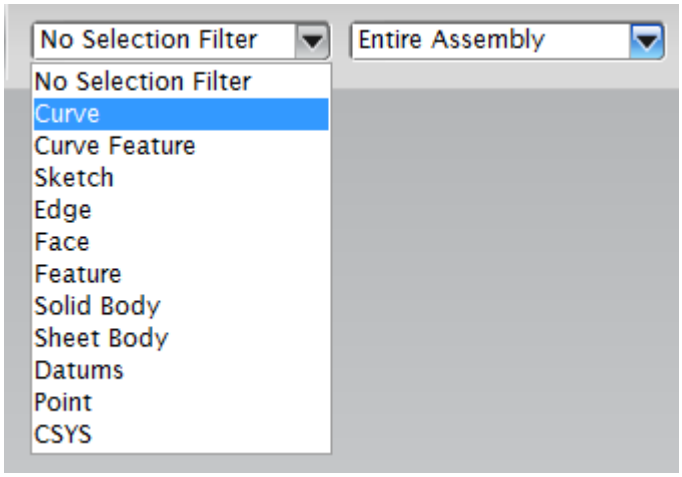
Geometry Selection properties are very advanced in NX9.0. You can filter the selection method, which facilitates easy selection of the geometry in a close cluster. In addition, you can perform any of the feature operation options that NX9.0 intelligently provides depending on the selected entity.

The Mouse cursor in the Graphics screen will normally be in the shape of a cross hair as shown in the figure. 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.

Feature Selection:

Clicking on any of the icons in the figure below will let 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.

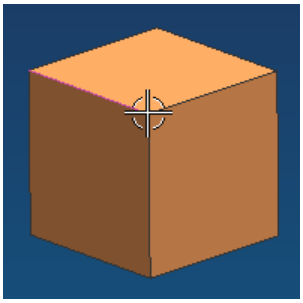
Besides that, the filtering of the features can be further narrowed down by selecting one of the desired options in the drop-down menu as shown in the figure below. For example, selecting ***CURVE*** from the option will highlight only the curves in the screen. The default is ***NO SELECTION FILTER***.



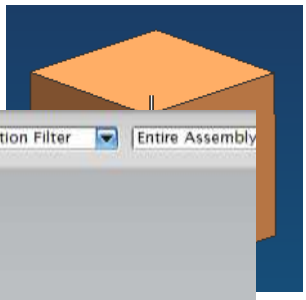
General Object Selection:

Clicking on the icon as shown in the figure below will let you select the general object entities displayed on the screen.

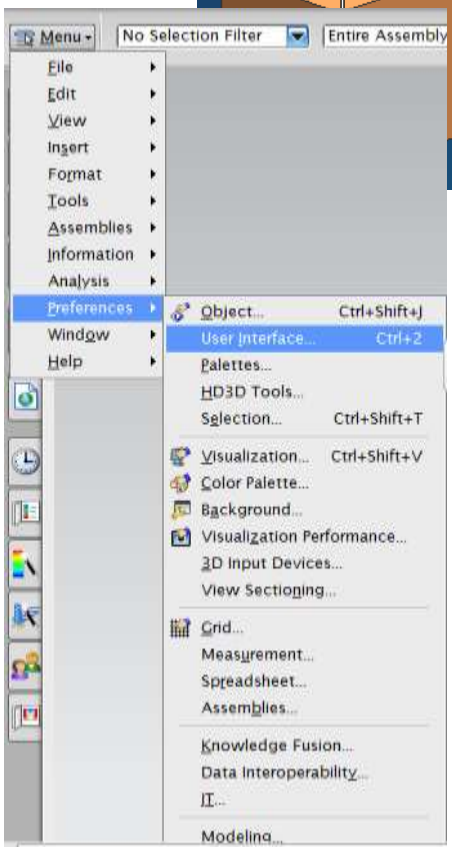
If you want to select any geometric entity, feature, or component, then navigate the mouse cursor closer to the entity until it is highlighted with a ***magenta (pink)*** color and click the left mouse button.

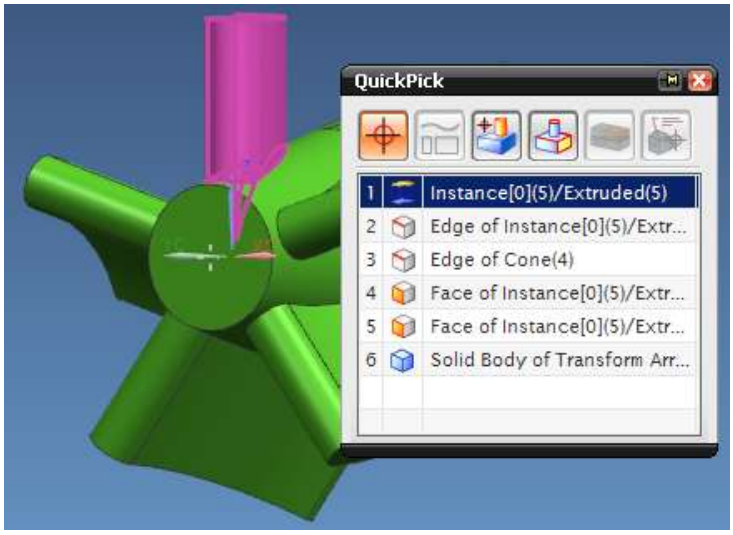


If you want to select an entity that is hidden behind the displayed geometry, then place the mouse cursor roughly close to that area on the screen such that the cursor ball occupies a portion of the hidden geometry projected on the screen. After a couple of seconds, the ball cursor turns into a **'*plus*'** symbol as shown in the figure.



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, NX9.0 will highlight the corresponding entity on the screen in a ***magenta*** color. For example, in the figure below, the face on the top is assigned the number '5'. Likewise, the hidden entities will also be allotted with a number in the list. You can browse through the numbers and click on the number that corresponds to the desired object or feature.





2.3.4 User Preferences

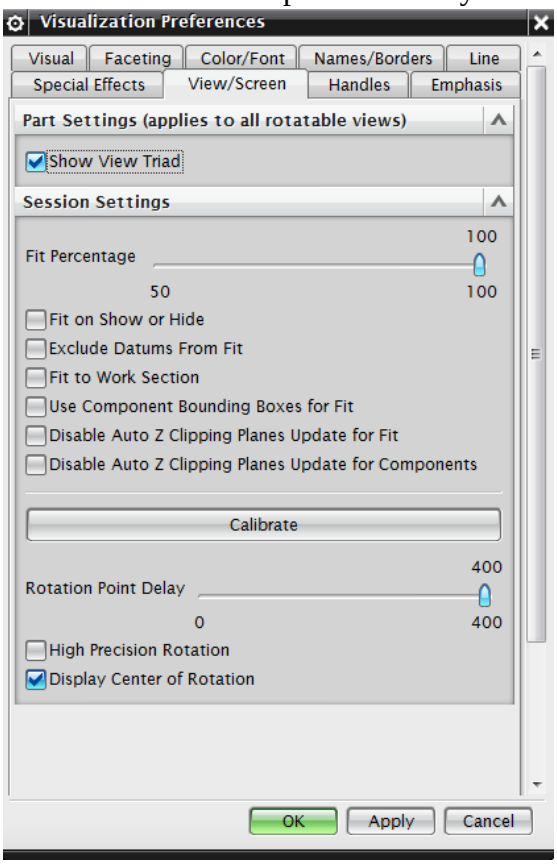
➢ Choose **PREFERENCES** on the ***Menu button [located to top right of the main window]*** to find the various options available

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.

User Interface

➢ Choose **PREFERENCES→USER INTERFACE** to find the options in the dialog box.

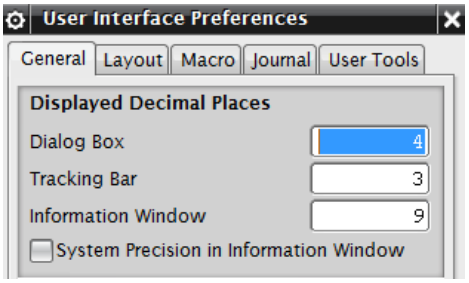
The ***User Interface*** option customizes how NX works and interacts to specifications you set. You can control the location, size and visibility status of the main window, graphics display, and information window. You can set the number of decimal places (precision) that the system uses for both input 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 General tab allows you to set the precision level as seen in the Information Window
- The Layout tab allows you to set the location of the Resource Bar
- The Macro tab allows you to set the pause while displaying animation

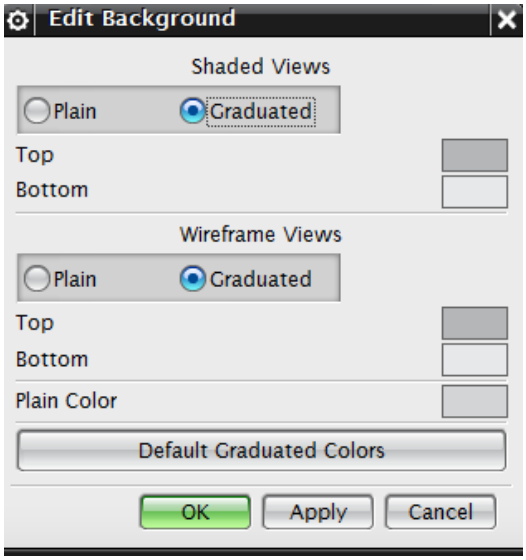
Visualization

➢ 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.

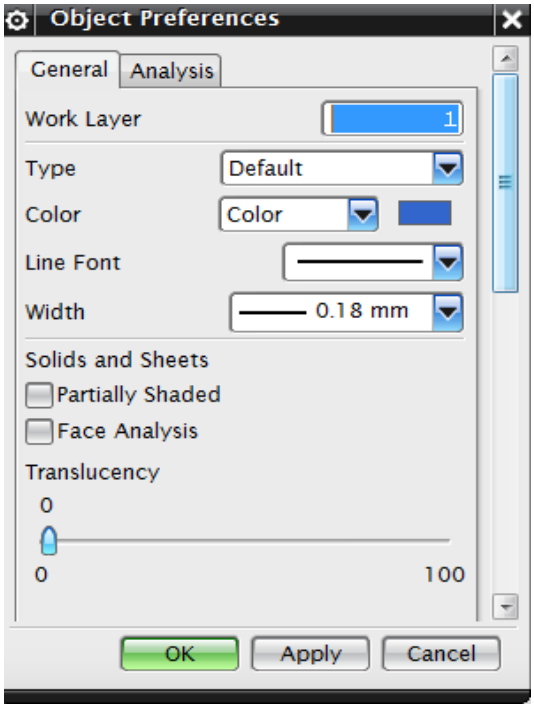


- Click on the different tab buttons to find the options available under each command.

- Choose **PREFERENCES → COLOR PALLETE** to find the options in the dialog box.

- Click on **EDIT BACKGROUND** to get another pop up Dialog box. You can change your background color whatever you want.

The background color refers to the color of the background of the graphics window. NX supports graduated backgrounds for all display modes. You can select background colors for Shaded or Wireframe displays. The background can be Plain or Graduated. Valid options for all background colors are 0 to 255.



- Click **OK** when you are done

You can also update the background of the graphic window using ***Preferences.***

- Choose **PREFERENCES → BACKGROUND**

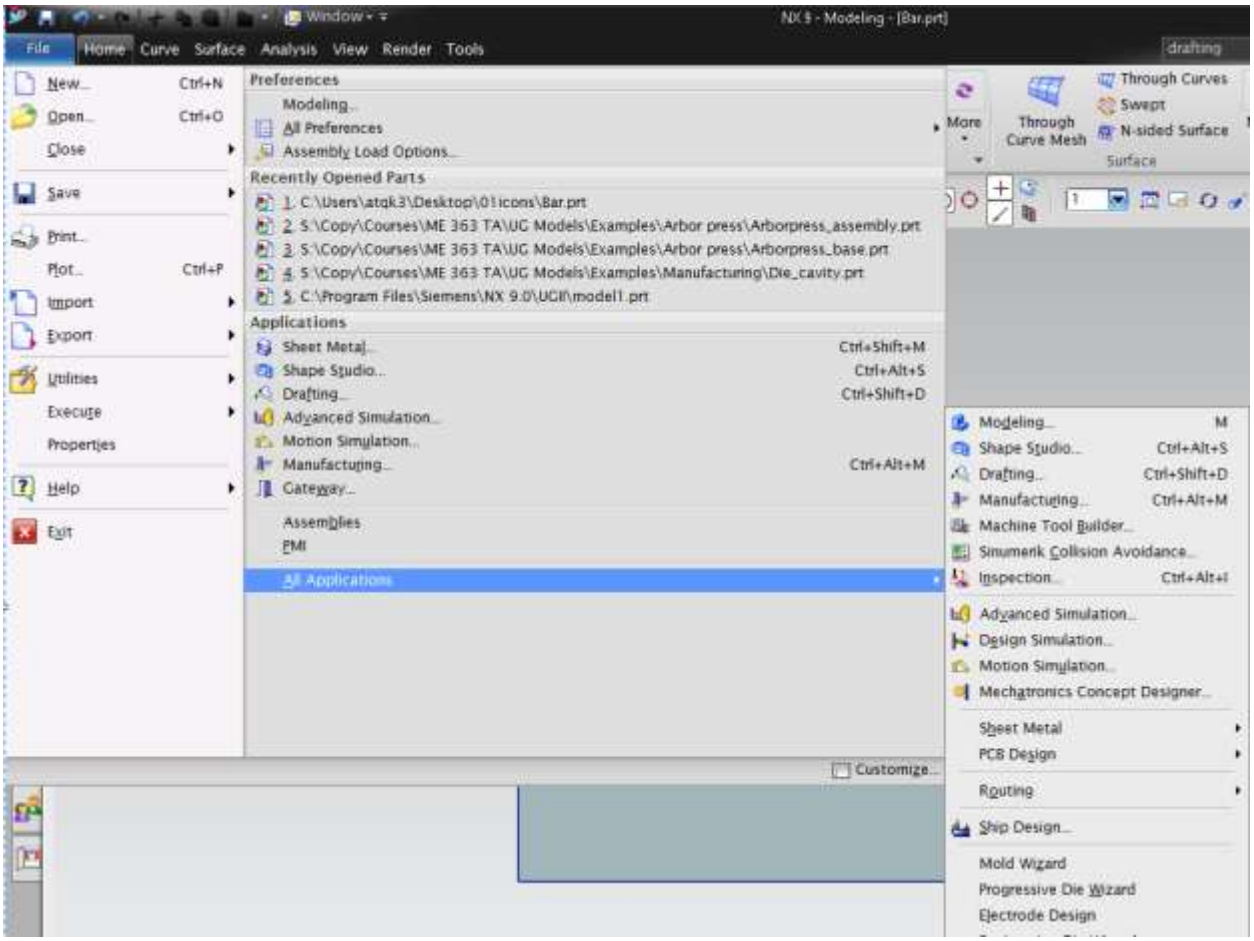
- You can also click **PREFERENCES → OBJECT**

This will pop up a dialog window **OBJECT PREFERENCES** or **EDIT OBJECT DISPLAY**. Change and observe the Color and Translucency of the solid object. This is not just limited to solid objects. 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

Unlike older versions of NX, **APPLICATIONS** can be opened using the **File** option located at the top left corner of the main window. You can select the type of application you want to run from the drop down menu. For example, you can select Modeling, Drafting, assembly, 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**.

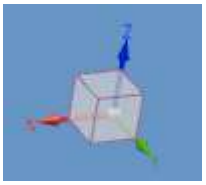


2.4 COORDINATE SYSTEMS

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

2.4.1 Absolute Coordinate System

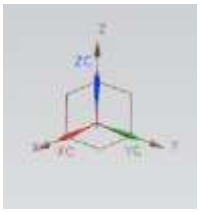
The ***Absolute Coordinate System*** is the coordinate system from which all objects are referenced. This is a fixed coordinate system and hence the locations and orientations of every object in NX9.0 modeling space are related back to this system. The Absolute Coordinate System (or "Absolute CSYS") 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 as well.



The View Triad is a visual indicator that represents the orientation of the Absolute coordinate system of the model

2.4.2 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").

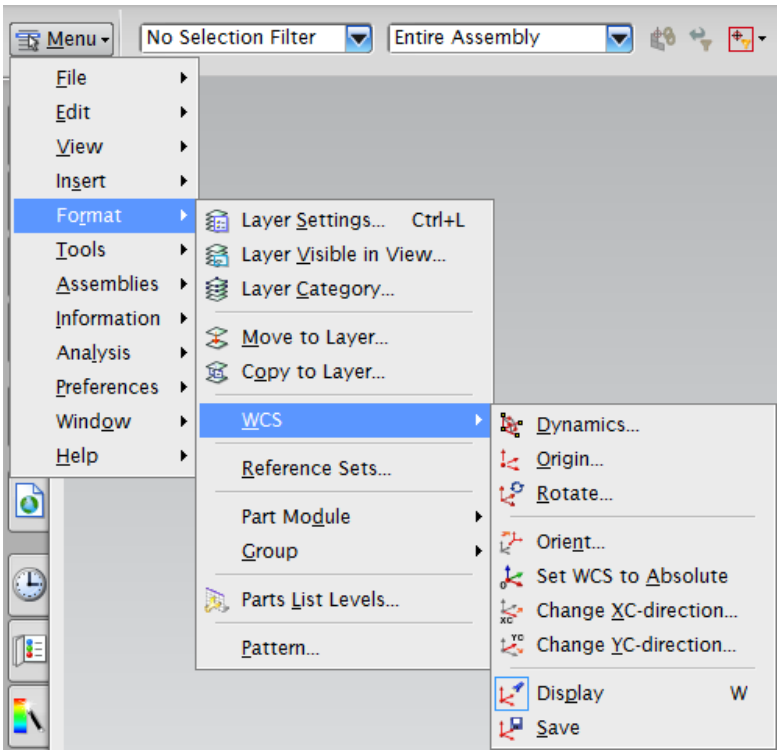


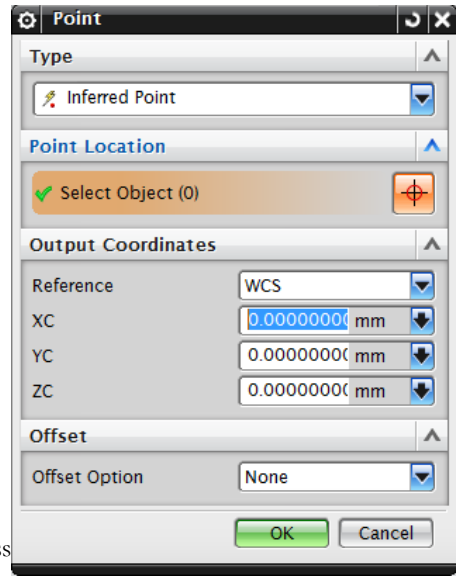
It is possible to have multiple coordinate systems in a part file, but only one of them can be the work coordinate system.

2.4.4 Move the WCS

Here, you will learn how to translate and rotate the WCS.

➢ Choose **MENU ⟶ FORMAT ⟶ WCS**





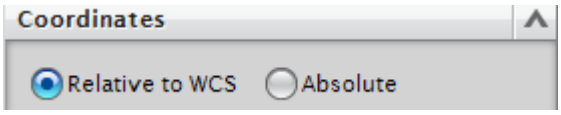
2.4.4.1 Translate the WCS

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.

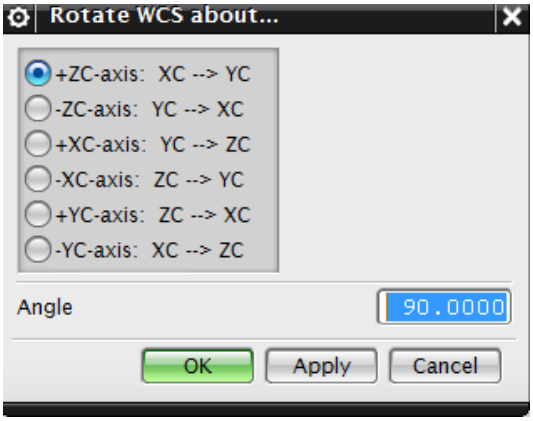
➢ Choose **FORMAT →WCS → ORIGIN**

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

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



The default action button is ***Inferred Point***. The button is highlighted as shown in the figure. The name of the active icon appears above the top row of action buttons. This is the point on the object, which is closest to the cursor. It can be any of the ***Snap Points*** such as the center of circle or end-point of a line and so on.



➢ Click **CANCEL**

2.4.4.2 Rotate the WCS

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

➢ Choose **FORMAT →WCS → ROTATE**

The Rotate WCS dialog is shown on the right side. 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.

➢ Click **CANCEL**

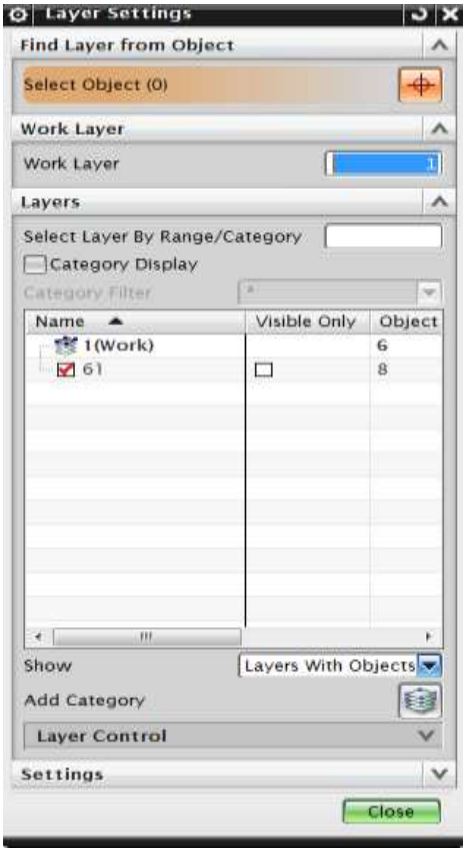
2.4.4.3 Save the Current Location and Orientation of the WCS

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

➢ Choose **FORMAT →WCS → SAVE**

2.5 USING 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 selectability of objects in your file.



2.5.1 Layer Control

With NX9.0, 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 NX9.0 must have. There are 256 usable layers in NX9.0, one of which is always the *Work Layer***.** Any of the 256 layers can be assigned to one of four classifications of status.

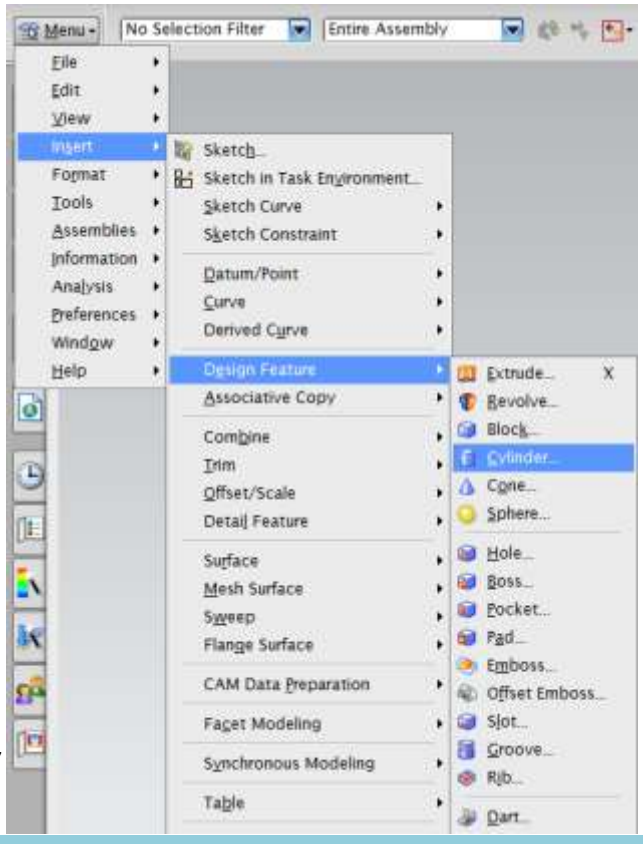
- *Work*
- *Selectable*
- *Visible Only*
- *Invisible*

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 **FORMAT → LAYER SETTINGS**

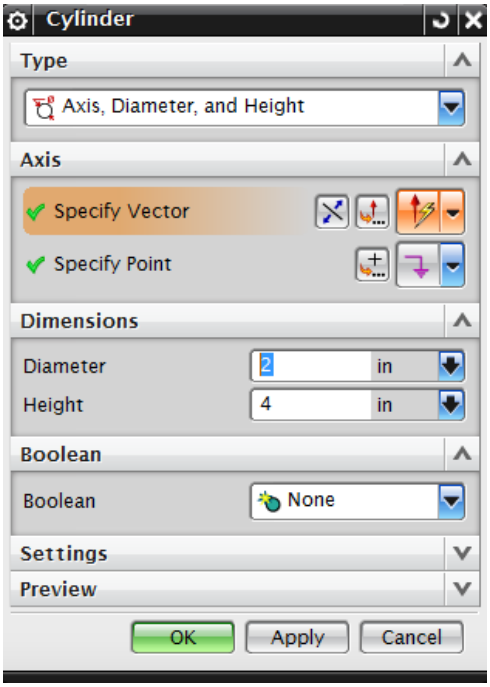
However, it should be noted that the use of company standards in regards to layers would be advantageous to maintain a consistency between files.



2.5.2 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 selected the units to be ***inches*** in the drop-down menu. Choose the file type as ***Model***

- Click **OK**
- Choose **MENU ⟶INSERT⟶ DESIGN FEATURE ⟶CYLINDER**
- Choose **AXIS, DIAMETER, HEIGHT** under **Type**
- Click on icon next to **Specify Vector**

The **Vector Constructor** dialog will appear. This is to specify the direction of the axis of the cylinder. The default direction will be in the ***Z*** direction.

- Click **OK** on the pop-up window
- Click on icon next to **Specify Point**

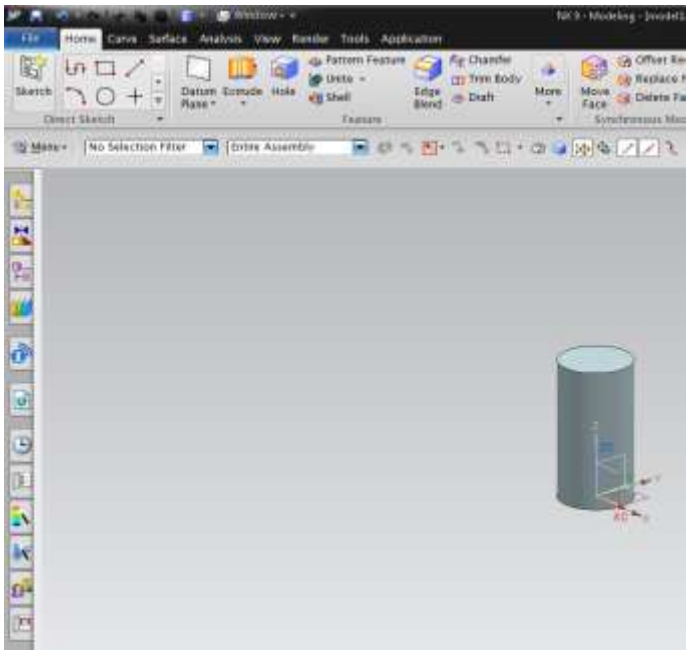
The **Point Constructor** window will appear for you to determine the location of the cylinder. The default location will be the origin ***(0,0,0)*** on the WCS.

- Click **OK**

If you would like to change the direction of Axis or the Point of origin, click on the boxes outlined in red ink as shown in the figure.

- Next type **2** inches for the diameter and **4** inches for the height under **Properties**
- Click **OK**
- Click **CANCEL** on any other window that pops up.

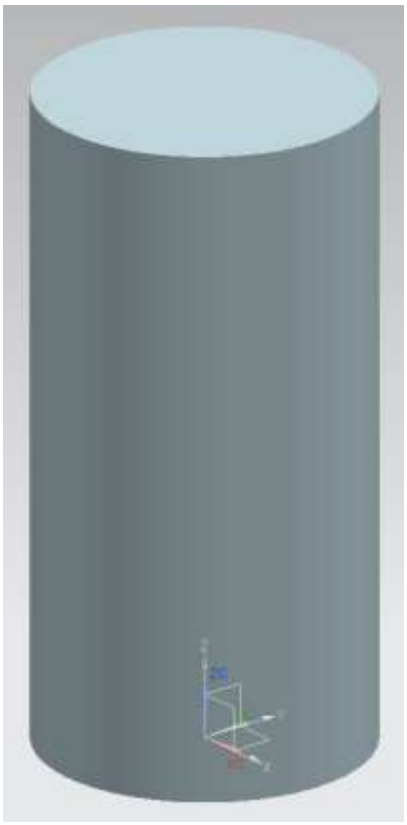
The screen will now look like the following figure.



- **Right-click** on the screen and choose **ORIENT VIEW → TRIMETRIC**

- If the solid is in wire-frame, **right-click** on the screen and choose **RENDERING STYLE → SHADED** OR click on the Shaded icon in the toolbar

Now you will be able to see a solid cylinder.



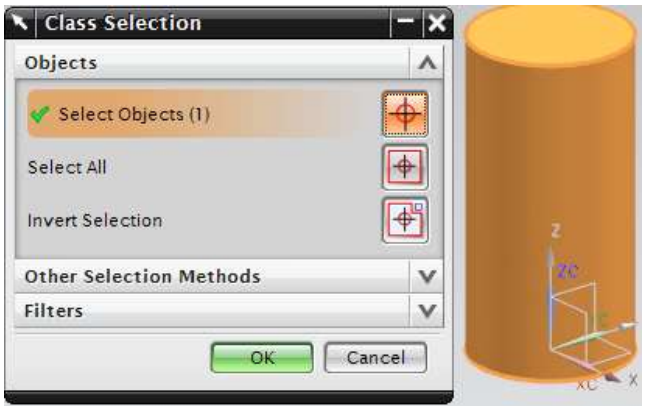
Now let us practice some ***Layer*** Commands.

- Choose **FORMAT → MOVE TO LAYER**

  You will be asked to select an object.

- Move the cursor on to the cylinder and click on it so that it becomes highlighted.

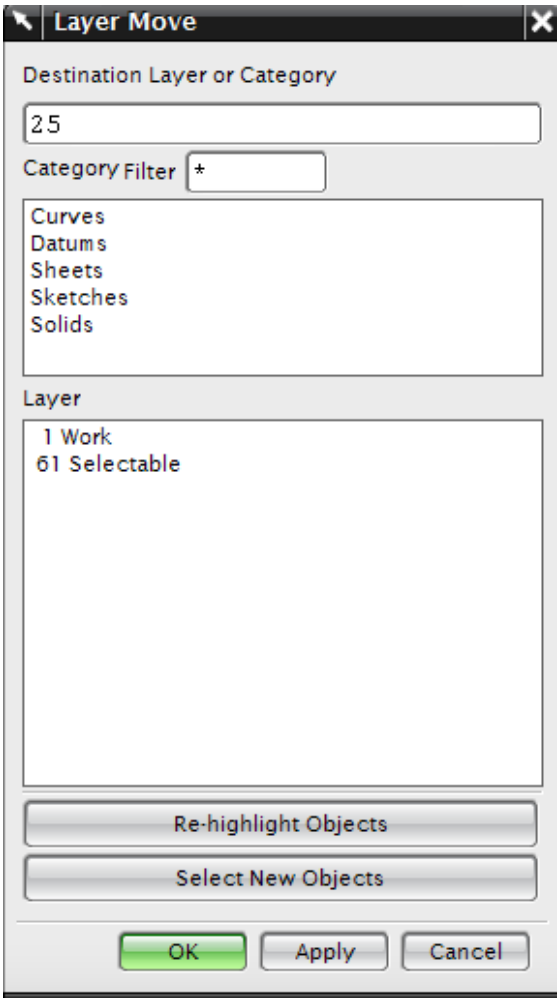
- Click **OK**

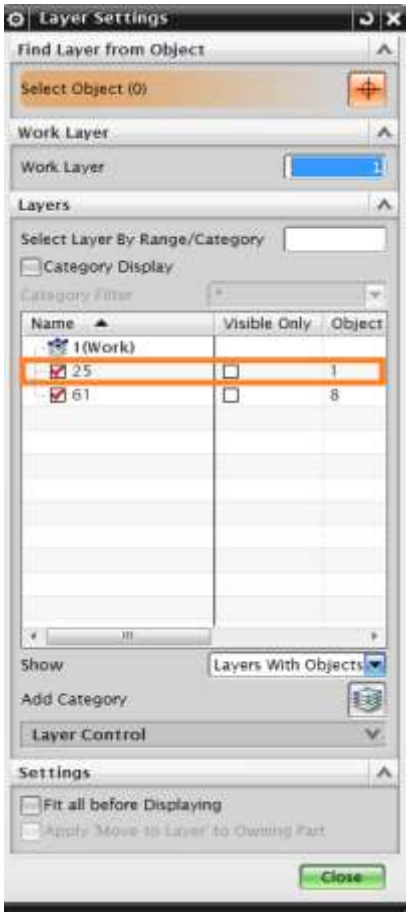


You will get the following pop-up window.

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

- Choose **APPLY**, then **CANCEL**





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

- To see the cylinder, click **FORMAT → LAYER SETTINGS**

You can see that the Layer 25 has the object whereas the default ***Work Layer 1*** has no objects.

- Click **OK**

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

2.6 IMPORTANT COMMANDS/DIALOGS

In this section, you will learn some important commands and dialogs which will be useful during modeling and sketching.

2.6.1 Toolbars

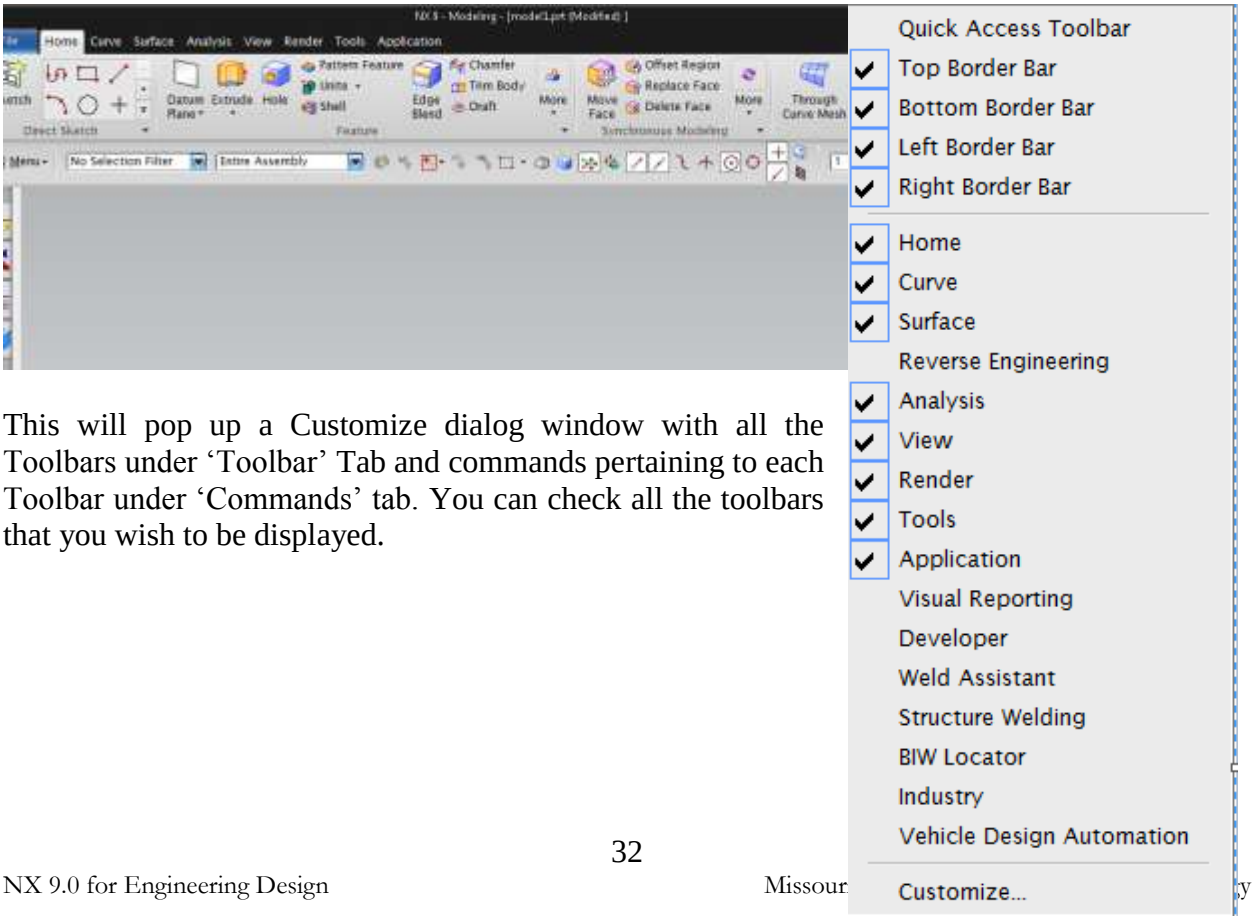
Toolbars contain icons, which serve as shortcuts for many NX9.0 functions. The figure on the right shows the main Toolbar items normally displayed. However, you can find many more icons for different feature commands, based on the module selected and how the module is customized.

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

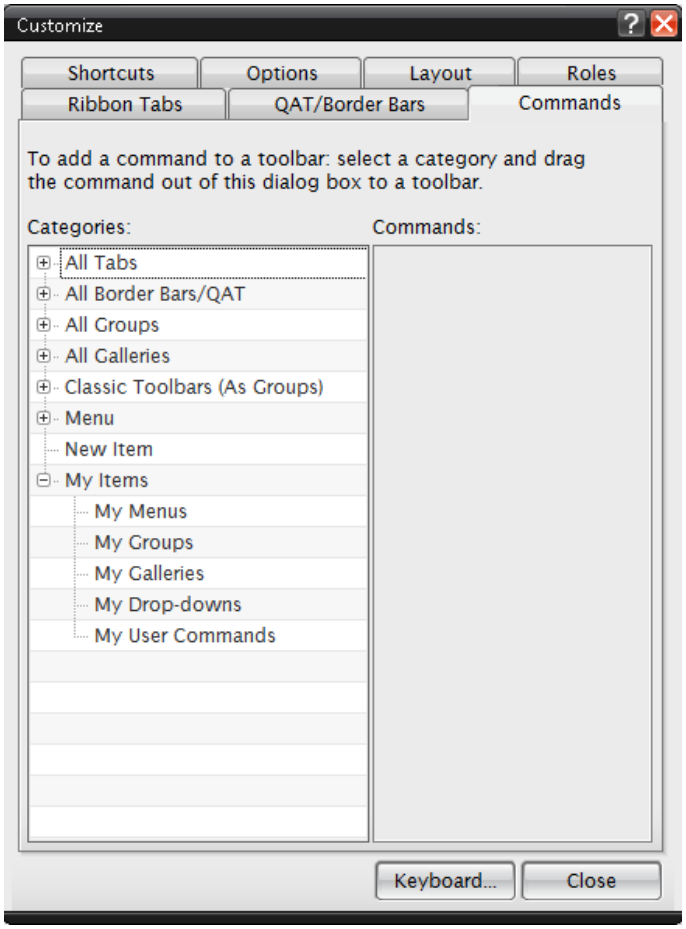
The list of toolbars you can see in the default option is Standard, View, Selection, Utility, etc. Normally, the default setting should be sufficient for most operations but during certain operations, you might need additional toolbars. If you want to add buttons pertaining to the commands and toolbars,

- Click on the pull-down arrow on any of the Toolbars and choose **ADD OR REMOVE BUTTONS**.

- Choose **CUSTOMIZE**.



This will pop up a Customize dialog window with all the Toolbars under 'Toolbar' Tab and commands pertaining to each Toolbar under 'Commands' tab. You can check all the toolbars that you wish to be displayed.



You can customize the settings of your NX9.0 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 NX9.0 interface.

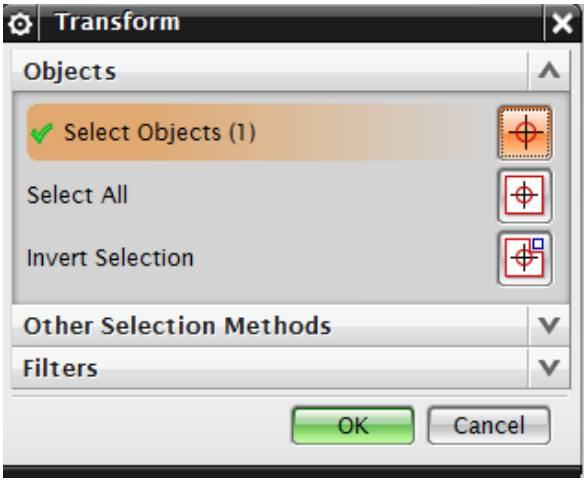
- It allows you to customize the toolbars you desire to be displayed in the Interface.

2.6.2 Transform Functions

Transform functions is powerful feature to scale, mirror and modify an existing part.

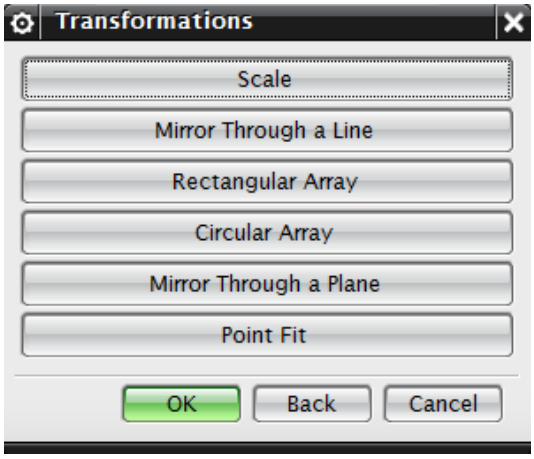
- Open the file that you created in section 2.5.2 with the cylinder.
- Click on **MENU ⟶ EDIT ⟶ TRANSFORM**

  Here, we have to choose an entity such as a solid body or curves or a sketch. You can select a single feature or multiple features by clicking on the features.



- Click on the cylinder so that it gets highlighted
- Click **OK**

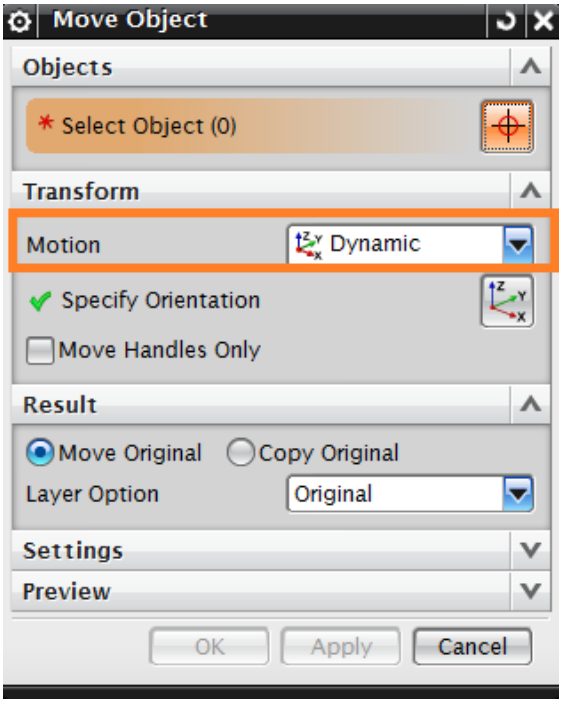
This opens a dialogue box that allows you to perform many functions like scaling, and mirroring part of a model as shown in the following figure.



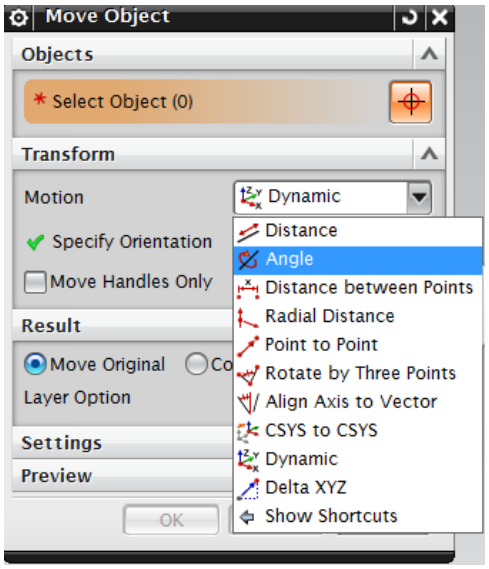
You can choose any of these menus. The ***Scale*** scales the model. You can create a copy of the object using the ***Mirror through a Line*** or ***Mirror through a Plane***, create multiple copies of a entity such as curve, surface or solid using the ***Rectangular*** and ***Circular Array.***

If you want to Move the an object with respect to fixed entity you can use the **Move** menu

- Click on **EDIT → MOVE OBJECT**

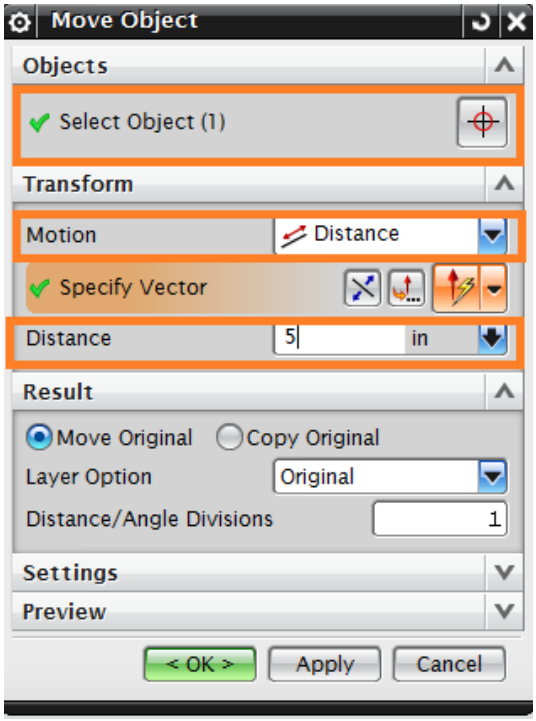


A dialogue box opens with options as shown in the figure above. Note the ***Object Handle*** that appears on the object in the form of a coordinate system. You can select the type of motion from the *MOTION* drop-down menu. The default option is ***Dynamic.*** With this you can move the object in any direction. There are several other ways of moving the object as shown in the figure below.



- **POINT TO POINT** – This option allows you to move the center of the cylinder to any destination point on the X-Y-Z axis that you want to move. The coordinates are based on the WCS.

- **DISTANCE** – This option moves the selected object in the X-Y-Z direction by the distance that you enter.



➢ Click on **SPECIFY VECTOR**
Select the direction.

➢ Type **5** in the **DISTANCE** box. This will translate the cylinder a distance of 5 inches along X-Axis

➢ Click **OK**

Note that **MOVE ORIGINAL** is the ***RESULT*** clicked.

The cylinder will move in the X-direction by a distance of 5 inches.

➢ Click **CANCEL**

As you can see, we have moved the cylinder in 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***. These are the basic commands that you will need initially.

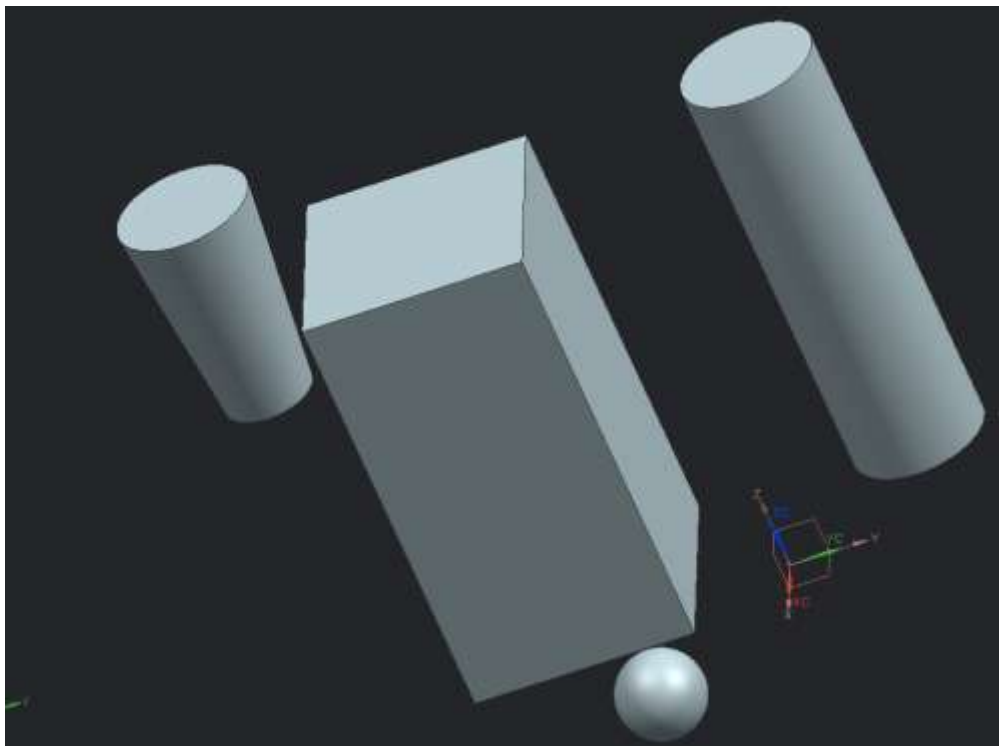
That completes an introduction of the basics of the NX9.0 interface and some basic feature operations that can be done. In the next chapter, we would learn more about the form features and some primitive object types.

CHAPTER 3 - FORM FEATURES

This chapter will discuss the basics of ***Form Features*** in NX9.0: Essentially, we will be discussing what a feature is, what the different types of features are, what primitives are and how to model features in NX9.0 using primitives. This will give a head start to the modeling portion of the NX9.0 and develop an understanding of the use of ***Form Features*** for modeling. In NX9.0 version, the features are categorized in different menus based on the functions and ease of identification.

3.1 OVERVIEW

In NX9.0 ***Features*** is a class of objects that have a defined parent. Features are associatively defined by one or more parents and that retain within the model the order of its creation and modification, 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.

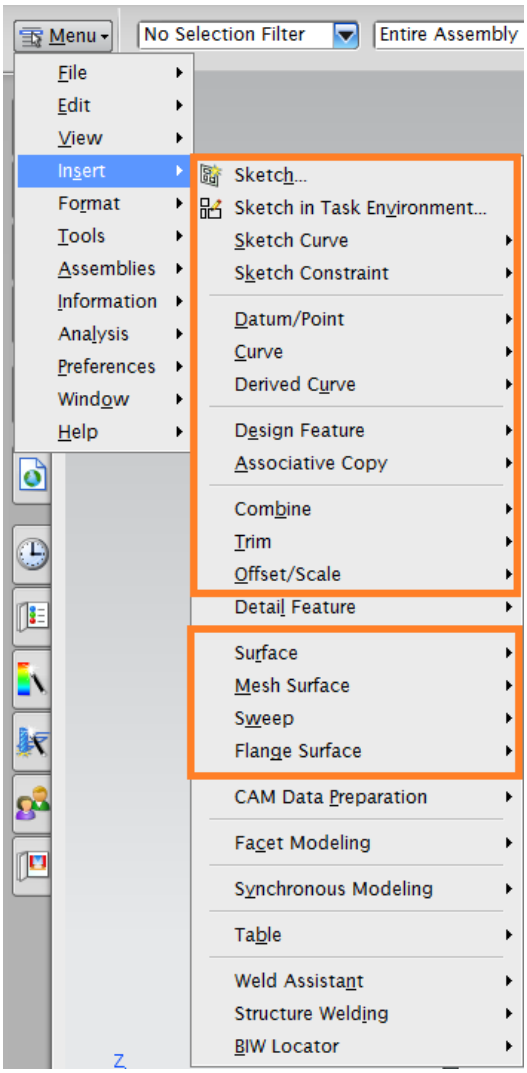


Commonly ***Features*** can be classified as following

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.

3.2 TYPES OF FEATURES

There are six types of Form features: Reference features, Swept features, Remove features, User-defined features, Extract features and Primitives. Just like the NX7 version, the NX9.0 version stores all the ***Form Features*** under the **INSERT** menu option. The form features are also available in the ***Form Features Toolbar***.



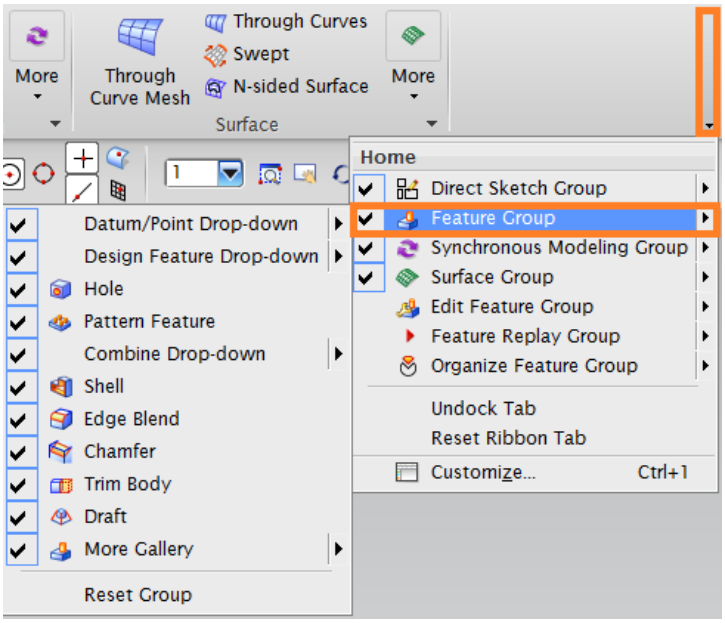
➢ Click **INSERT** on the ***Menu button***

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 in ***Home Toolbar***

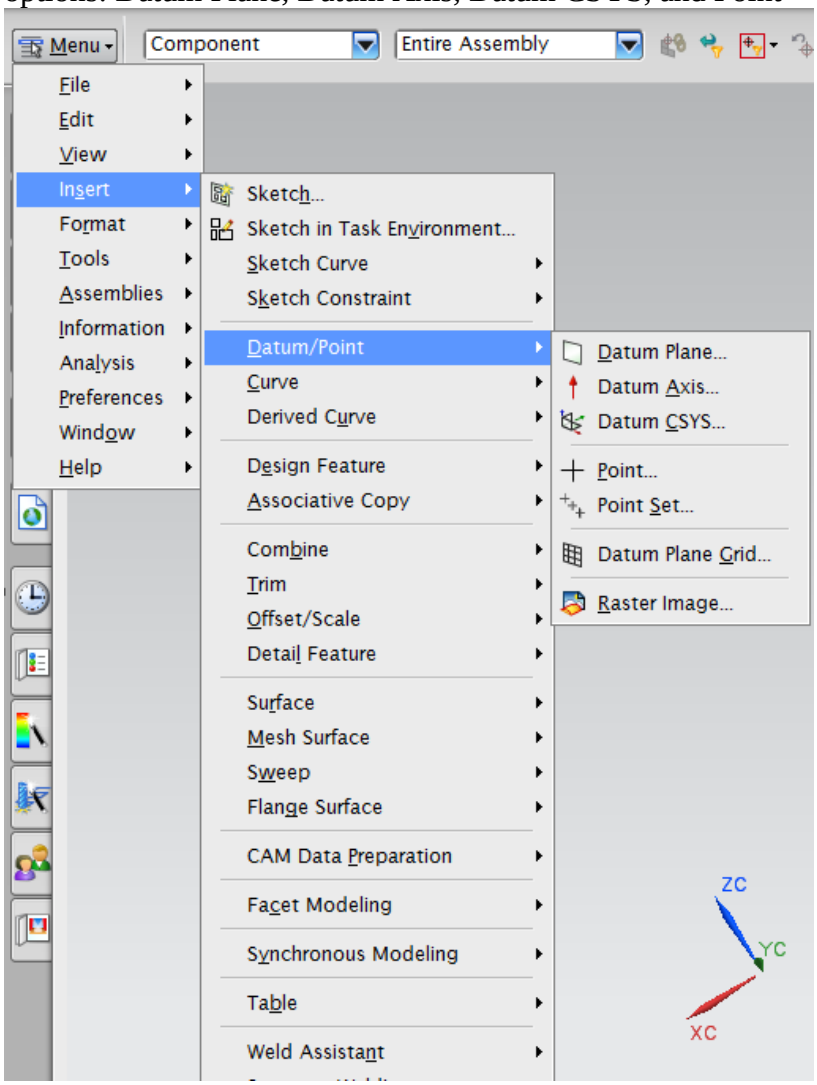
➢ Choose **FEATURE GROUP**



Reference Features

These let you create reference planes or reference axes. These references can assist you in creating features on cylinders, cones, spheres and revolved solid bodies.

➢ Click on **MENU⟶INSERT ⟶DATUM/POINT** to view the different Reference Feature options: Datum Plane, Datum Axis, Datum CSYS, and Point



Swept Features

These let you create bodies by extruding or revolving sketch geometry. Swept Features include:

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

To select a swept feature you can do the following:

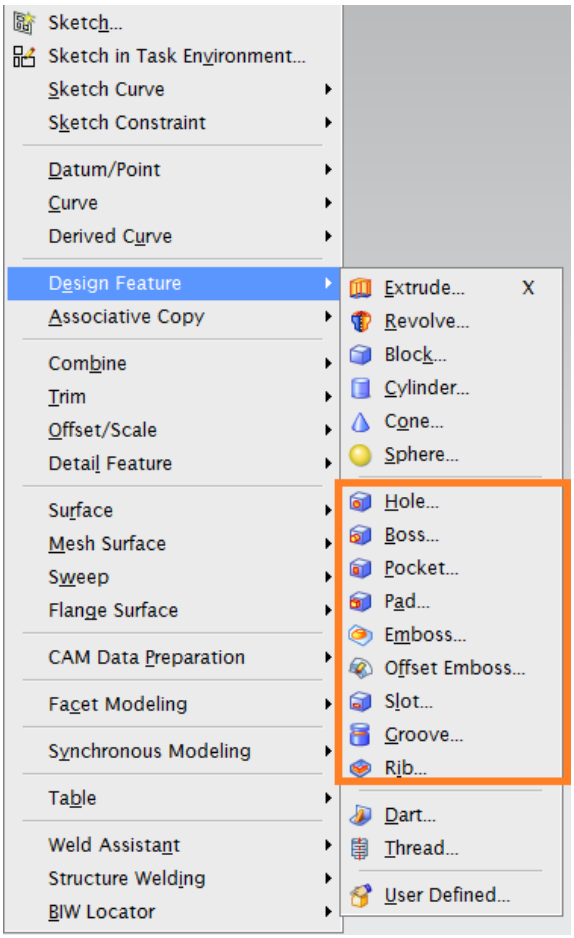
➢ Click on **INSERT → DESIGN FEATURE** for Extrude and Revolve

or

➢ Click on **INSERT → SWEEP** for the rest of the options

Remove Features

Remove Features let you create bodies by removing solid part from other parts.



➢ Click on **INSERT → DESIGN FEATURE**

Remove Features include,

- Hole
- Boss
- Pocket
- Pad
- Slot
- Groove

You can also select the features by clicking on the icons

User-Defined features

These allow you to create your own form 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**

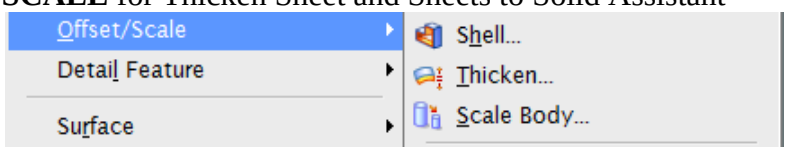
Extract Features

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

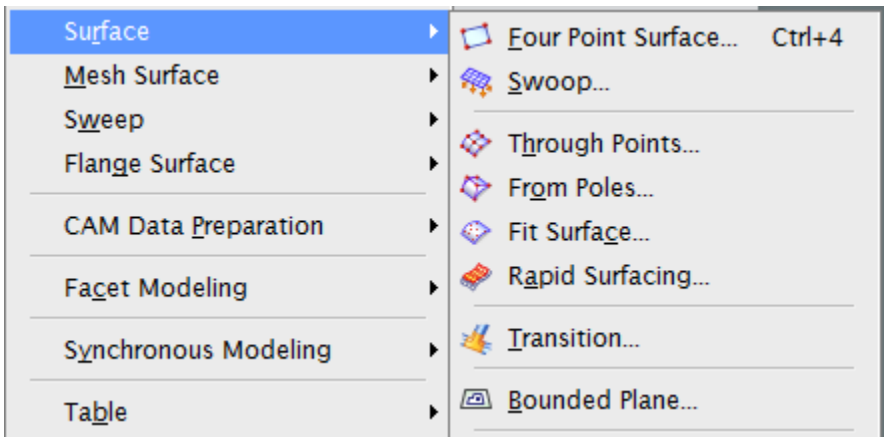
- Extract
- Sheet from curves
- Bounded plane
- Thicken Sheet
- Sheet to Solid Assistant

➢ Click on **INSERT → ASSOCIATIVE COPY → EXTRACT** for Extract options

➢ Click on **INSERT → OFFSET/SCALE** for Thicken Sheet and Sheets to Solid Assistant



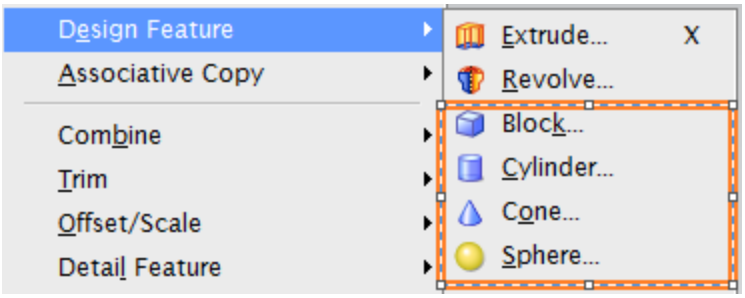
➢ Click on **INSERT → SURFACE** for Bounded Plane and Sheet from curves

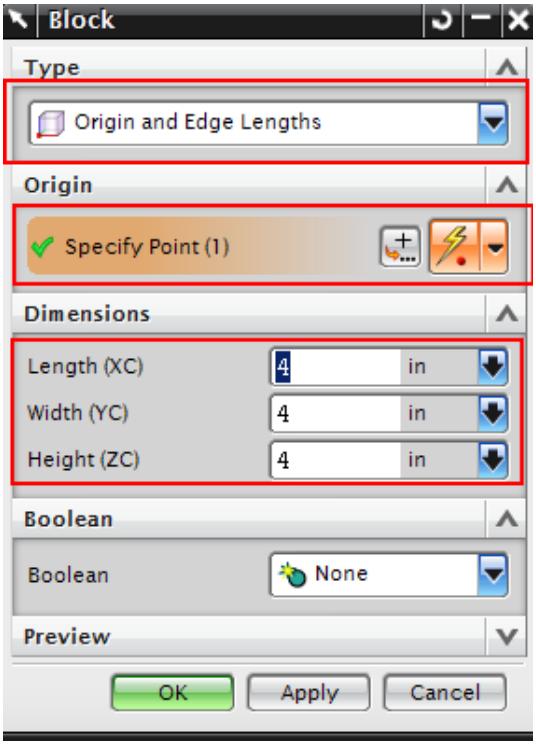


Primitives

They let 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.

3.3 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. The parameters of these primitive objects can be changed.

Now let us start modeling of some basic objects.

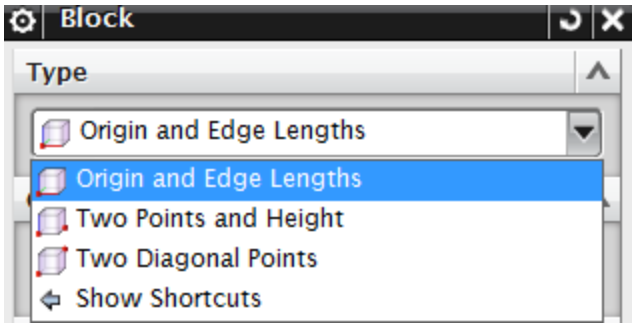
3.3.1 Model a Block

➢ Create a new file and name it as **Arborpress_plate.prt**

Now let us model a plate.

- Choose **INSERT → DESIGN FEATURE → BLOCK** or click on the ***Block*** icon in the ***Form Feature*** Toolbar

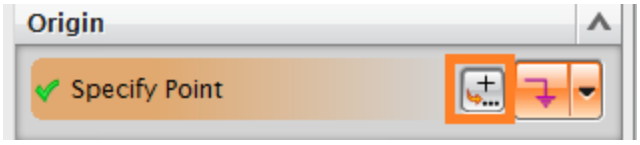
The ***Block*** window appears. There are three main things to define a block. They include the *Type*, *Origin* and the *Dimensions* of the block. To access the *Types* scroll the drop-down menu under *Type*. There are three ways to create a block primitive.



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

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

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



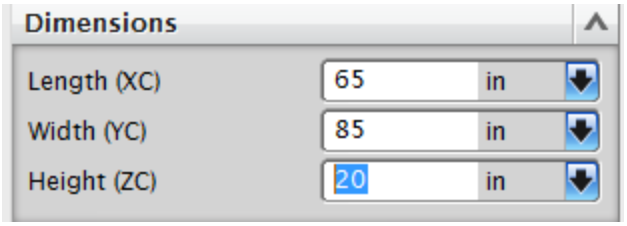
- Click on the **POINT CONSTRUCTOR** icon under the *Origin*

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

- Click **OK**

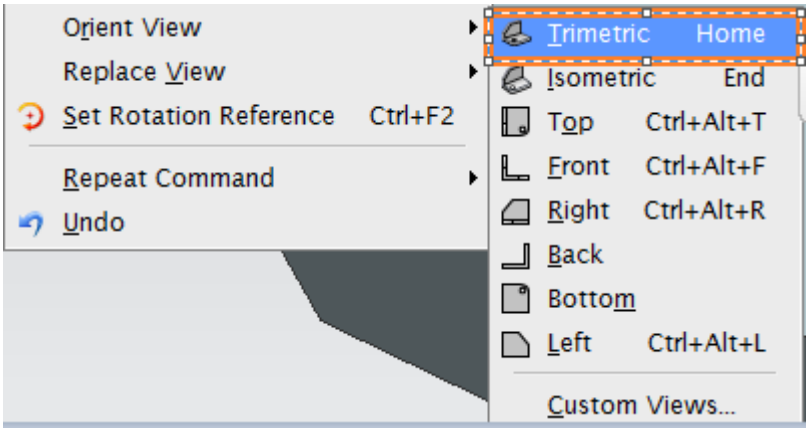
The Block window will reappear.

- Type the following dimensions in the window.
  Length (XC) = **65** inches
  Width (YC) = **85** inches
  Height (ZC) = **20** inches

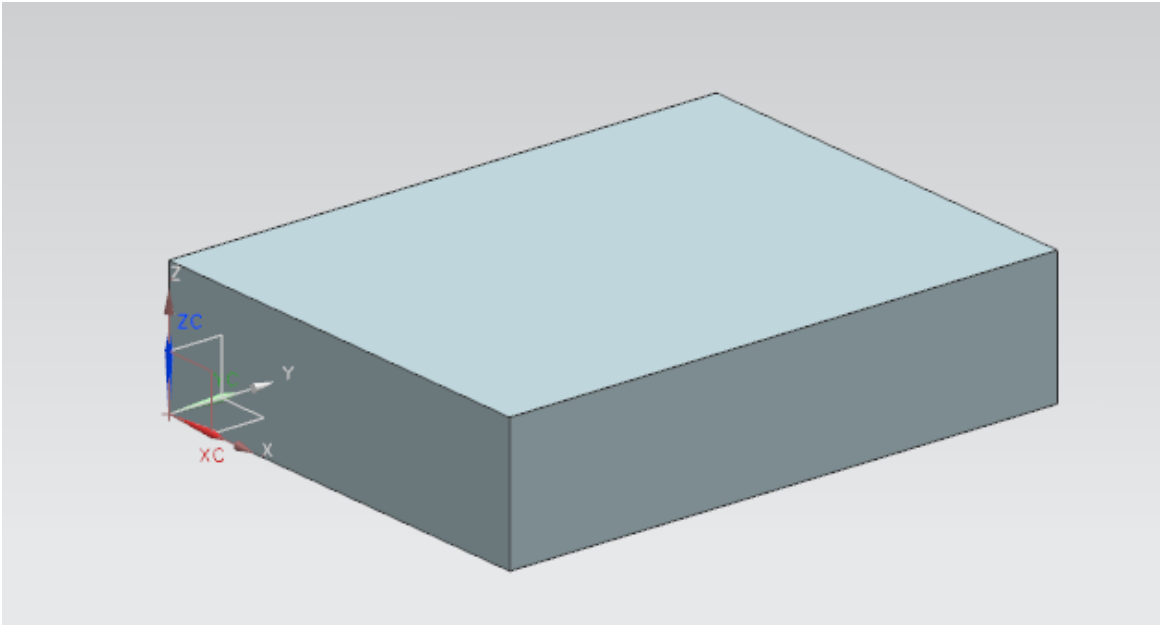


- Click **OK**

- If you do not see anything on the screen, right-click and select ***FIT***. You can also press **<Ctrl> + F**
- Right-click on the screen and click on **ORIENT VIEW → TRIMETRIC**



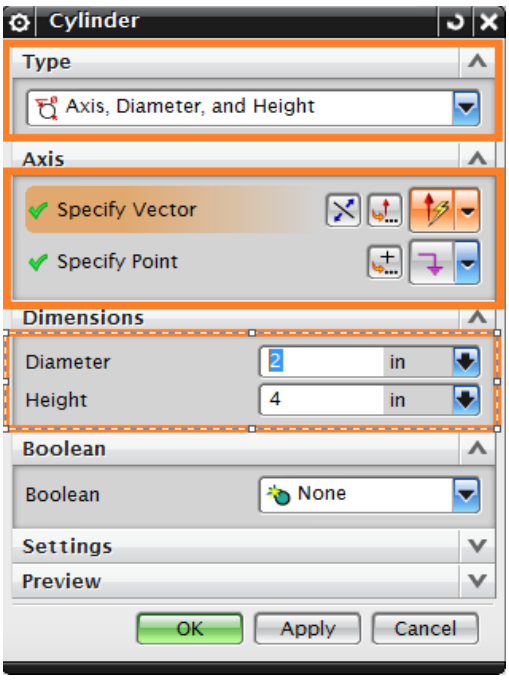
You should be able to see the complete plate solid model. Save and close the part file.



3.3.2 Model a Shaft

After modeling a basic block, we will now model a shaft having two cylinders and one cone joined together.

- Create a new file and save it as **Impeller_shaft.prt**
- Choose **INSERT → DESIGN FEATURE → CYLINDER**



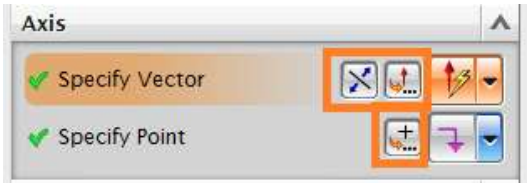
Similar to the ***Block*** there are three things that need to be defined to create a cylinder, *Type, Axis & Origin, Dimensions.*

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

- Axis, Diameter, Height
- Arc, Height

➢ Select **AXIS, DIAMETER, HEIGHT**

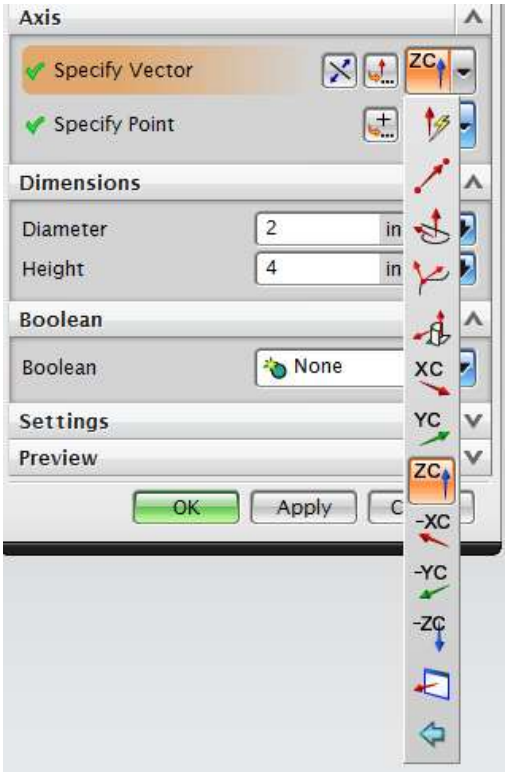
➢ Click on the **Vector Constructor** icon next to **Specify Vector** as shown on the second figure on right.



➢ Click on the **ZC** Axis icon.

➢ Leave the other options as default and click **OK**

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

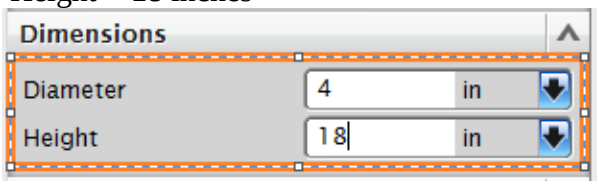


➢ Set all the XC, YC, and ZC coordinates to be 0

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 as shown in figure

Diameter = **4** inches
Height = **18** inches



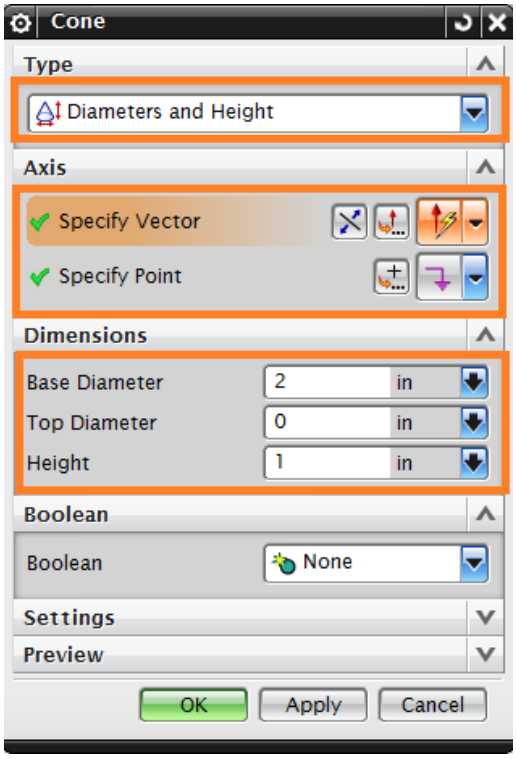
➢ Click **OK**

➢ Click **CANCEL** on any other windows that appear

➢ Right-click on the screen, choose **ORIENT VIEW → ISOMETRIC**

You can change the color of the solid body and the background as mentioned in the Chapter 2.3.4. The cylinder will look as shown below.

Now we will create a cone at one end of the cylinder.



- Choose **INSERT → DESIGN FEATURE → CONE**

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

- Select **DIAMETERS, 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 **OK**

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

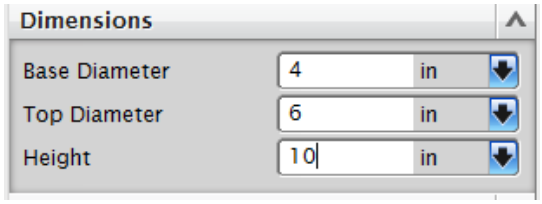
The ***Point Constructor*** window will appear next.

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

OR

- For the Base Point coordinates, 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

- Click **OK**

- On the Boolean Operation window, choose **UNITE**

Now the cone will appear on top of the cylinder.

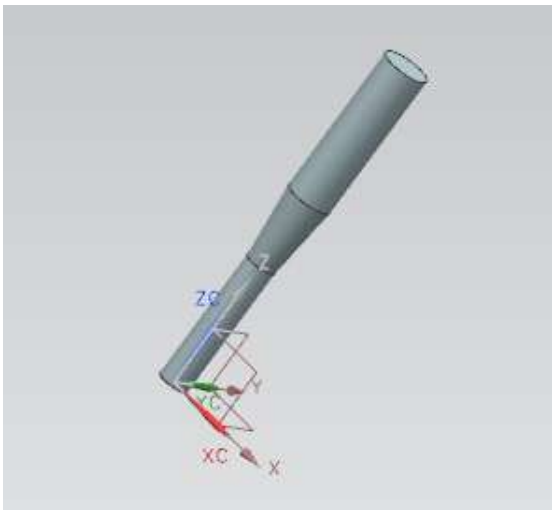
- Click **CANCEL** on any other windows
- Press <Ctrl> + F OR right-click and select ***FIT***

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 below. Remember to save the model.

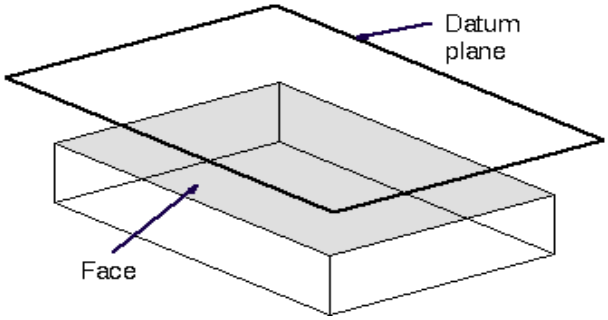


3.4 REFERENCE FEATURES

3.4.1 Datum Plane

Datum Planes are reference features that can be used as a base feature in building a model. ***Datum planes*** 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 face.

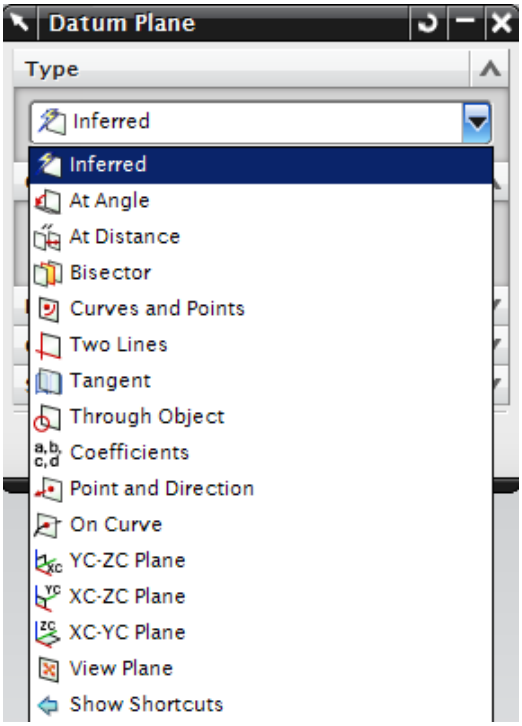


- Open the model **Arborpress_plate.prt**

- Choose **INSERT → DATUM/POINT → DATUM PLANE**

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

The ***Datum Plane*** window, shown on the right side, allows you to choose the method of selection. However, NX9.0 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.

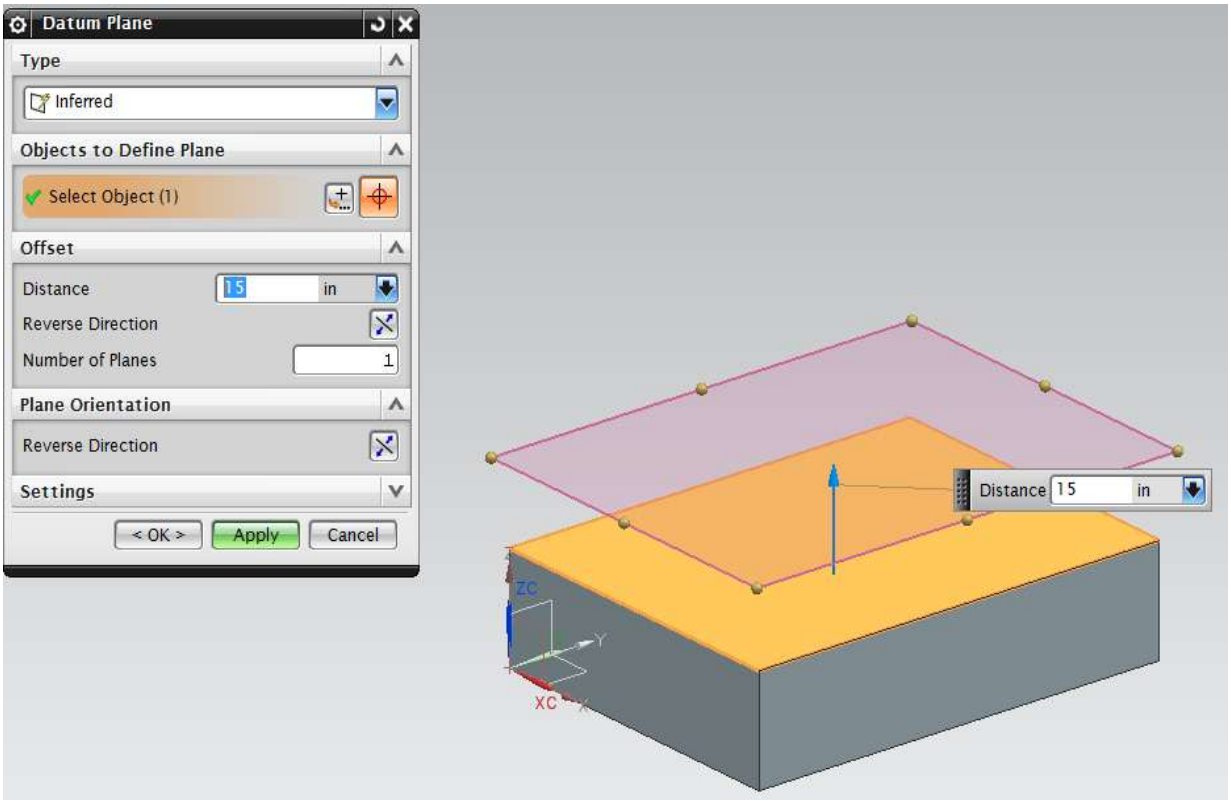


- Click on the top surface of the block so that it becomes highlighted

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.

- Insert the ***Offset Distance*** value as **15** inches in the dialog box and click **APPLY** on the Datum Plane Window

The offset Datum Plane will look as below.

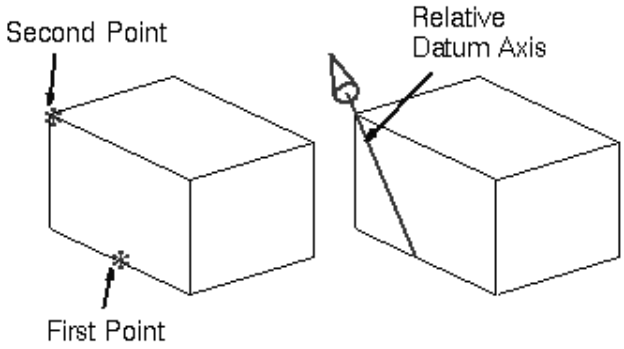


- Click **CANCEL** once you are done

- If you don't see the complete model and plane, right-click and select ***FIT***

3.4.2 Datum Axis

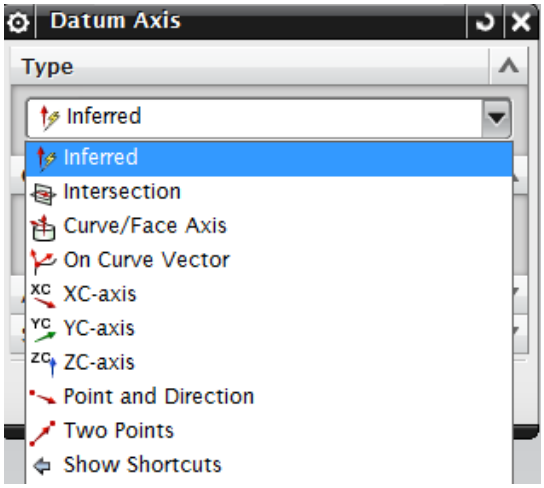
In this part, you are going to create a ***Datum Axis***. A ***Datum Axis*** is a reference feature that can be used to create ***datum planes***, ***revolved features***, ***extruded*** bodies, etc.



Datum Axes 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/POINT → DATUM AXIS**

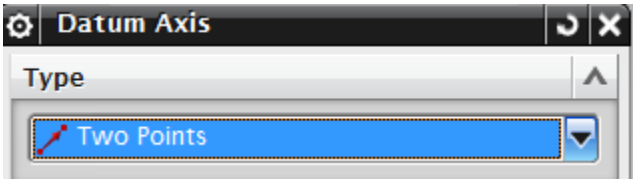
The ***Datum Axis*** dialog can also be opened by clicking the icon as shown in the figure on the right from the ***Feature toolbar***.

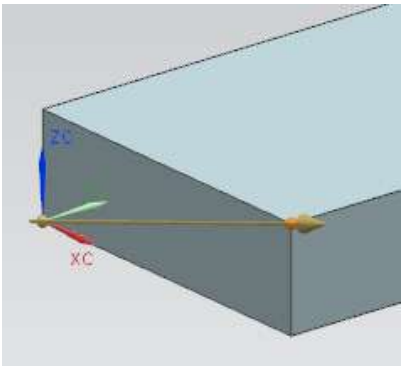


The next window allows you to choose the method of selecting the axis. However, NX9.0 can judge which method to use depending on the entity you select.

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

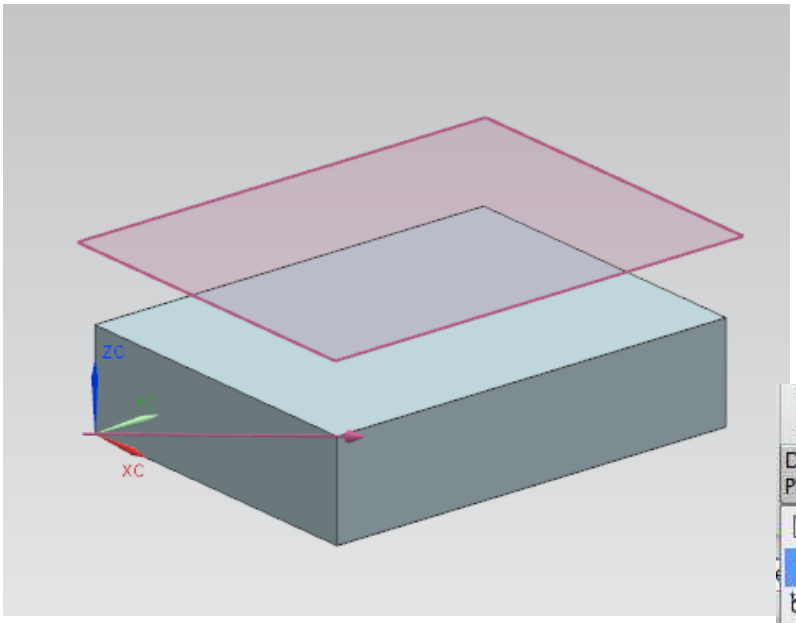
- Select the ***Two Points*** icon at the top right of the ***Datum Axis*** window

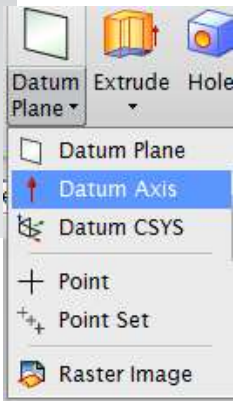




- Select the two points on the block as shown in the figure on the right
- Click **OK**

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

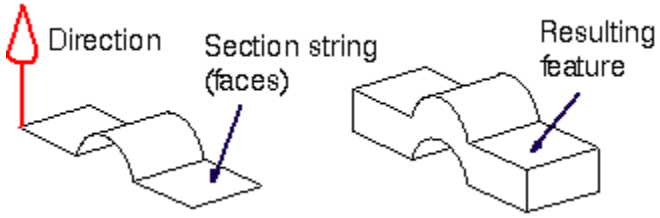




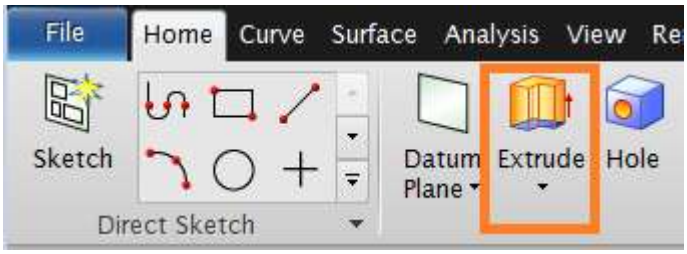
3.5 SWEPT FEATURES

3.5.1 Extruded Body

The ***Extruded Body*** option lets you create a solid or sheet body by sweeping generator geometry (curves, solid faces, solid edges, sheet body) in a linear direction for a specified distance.

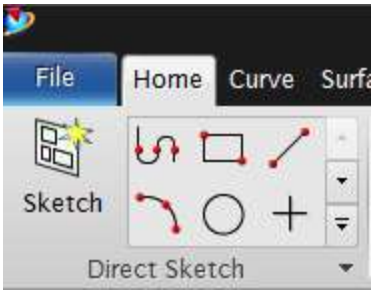


In this part, we will extrude a rectangle into a rectangular block as follows.



- Create a new part file and save it as **Arborpress_rack.prt**
- Right-click, then choose **ORIENT VIEW → ISOMETRIC**

To learn the extrude command, we will create a 2D rectangle first and then extrude this rectangle to form a solid.



- Choose **INSERT → CURVE → RECTANGLE**

You can also choose the ***Rectangle*** icon in the **Sketch Toolbar** as shown below.

The ***Point Constructor*** window will open as shown on the right. Note the ***Cue Line*** instructions. The ***Cue Line*** provides the step that needs to be taken next. You need to define the corner points for the ***Rectangle.***

We have three options for creating a rectangle
-Two point
-Three points
-By center

Select **two points**, and select input mode as **XY**

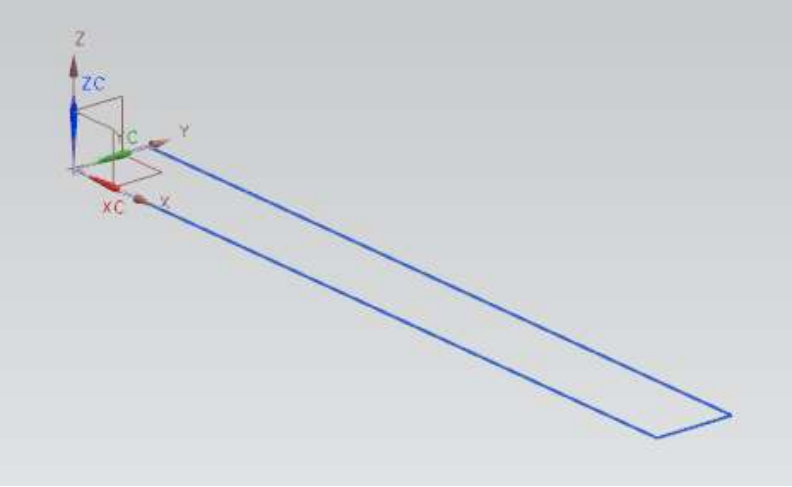
For *Corner Point* 1,

- Type in the coordinates XC = 0, YC = 0, ZC = 0 and click **OK**

Another ***Point Constructor*** window will pop up, allowing you to define the 2nd *Corner Point*

- Type in the coordinates XC = 240, YC = 25, ZC = 0 and click **OK** and then click **Apply**
- Right-click on the screen and choose **FIT**

You should see the rectangle as seen below.



Now we will extrude the rectangle to form a solid.

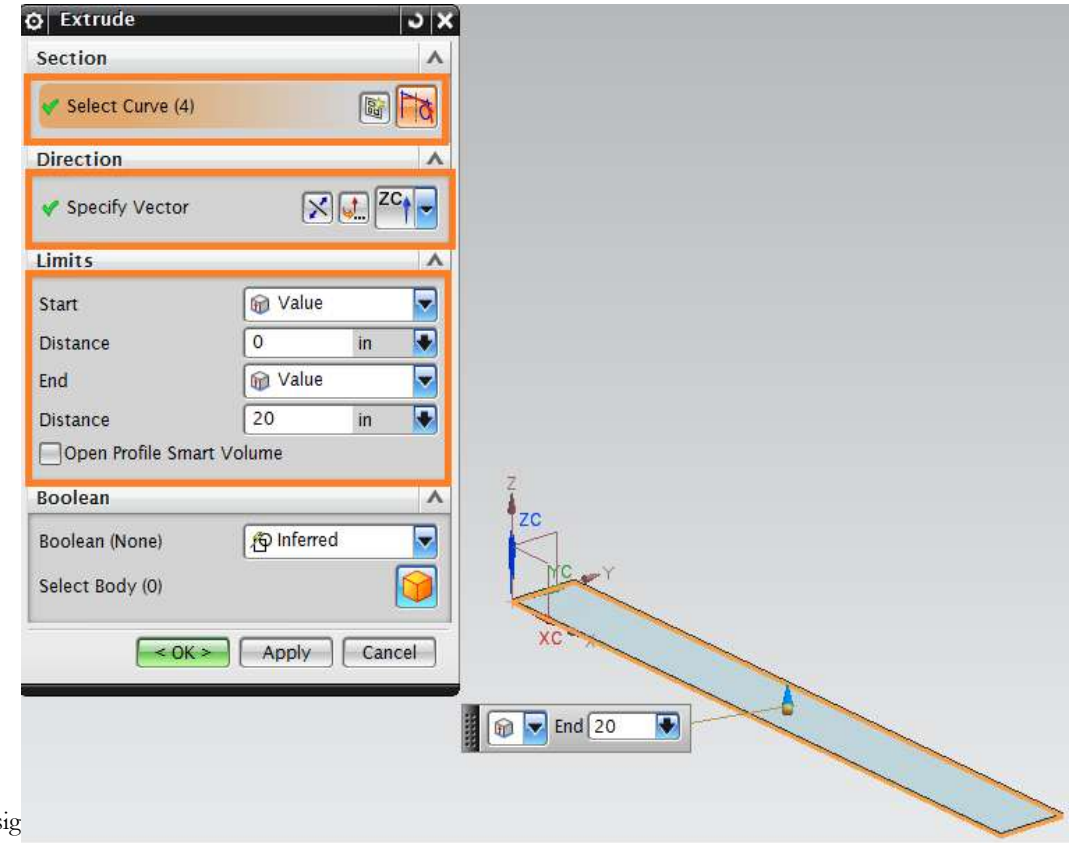
- Choose **INSERT → DESIGN FEATURE → EXTRUDE**

OR

- Click on the ***Extrude*** icon on the left of the ***Form Feature Toolbar*** as shown in the figure below

The ***Extrude*** dialog box will pop up.

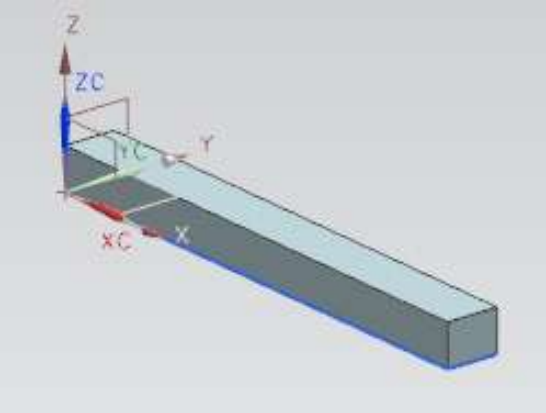
- Click on the ***rectangle***. You can find the preview on the Graphic screen as you proceed with the selection of the lines.
- Choose the default **positive ZC**-direction as the *Direction*
- In the *Limits* window, type in the following values:
  Start = **0**



End = **20**

- Click **OK**

The extruded body will appear as shown below. Save your work and close the file.

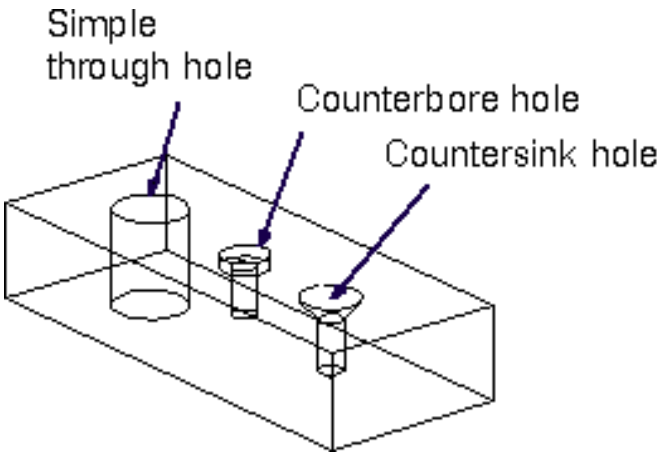


Similar to the Extrude function, we can also perform functions such as Revolve, Tube, etc.

3.6 REMOVE FEATURES

Some features allow you to remove a portion of the existing object to create an object with additional features that are part of the design. These are illustrated below.

Hole:
This option lets you create simple, counter-bored and countersunk holes in solid bodies.



Boss
This option lets you create a simple cylindrical protrusion on a planar face or datum plane.

Pocket
This creates a cavity in an existing body. It can be cylindrical or rectangular.

Pad
Use the Pad option to create a rectangle on an existing solid body.

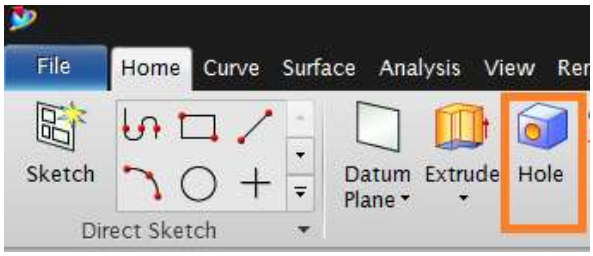
Slot
This option lets you create a passage through or into a solid body in the shape of a straight slot. An automatic subtract is performed on the current target solid. It can be rectangular, T-slot, U-Slot, Ball end or Dovetail.

Groove
This 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 rotating part, as with a turning operation.

Thread
This option allows you to create symbolic thread or a detailed thread on a cylindrical face of a solid body.

We will now learn to create holes.

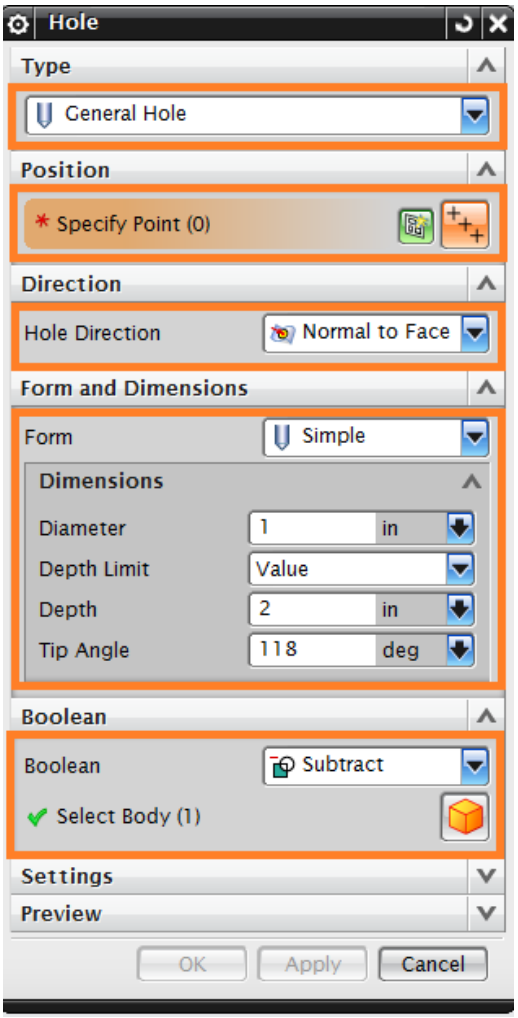
- Open the file **Arborpress_plate.prt**
- Choose **INSERT → DESIGN FEATURES → HOLE** or click on the icon in the ***Feature Toolbar*** as shown



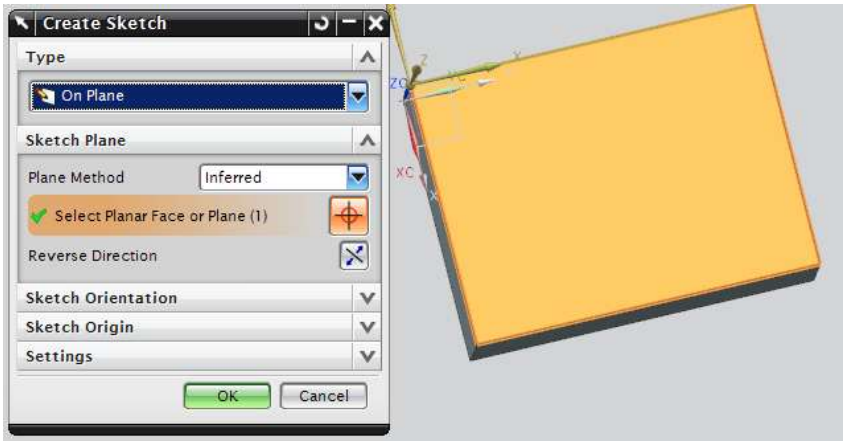
The ***Hole*** window will open. There are various selections that need to be done prior to making the holes. First you need to select the *Type* of the hole.

- Select the default **General Hole**

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

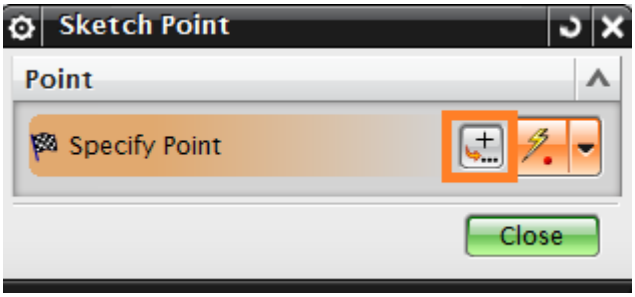


- Click on the ***Sketch*** icon in the *Position* dialog box and choose the top face of the plate as the *Type of Sketch*



- Click **OK**

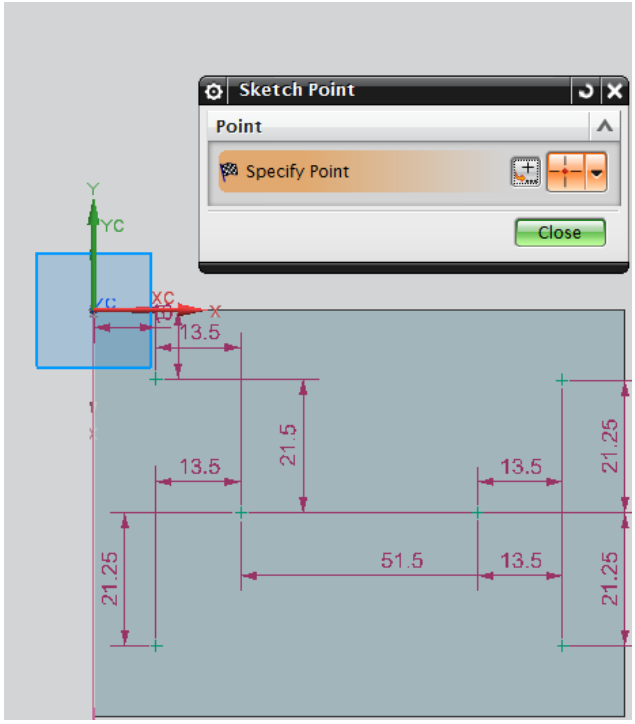
This will take you the ***Sketch Plane.***



- Click on the **POINT CONSTRUCTOR** icon and specify all the points as given in the Table below.

You can specify either one point or all the points at the same time.

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

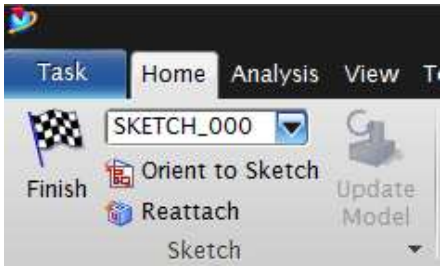


- Click **OK** after you enter the coordinates of each point.

  Note the dimensions of the hole appear automatically. You can hide all the dimensions by selecting, and right clicking and selecting **Hide**

- Click **CLOSE** once you have entered all the points.

- Click on **FINISH SKETCH** 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.

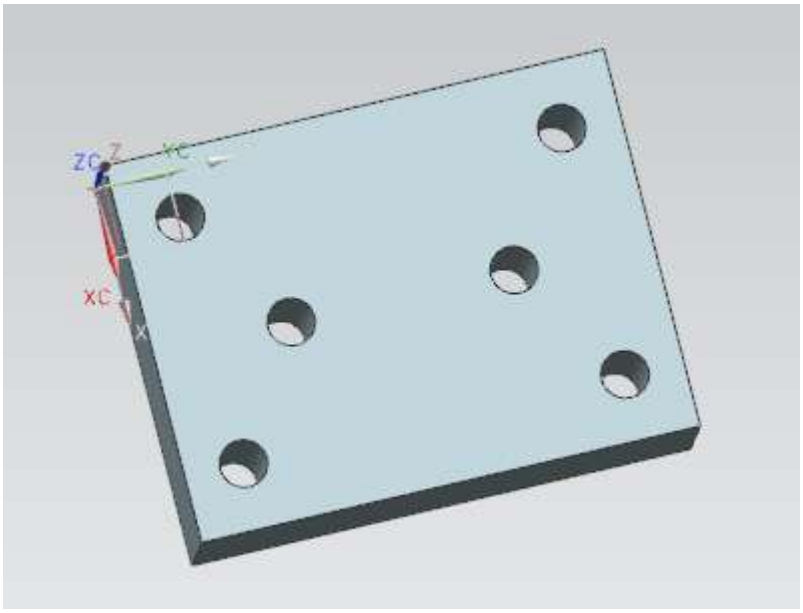
- In the *Form* dialog choose the default option of ***Simple Hole.***

- 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

- Click **OK**

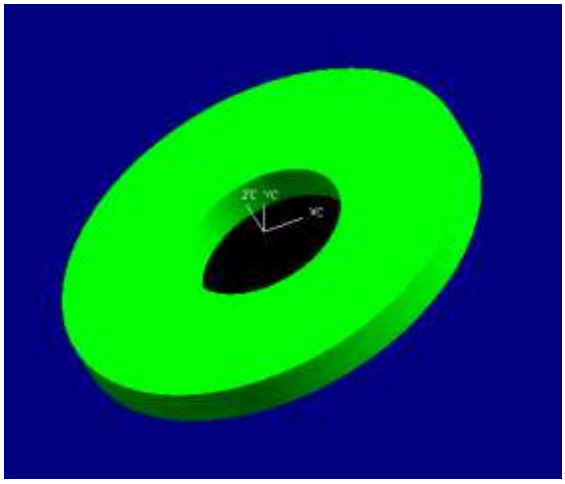
The final plate will be as shown below.



We have now completed the basic form features. The user-defined form features are advanced options in which new form features are added into the library.

3.7 EXERCISE - MODEL A WASHER

As an exercise, model a washer as shown in the figure.



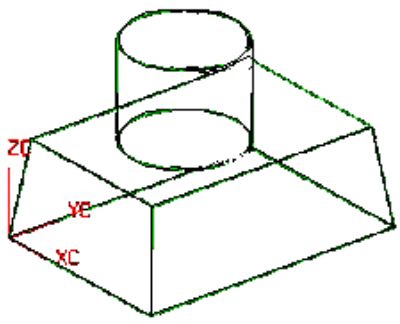
The washer has the following dimensions.
Outer diameter = 0.73 inches
Inner diameter = 0.281 inches
The thickness of the washer can vary within realistic limits. For practice take the value to be 0.05 inches.

CHAPTER 4 – FEATURE OPERATIONS

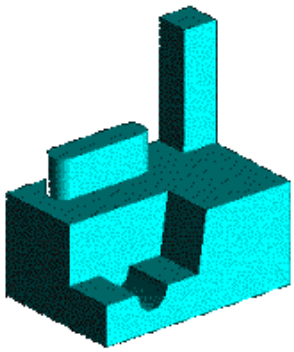
Feature operations are the continuation of ***Form Features***. In this chapter, you will learn some of the functions that can be applied to the faces and edges of a solid body or feature you have created. These include taper, edge blend, face blend, chamfer, trim, etc. After explaining the feature operations, the chapter will walk you through some examples. As mentioned in the beginning of Chapter 3, ***Feature Operations*** are categorized into different options under the **INSERT** menu. Therefore, you will not find a single menu group as '***Feature Operations***' under the **INSERT** menu, however all the ***Feature Operations*** are grouped in the ***Features Toolbar***.

4.1 OVERVIEW

Feature operations are performed on the basic ***Form Features*** to smooth corners, create tapers, and unite or subtract certain solids from other solids. Some of the feature operations are shown below.



TAPER



BLEND

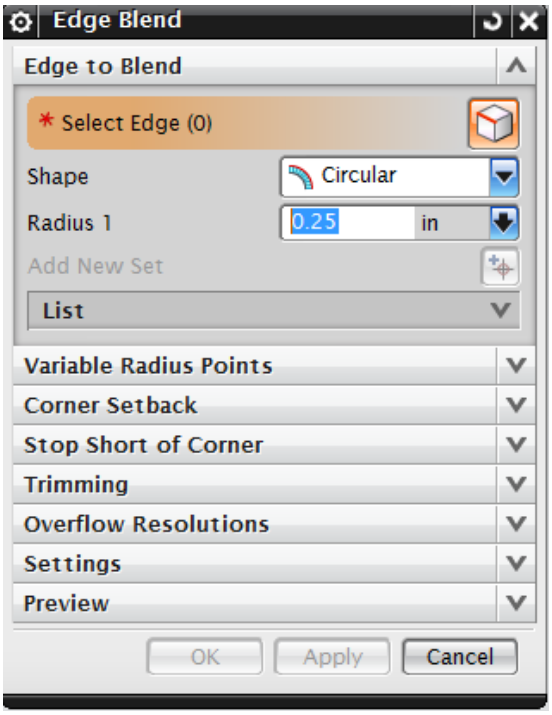
Let us see the different types of feature operation commands in NX9.0 and the function of each command.

4.2 TYPES OF FEATURE OPERATIONS

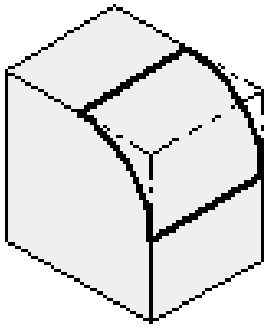
The ***Features Operations*** used in NX9.0 include Edge blend, Face blend, Soft blend, Chamfer, Hollow, Instance, Sew, and Patch. Let us see some of the important operations in details.

The ***Feature Operations*** can be categorized into five main components found under the **INSERT** menu. They are as ***Associative Copy***, ***Combine***, ***Trim***, ***Offset/Scale*** and ***Detail Feature.*** In this chapter we will be lookinssg at some of these operations only.

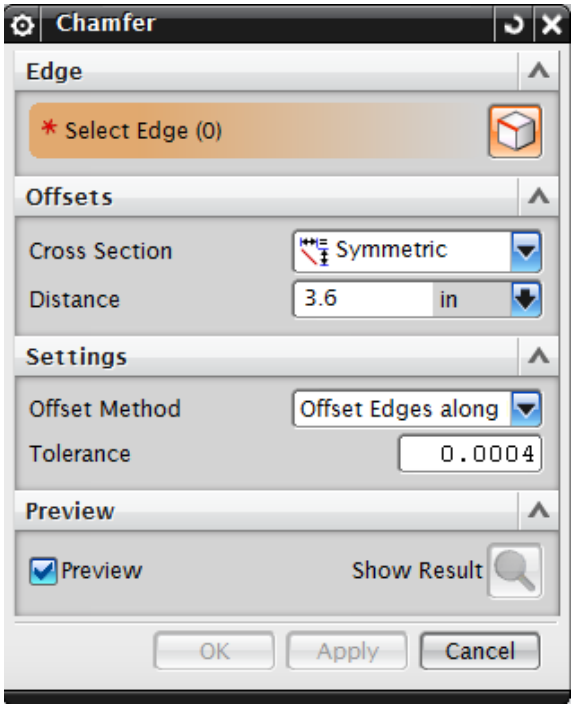
Edge Blend



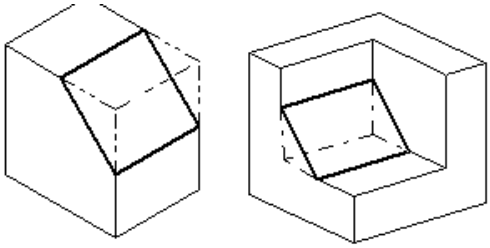
An ***Edge Blend*** is a radius blend that is tangent to the blended faces. This feature modifies a solid body by rounding selected edges. This command can be found under **Insert → Detail Feature → Edge Blend**
In this case you need to select the edges to be blended and define the *Radius* of the blend.
Similar to ***Edge Blend*** you can also do a ***Face Blend***.



Chamfer



The ***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 can also be found under the **Insert → Detail Feature → Chamfer** menu.

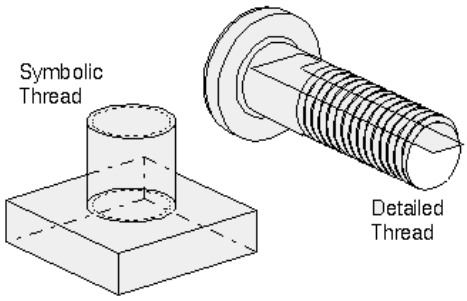


You can preview the result of chamfering and if you are not happy with the result you can undo the operation.

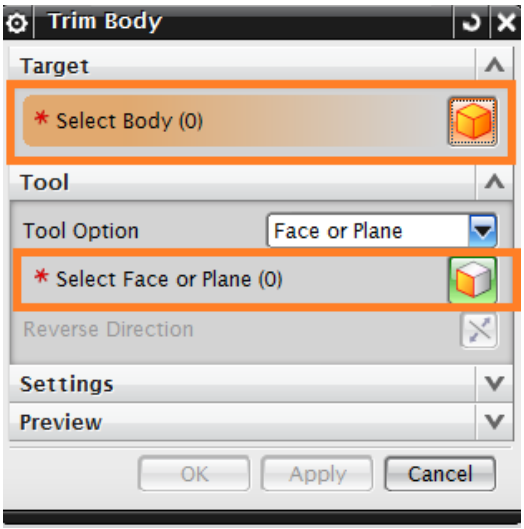
Thread

Threads can only be created on cylindrical faces. The Thread function lets you create symbolic or detailed threads (on solid bodies) that are right or left hand, external or internal, parametric, and associative threads 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**



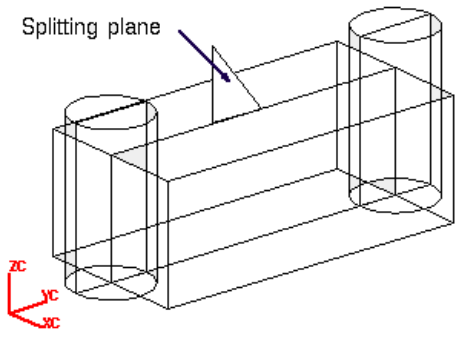
Trim Body

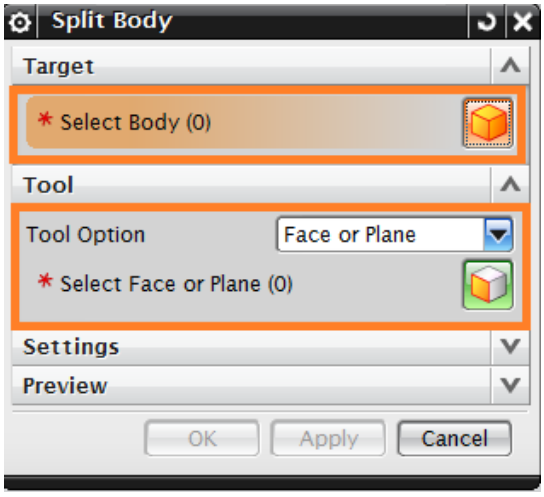


A solid body can be trimmed by a sheet body or a datum plane. You can use the ***Trim Body*** function to trim a solid body with a sheet body and at the same time retain parameters and associativity. To use this command, go to **Insert → Trim → Trim Body**

Split Body

A solid body can be split into two just like trimming it. It can be done by a plane or a sheet body. **Insert → Trim → Split Body**

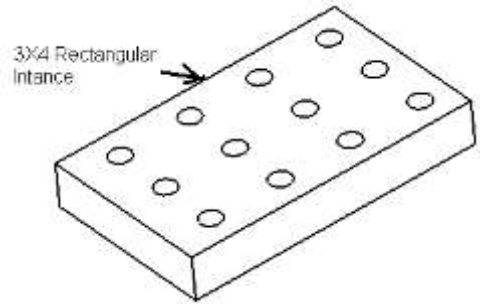


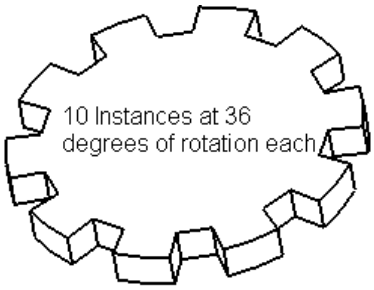


Instance

A **Design Feature** or a **Detail Feature** can be made into dependent copies in the form of an array. It can be *Rectangular* or *Circular* array or just a ***Mirror***. This particularly helpful feature saves plenty of time and modeling when you have similar features. For example threads of gear

or holes on a mounting plate, etc. This command can be found by going to **Insert → Associative Copy → Instance Feature**.



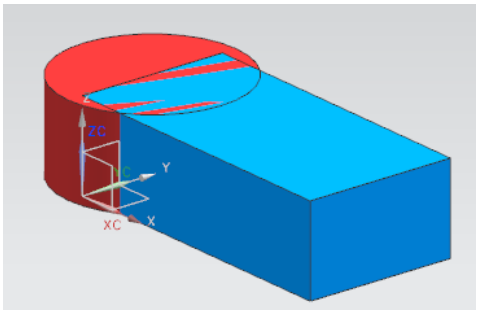


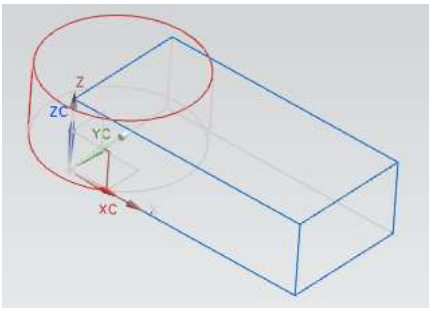
Boolean Operations

Boolean operations are:

- Unite
- Subtract
- Intersect

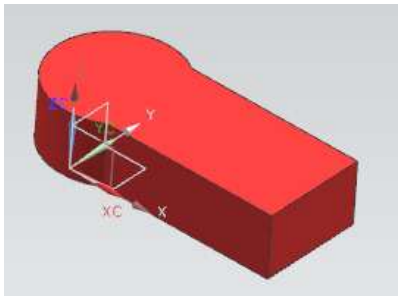
These options can be used when two or more solid bodies share the same model space in the part file. To use this command, go to **Insert → Combine Bodies**.
Consider two solids given. The block and the cylinder are next to each other as shown below.

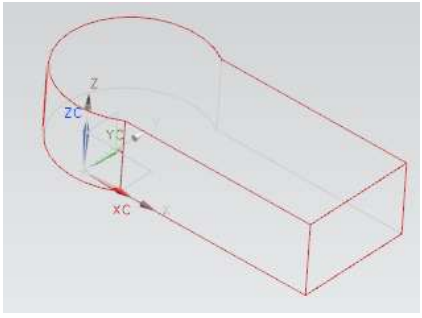




Unite:

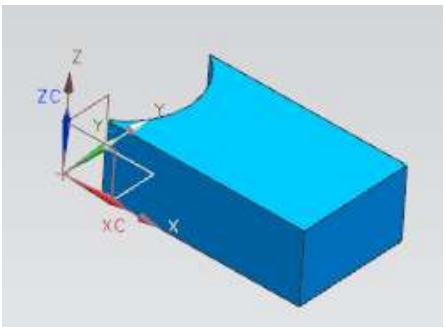
The unite command adds the *Tool body* with the *Target body*. For the above example, the output will be as follows if ***Unite*** option is used.

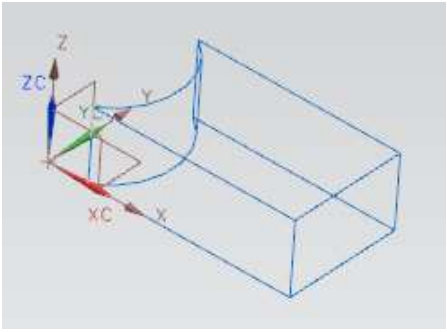




Subtract:

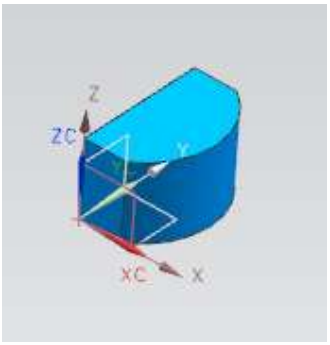
When using the subtract option, the ***Tool body*** is subtracted from the ***Target body***. The following would be the output if the rectangle is used as the Target and the cylinder as the Tool.

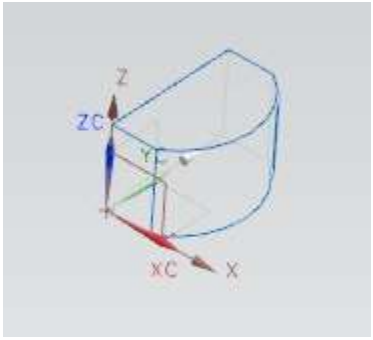




Intersect:

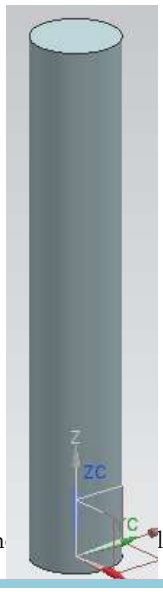
This command leaves the volume that is common to both the ***Target body*** and the ***Tool body***. The output is shown below.





4.3 FEATURE OPERATIONS ON MODELS

In the previous chapter, we dealt with some of the ***Form Features***. In this chapter, we will see how the *primitives* and basic ***form features*** can be converted into complex models by using ***Feature Operations***. The following are a set of examples that will guide you in creation of some simple models.



4.3.1 Model a 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.

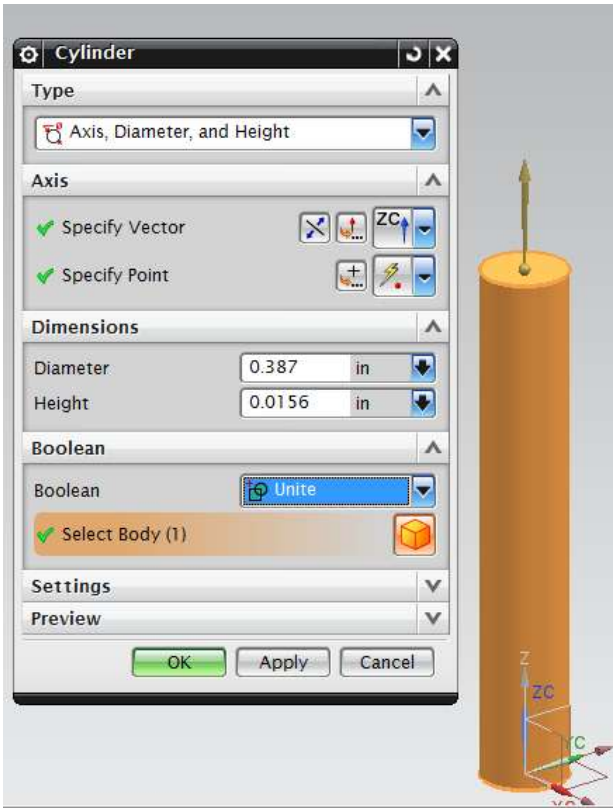
- The dimensions of this cylinder are,
  Diameter = **0.387** inches
  Height = **0.0156** inches

- Click on the top face of the existing cylinder as shown in the following figure

- On the **Point Constructor** window, choose the *Arc/Ellipse/Sphere Center* icon from the drop-down ***Type*** menu

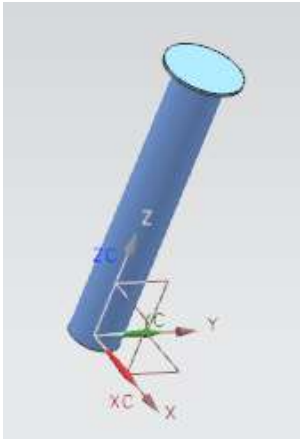
- Click **OK** to get out of **Point Constructor** window.

- Under the **Boolean** drop-down menu, choose **UNITE**

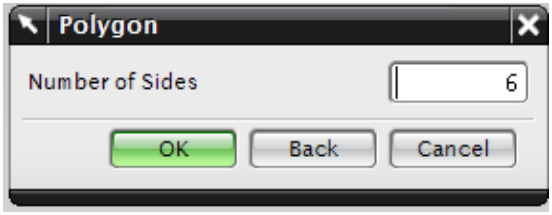


The two cylinders should look like the figure shown below. Save the model.

Next, we will create a hexagon for the head of the bolt.



- Choose **INSERT → CURVE → POLYGON**

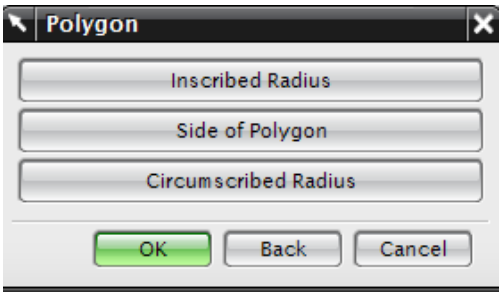


- On the **Polygon** window, type **6** for the number of sides

- Click **OK**

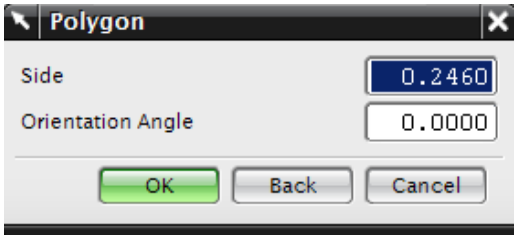
There are three ways to draw the polygon.

- Inscribed Radius
- Side of Polygon
- Circumscribed Radius



➢ Choose **SIDE OF POLYGON**

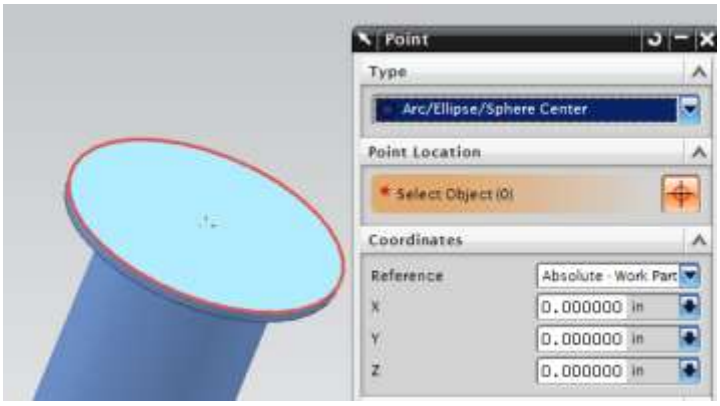
➢ On the next window, enter the following dimensions.
Side = **0.246** inches
Orientation Angle = **0.00** degree



➢ Click **OK**

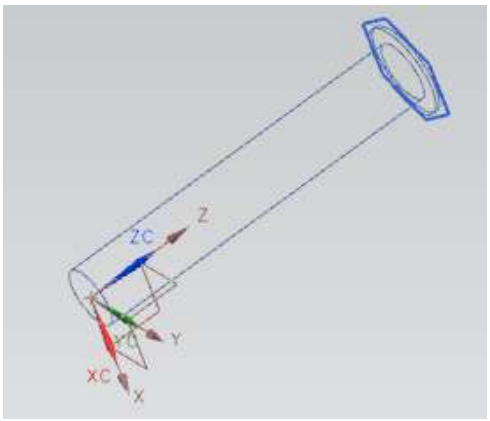
➢ On the **Point Constructor** window, again choose *Arc/Ellipse/Sphere Center* icon

➢ Click on the top face of the last cylinder drawn (small cylinder)



➢ Click **CANCEL**

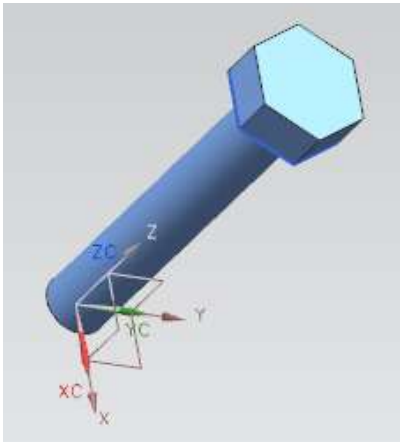
The polygon will be seen as shown below. If the model is not in wireframe, click on the ***Wireframe*** icon in the **View Toolbar**



Now we will extrude this polygon.

- Choose **INSERT → DESIGN FEATURE → EXTRUDE**
- Click on all six lines of the hexagon to choose the surface that is required to be extruded
- Enter the End Distance as **0.1876** inches

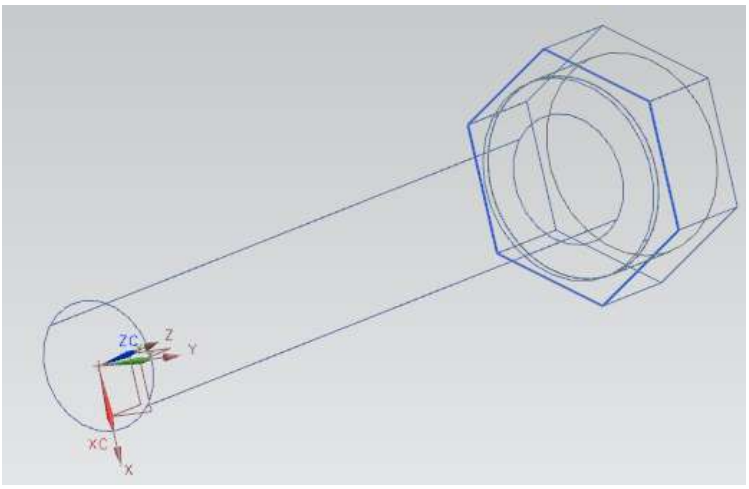
The model looks like the following after extrusion



- 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

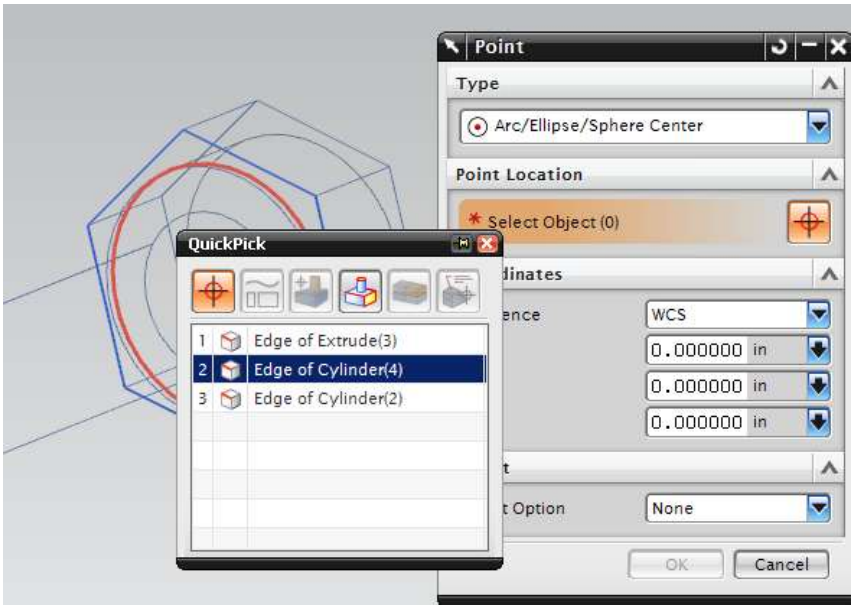
Remember to select the *Arc/Ellipse/Sphere Center* icon in the drop-down menu of *Type* in the ***Point Constructor*** window and select the top face of the cylinder with diameter of 0.387.

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



We will now use the feature operation ***Intersect***.

- Choose **INSERT → DESIGN FEATURE → SPHERE**
- Choose **DIAMETER, CENTER**
- On the **Point Constructor** window, choose the **Center** icon
- Select the bottom of the last cylinder drawn, which is inside the hexagon head and has a diameter of **0.387** inches and a height of **0.1875** inches as shown below



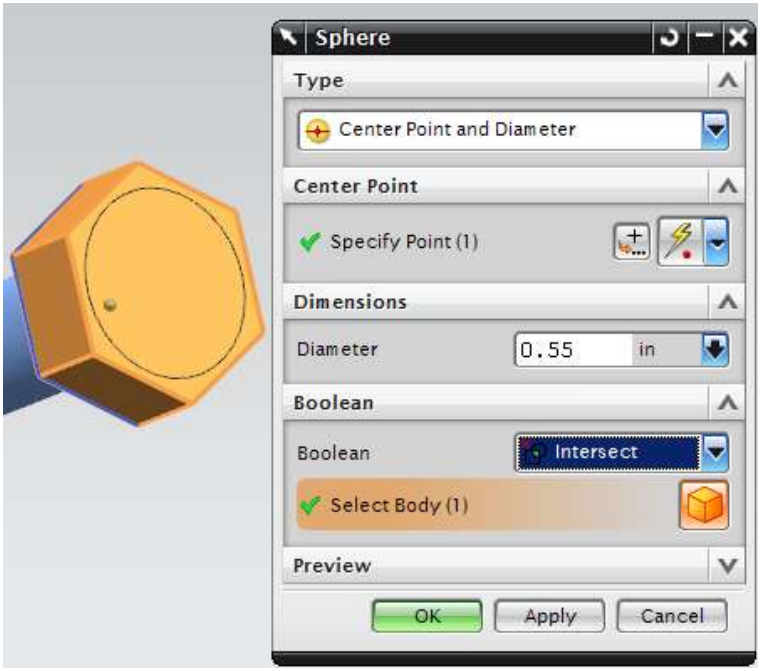
- Click **OK**
- Give 0.55 as the diameter

This will give take you the next Dialog box which will ask you to choose the Boolean operation to be performed.

- Choose **INTERSECT**

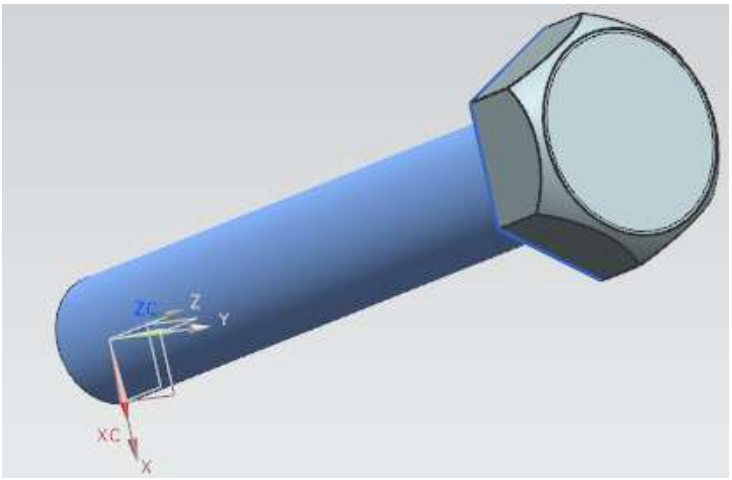
It will ask you to select the target solid

- Choose the hexagonal head as shown on right



- Click **OK**
- Click **CANCEL**

This will give you the hexagonal bolt as shown below.



*** NOTE: Take care when creating the different features (three cylinders, extrusion of hexagon), the Boolean dialog box has the value *"NONE"***

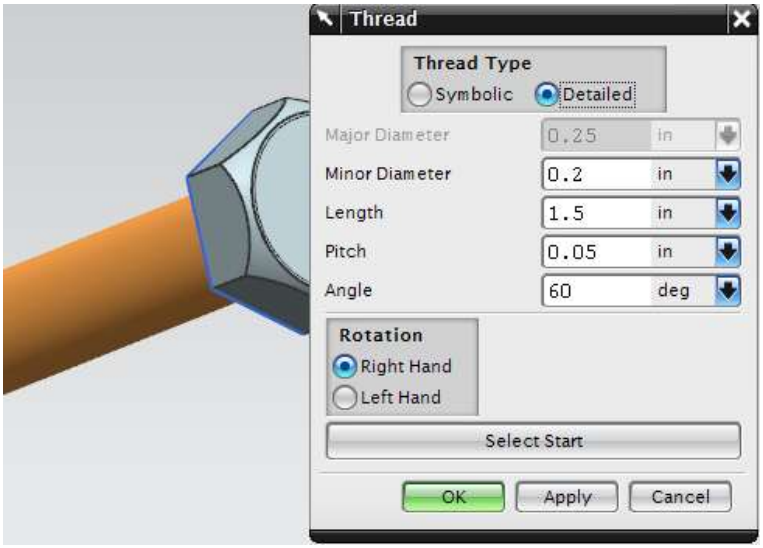
Now we will add threading to the hexagonal bolt.

- Choose **INSERT → DESIGN FEATURE → THREAD**

Here you will see the threading dialog box as shown below.

There are two main options in Threading:
1) Symbolic and 2) Detailed.

- Click on the **DETAILED** radio button



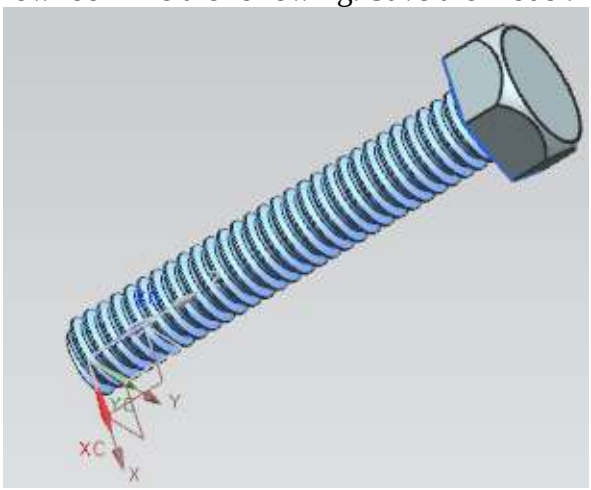
- Keep the thread hand to be **RIGHT HAND**

- Click on the bolt shaft, the long cylinder below the hexagon head

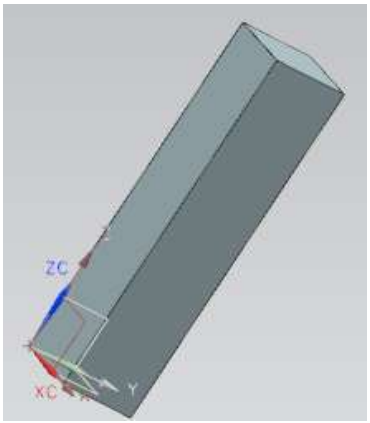
Once the shaft is selected, all the values will be displayed in the ***Thread*** dialog box. Keep all these default values.

- Click **OK**

The hexagon bolt should now look like the following. Save the model.



4.3.2 Model an L-Bar

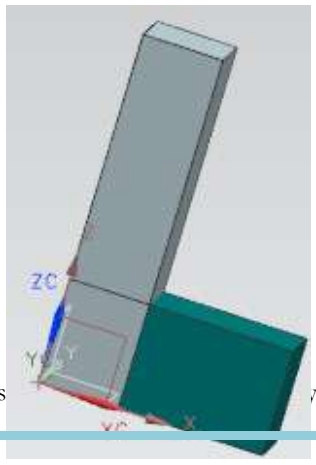


Here we will make use of some ***Feature Operations*** such as **Edge Blend**, **Chamfer**, and **Subtract**.

- Create a new file and save it as **Arborpress_L-bar**

- Choose **INSERT → DESIGN FEATURE → BLOCK**

- Create a block with the following dimensions.
  Length = **65** inches
  Width = **65** inches
  Height = **285** inches



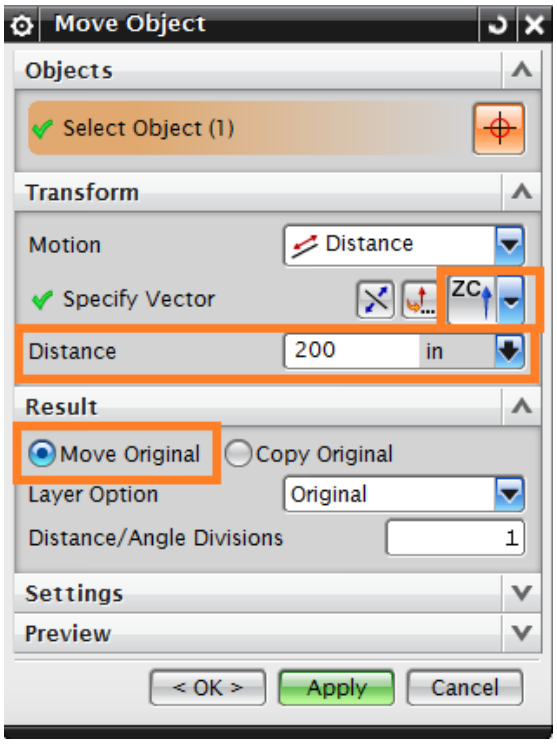
- Create the block at the **origin**

- Create a second block also placed at the origin with the following dimensions.
  Length = **182** inches
  Width = **65** inches

Height = **85** inches

We have to move the second block to the top of the first block.

- Click **EDIT → MOVE OBJECT**

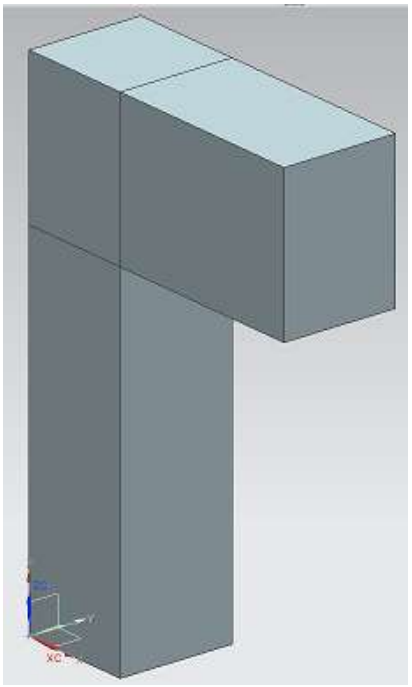


- Select the second block (green) that you inserted which is longer in the **XC**-direction
- Click **OK**
- Choose the *Motion* as **DISTANCE**
- Select the positive **ZC** in the *Specify Vector* dialog
- Enter **200** as the *Distance* value

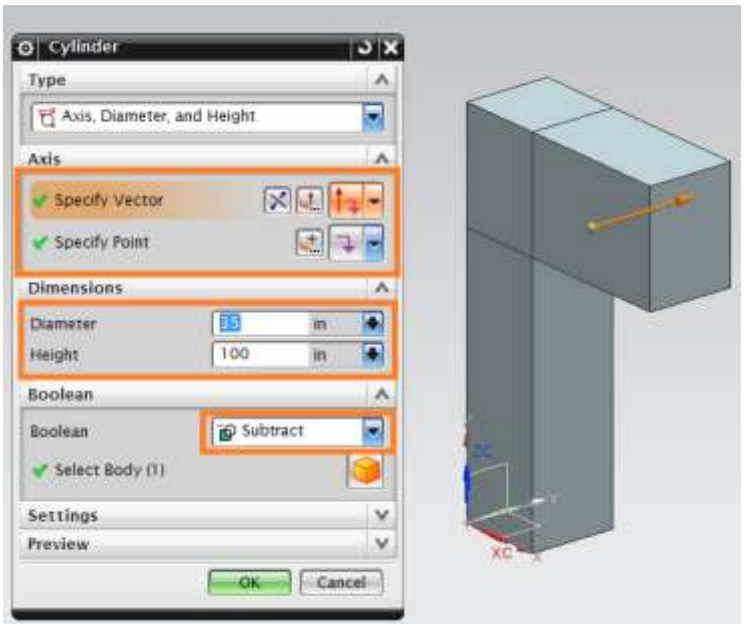
Make sure that *Move Original* radio button is checked.

- Click **OK**
- Click **MOVE** and then **CANCEL** on the next window so that the operation is not repeated

After transformation, it will look like the following.



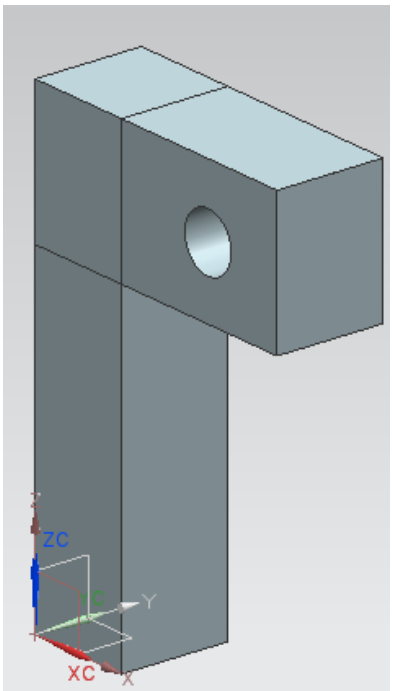
Now we will create a hole. There are many ways to create a hole. We will do so by first creating a cylinder and then using the ***Subtract*** function.



- Choose **INSERT → DESIGN FEATURE → CYLINDER**
- On the **Vector Constructor** window, select the **YC** Axis icon
- In the **Point Constructor** dialog box, enter the following values

Axes	**XC**	**YC**	**ZC**
Dimension	130	-5	242

- The cylinder should have the following dimensions.
  Diameter = **35** inches
  Height = **100** inches
- Under the **Boolean** drop-down window, choose **SUBTRACT**
- Select the horizontal block at the top as shown in the figure on the right side

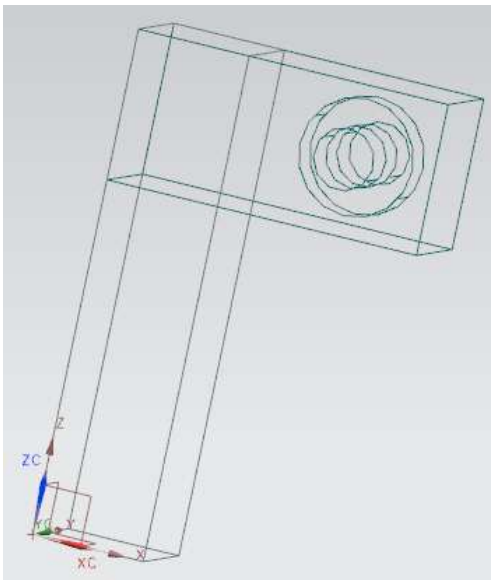


The hole should look like the one in the figure. Save your model.

Now we will create another cylinder and subtract it from the upper block.

- The cylinder should be pointing in the positive Y-direction set at the following point: **XC = 130; YC = 22.5** and **ZC = 242**
- The cylinder should have the following dimensions.
  Diameter = **66** inches
  Height = **20** inches
- **Subtract** this cylinder from the same block as before using the **Boolean** drop-down menu

The wireframe model will be seen as shown below.



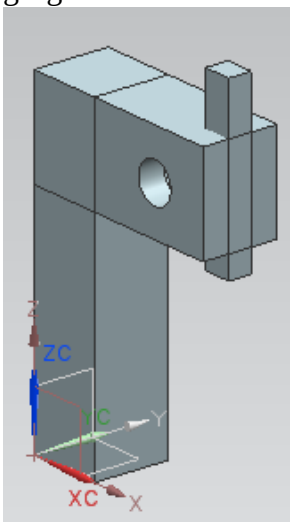
Now we will create a block.

- Choose **INSERT → DESIGN FEATURE → BLOCK**

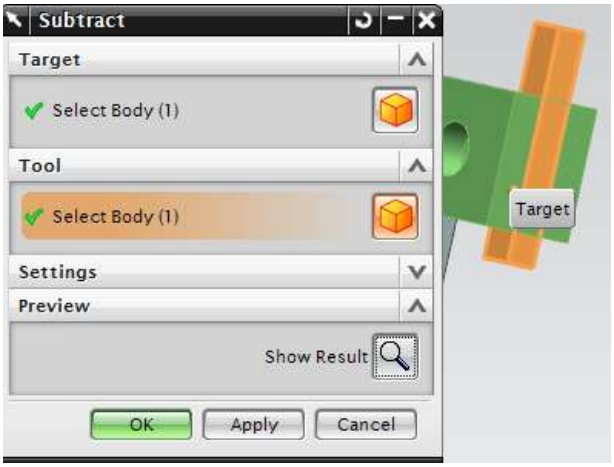
- Create a block with the following dimensions.
  Length = **25** inches
  Width = **20** inches
  Height = **150** inches
- Click on the **Point Constructor** icon in the ***Block*** window and enter the following values

Axes	**XC**	**YC**	**ZC**
Values	157	22.5	180

The model will look like the following figure.

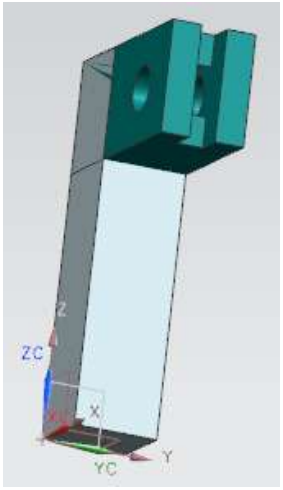


Now we will subtract this block from the block with the hole.



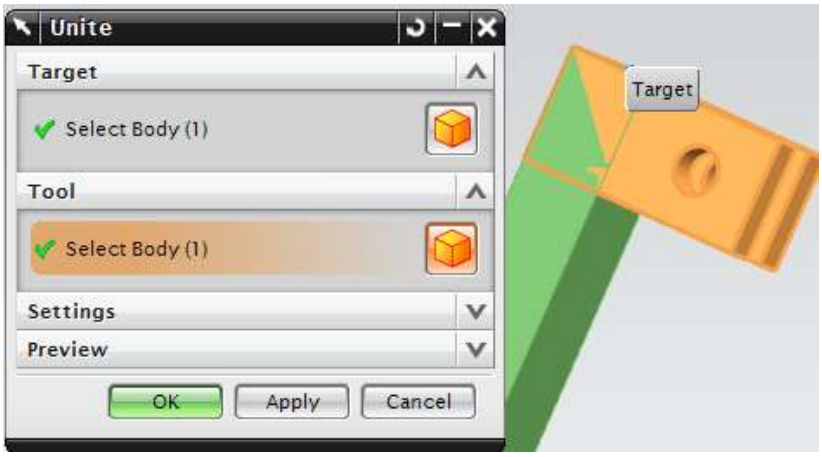
- Choose **INSERT → COMBINE BODIES → SUBTRACT**
- Click on the block with the two holes (green) as the ***Target***
- Select the newly created block (brown) as ***Tool***
- Click **OK**

The model will be seen as shown below.



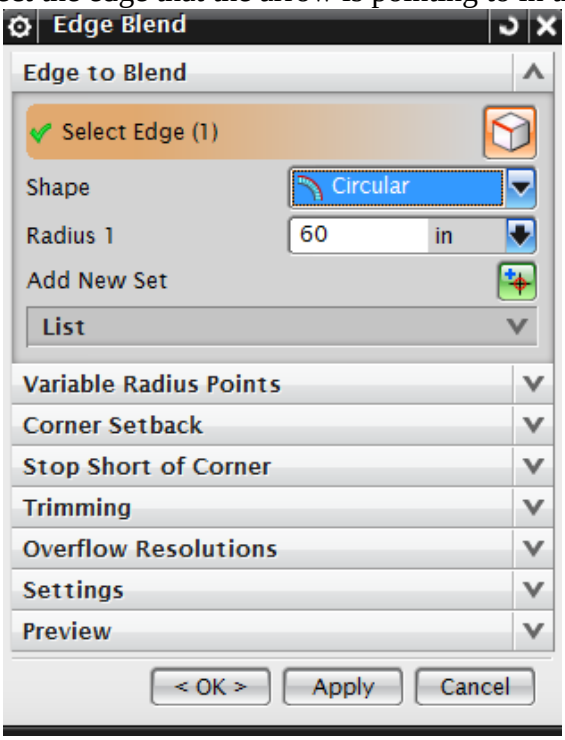
Now we will use the ***Blend*** function in the feature operations. In order to do so, we must first unite the two blocks.

- Choose **INSERT → COMBINE BODIES → UNITE**
- Click on the two blocks and click **OK**



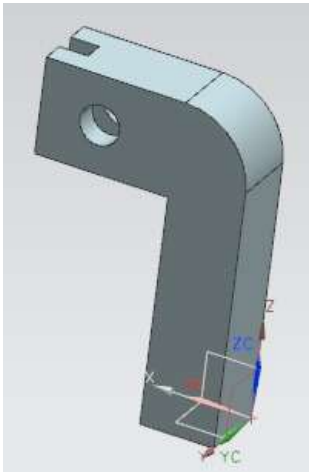
The two blocks are now combined into one solid model.

- Choose **INSERT → DETAIL FEATURE → EDGE BLEND**
- Change the **Radius** to **60**
- Select the edge that the arrow is pointing to in the figure



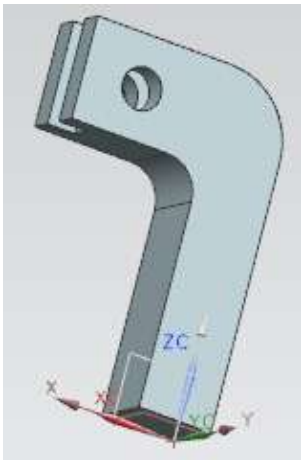
- Click **OK**

The blend will look as shown below.



- Repeat the same procedure to ***Blend*** the inner edge of the blocks. This time, the **Radius** should be changed to **30**

The blended figure is shown below. Remember to save the model.



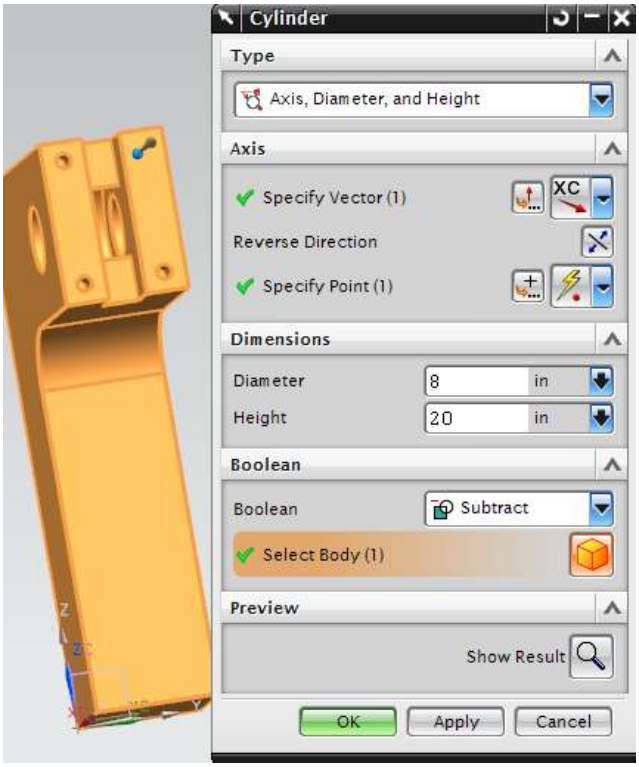
We will now make four holes in the model. You can create these holes by using the ***Hole*** option as illustrated in Chapter 3; however, to practice using ***Feature Operations***, we will subtract cylinders from the block.

- Insert four cylinders individually. They should be pointing in the **positive XC**-direction and have the following dimensions.
  Diameter = **8** inches
  Height = **20** inches

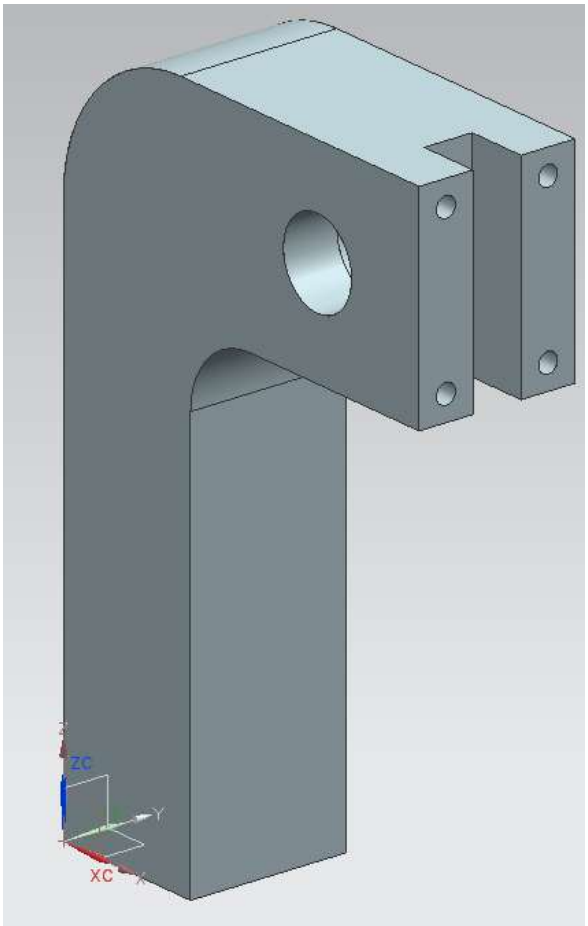
They should be constructed in the **XC**-direction at the following point coordinates.

	1	**2**	**3**	**4**
X	162	162	162	162
Y	11.25	11.25	53.75	53.75
Z	210	275	210	275

- **SUBTRACT** these cylinders from the block in the ***Boolean*** dialog box



The block with holes looks like as shown in figure below

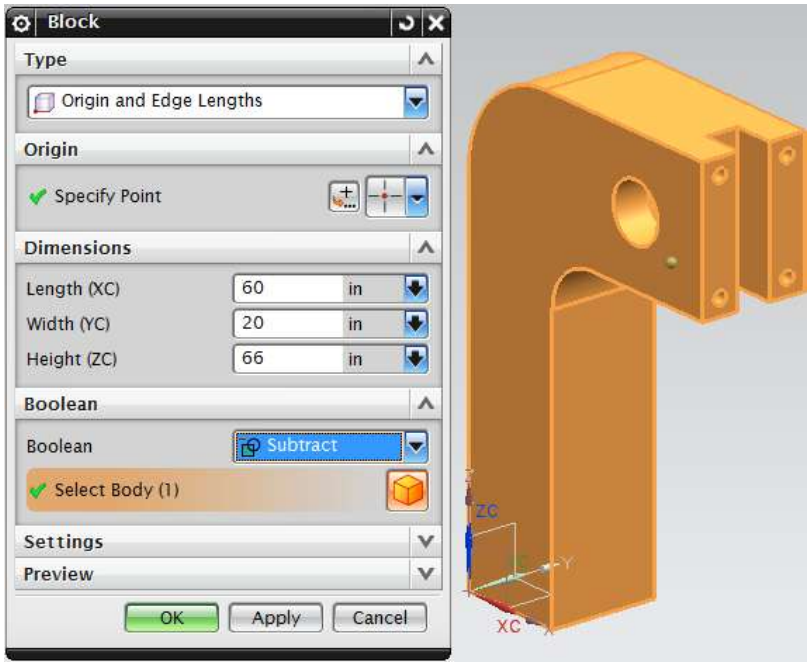


The last operation on this model is to create a block and subtract it from the top block.

- Create a ***Block*** with the following dimensions.
  Length = **60** inches
  Width = **20** inches
  Height = **66** inches

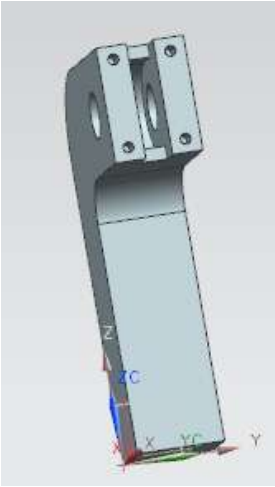
- Enter the following values in the **Point Constructor** dialog by clicking the icon

Axes	**XC**	**YC**	**ZC**
Dimension	130	22.5	209.5



- After creating the block, **subtract** this block from the block at the top by first selecting the original block and then clicking on the newly created block.

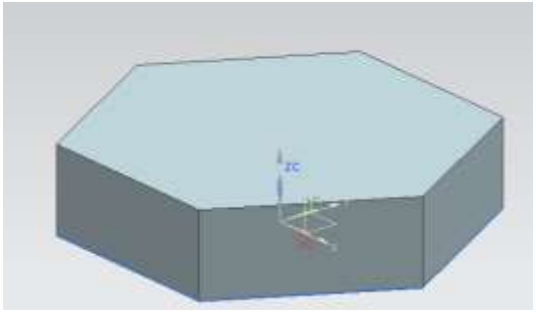
The final figure will look like this. Save and close the file.



4.3.3 Model a Hexagonal Nut

- Create a new file and save it as **Impeller_hexa-nut.prt**
- Choose **INSERT → CURVE → POLYGON**
- Input number of sides to be **6**
- Create a hexagon with the option **SIDE OF POLYGON** and each side measuring **0.28685** inches and constructed at the ***Origin***.
- Choose **INSERT → DESIGN FEATURE → EXTRUDE**
- Click on all six lines of the hexagon to choose the surface that is required to be extruded
- Enter the End Distance as **0.125** inches

The figure of the model is shown below.

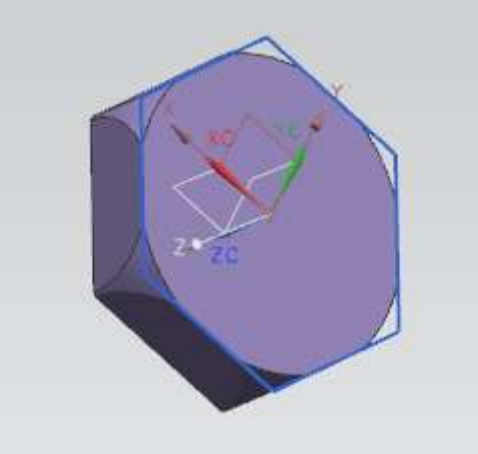


- Choose **INSERT → DESIGN FEATURE → SPHERE**
- Choose **CENTER, DIAMETER**
- Enter the diameter value **0.57** inches
- Enter the center point location in the **Point Constructor** window as follows

Axes	XC	YC	ZC
Dimension	0.0	0.0	0.125

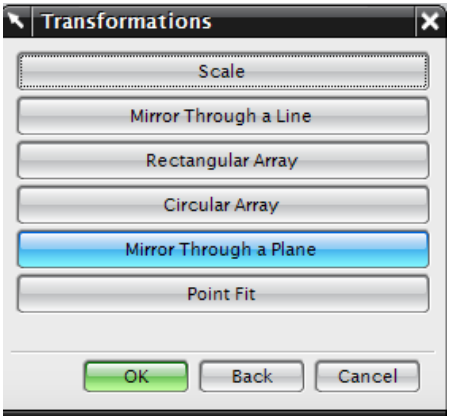
- In the ***Boolean*** operations dialog box select **INTERSECT**

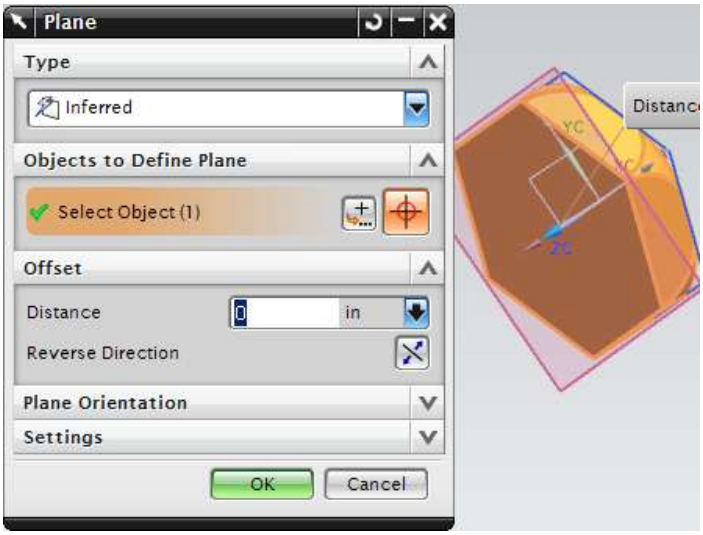
The model will look like the following.

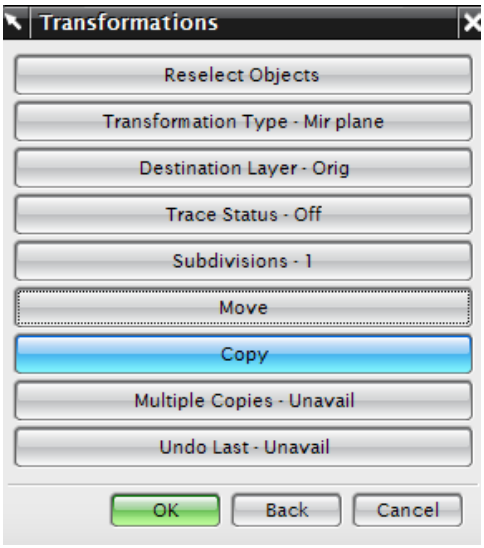


We will now use a ***Mirror*** command.

- Choose **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

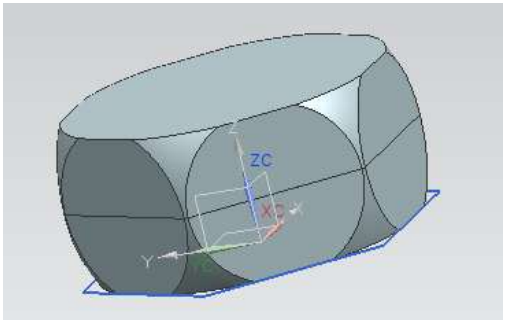




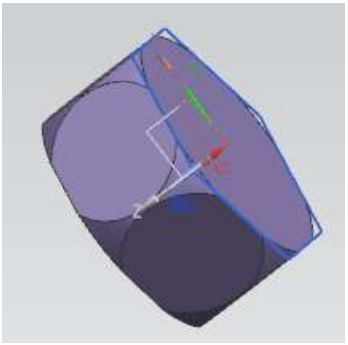


- Click on **OK**
- Click on **COPY**
- Click **CANCEL**

You will get the following model.

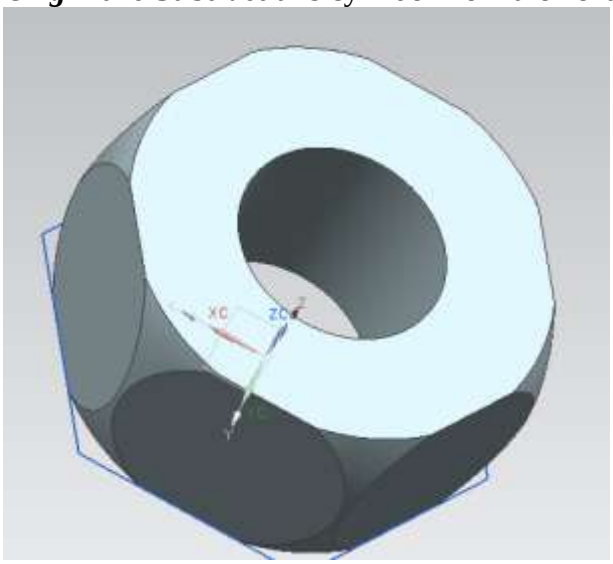


- Choose **INSERT → COMBINE BODIES → UNITE**
- Select the two halves and **Unite** them



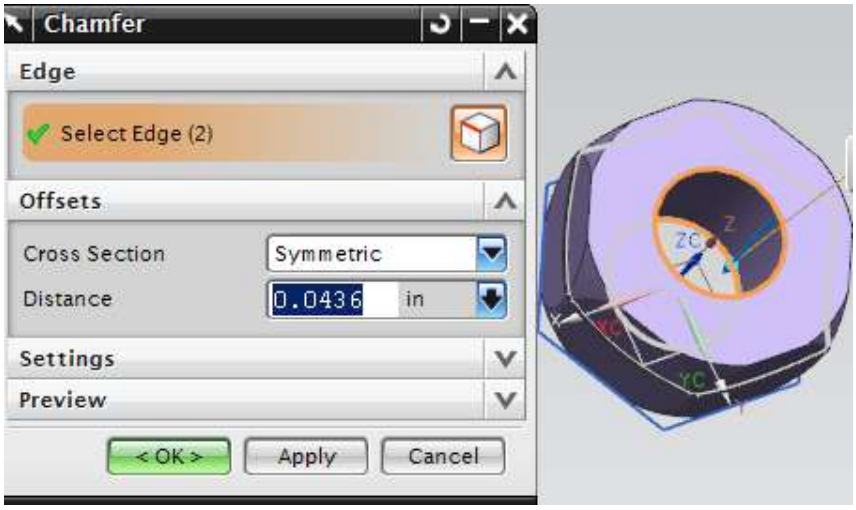
- Insert a ***Cylinder*** with the vector pointing in the **ZC**-Direction and with the following dimensions.
  Diameter = **0.25** inches
  Height = **1** inch

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



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

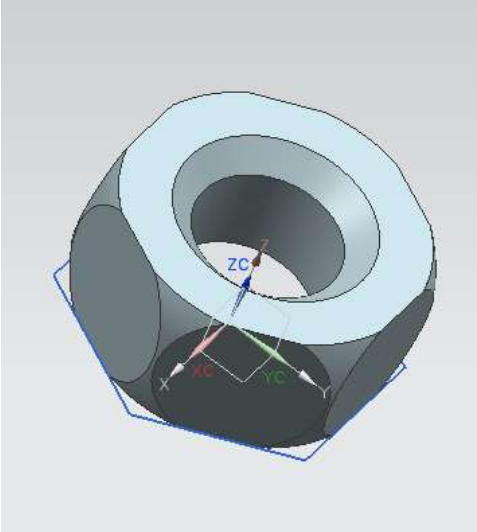
- Choose **INSERT → DETAIL FEATURE → CHAMFER**



- Select the two inner edges as shown and click **OK**

- Enter the **Chamfer Offset Diameter** as **0.0436** inches and click **OK**

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

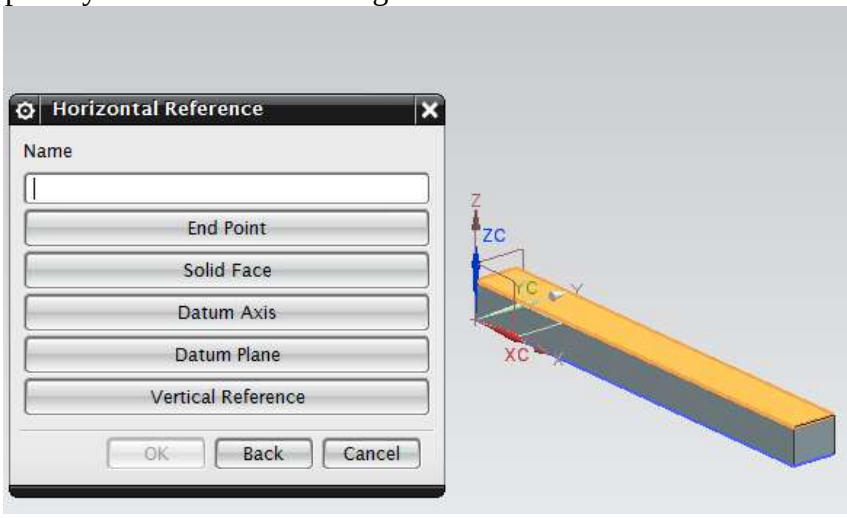


4.3.4 Model a Rack with Instances

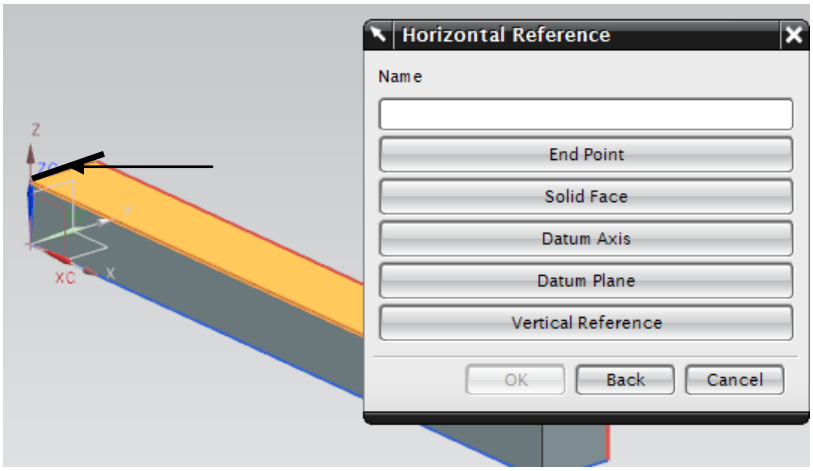
In this part, we will practice to create instances of a given object.

- Open the file **Arborpress_rack.prt**
- Choose **INSERT → DESIGN FEATURE → POCKET**
- Choose **RECTANGULAR** in the pop up window
- Click on the top surface of the rack as shown in the figure for the placement surface.

Keep an eye on the ***Cue Line***. It gives the next instruction.

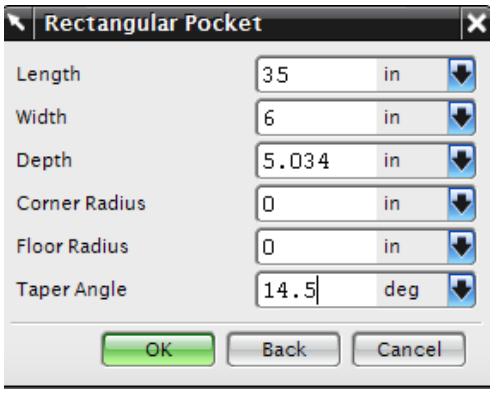


- Click on the edge as shown in the figure for the ***Horizontal Reference***



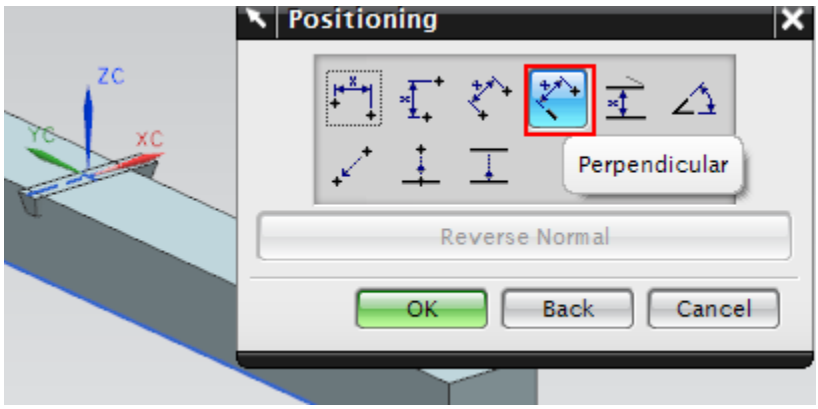
This will pop up the parameters window.

- Enter the values of parameters as shown in the figure and choose **OK**

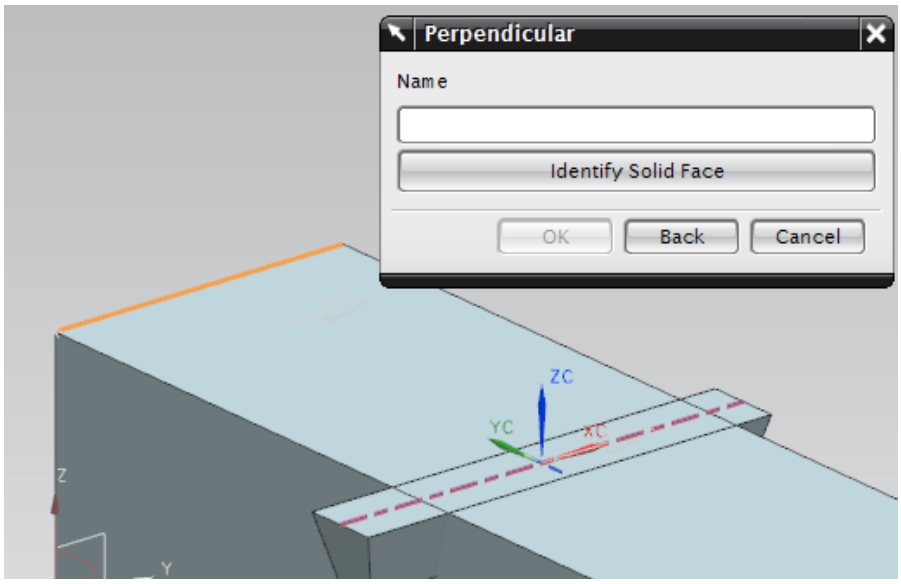


- Choose **Wireframe** option in the ***Display*** mode for more clarity

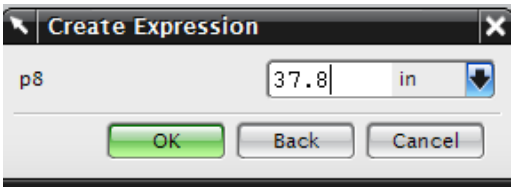
- When the **Positioning** window pops up, choose the **PERPENDICULAR** option



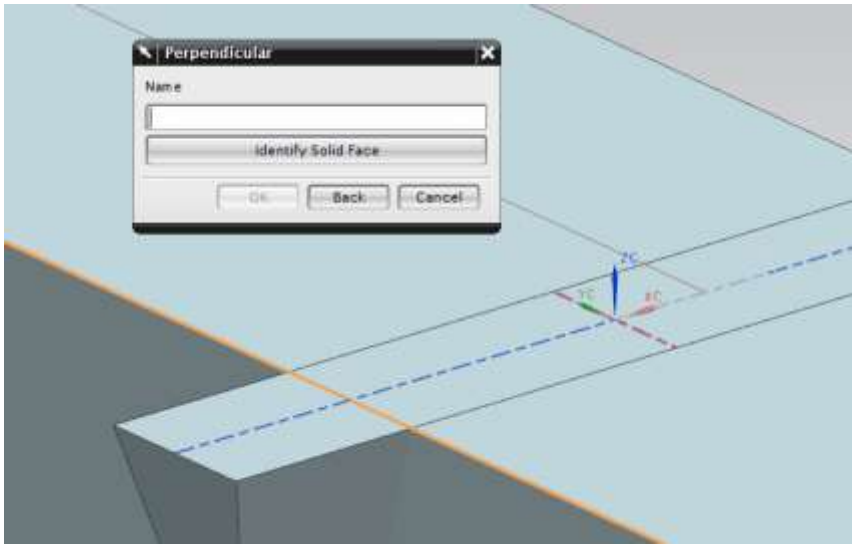
- Click on the edge on the solid and then click on the blue dotted line as shown below



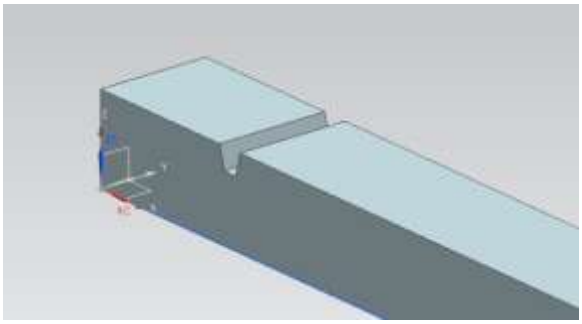
- Enter the expression value as **37.8** and Choose **OK**



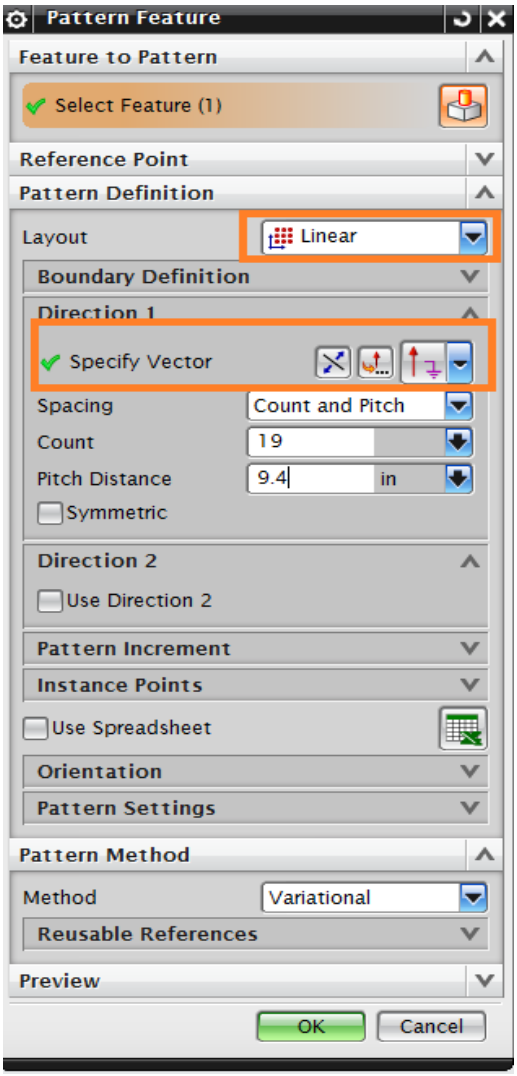
- Once again pick the **PERPENDICULAR** option and then choose the other set of the edges along the Y-Axis, as shown in the figure below.
  NOTE: Select the blue line along Y axis (the one perpendicular to the last blue line selected)



➢ Enter the expression value as **10** and choose **OK** twice. Choose **CANCEL**.
The model will now look as follows.



Let us create the instances of the slot as the teeth of the Rack to be meshed with pinion.



➢ Choose **INSERT → ASSOCIATIVE COPY→ PATTERN FEATURE**

➢ Click select feature. Now click on the rectangular pocket created.

➢ Select layout as linear

➢ Specify vector as positive XC direction

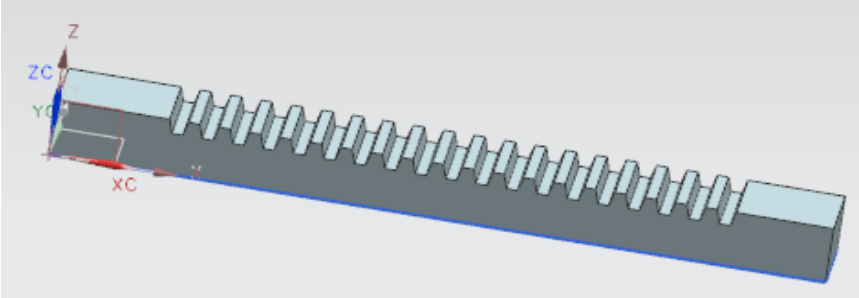
➢ Choose **COUNT AND PITCH** in spacing option and enter value for count as 19 and that for pitch distance as 9.4

➢ Choose **OK**

➢ Click **YES**

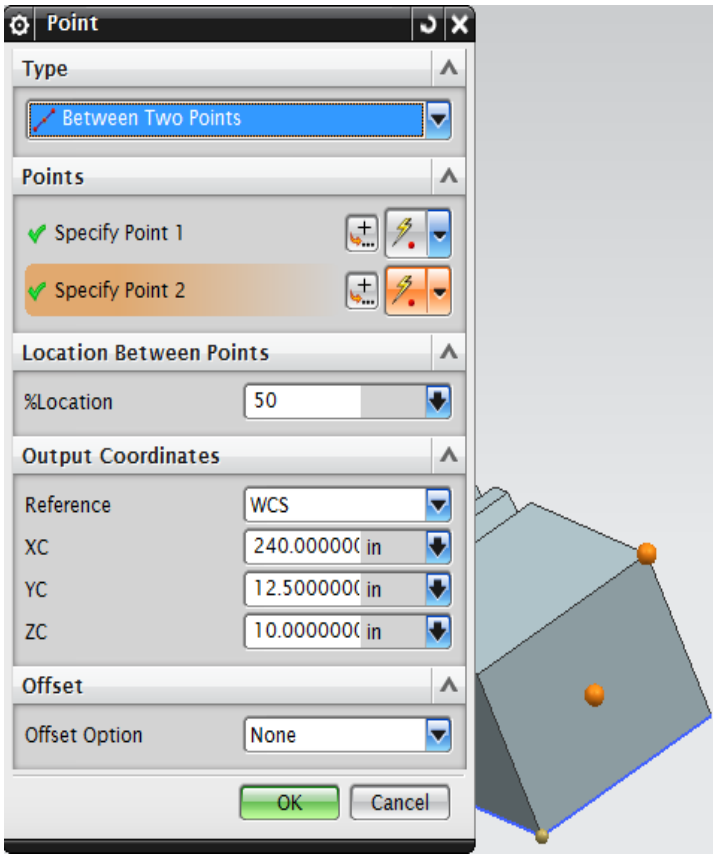
➢ Click **CANCEL**

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

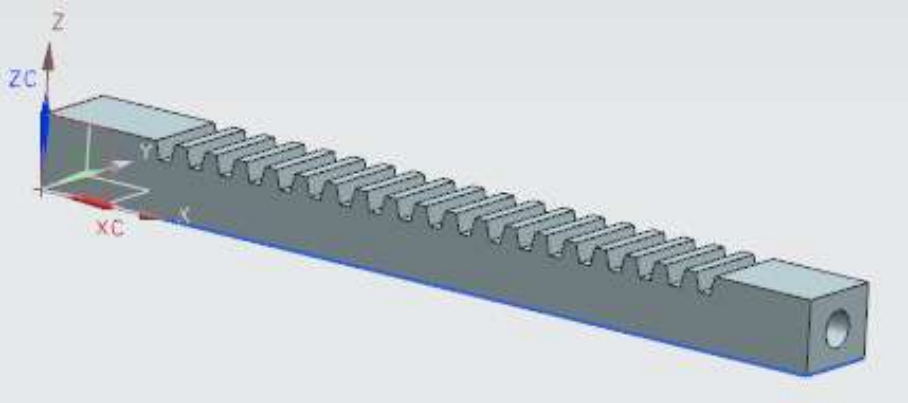


We will create a hole with **diameter 10** inches and **depth 20** inches at the center of the rectangular cross section.
To determine the center of the cross-section of the rectangular rack, we make use of the ***Snap Points***

- Choose **INSERT → DESIGN FEATURE → CYLINDER**
- Choose **–XC**-Direction to in the ***Specify Vector*** dialog box
- Click on the **POINT CONSTRUCTOR**
- In the ***Points*** dialog box select *Between Two Points* option and select the points as shown in the figure on the right.
- Click **OK**
- Enter the following values in the ***Dimension*** dialog box
  Diameter – **10** inches
  Height – **20** inches
- Choose ***Subtract*** in the ***Boolean*** dialog box.



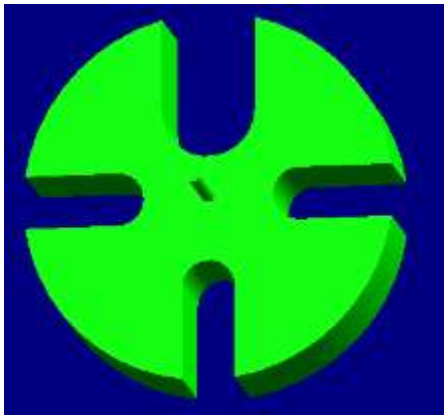
- The final model is shown below.



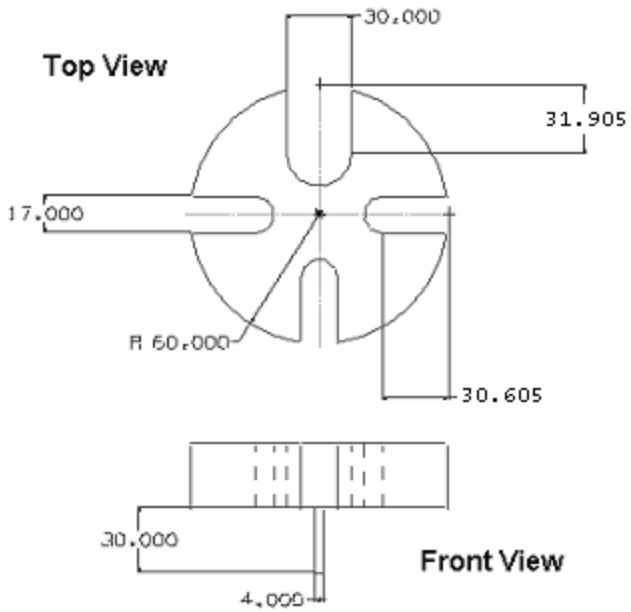
4.4 EXERCISE - MODEL A CIRCULAR BASE

As an exercise, model a circle base as shown below using the following dimensions:

Outer diameter = 120 inches
Distance of 3 small slots = 17 inches
Distance of the large slot = 30 inches
Diameter of the central rod = 4 inches and length = 30 inches
Length of slots may vary.



Top and Front view dimensions are shown in the figure below.

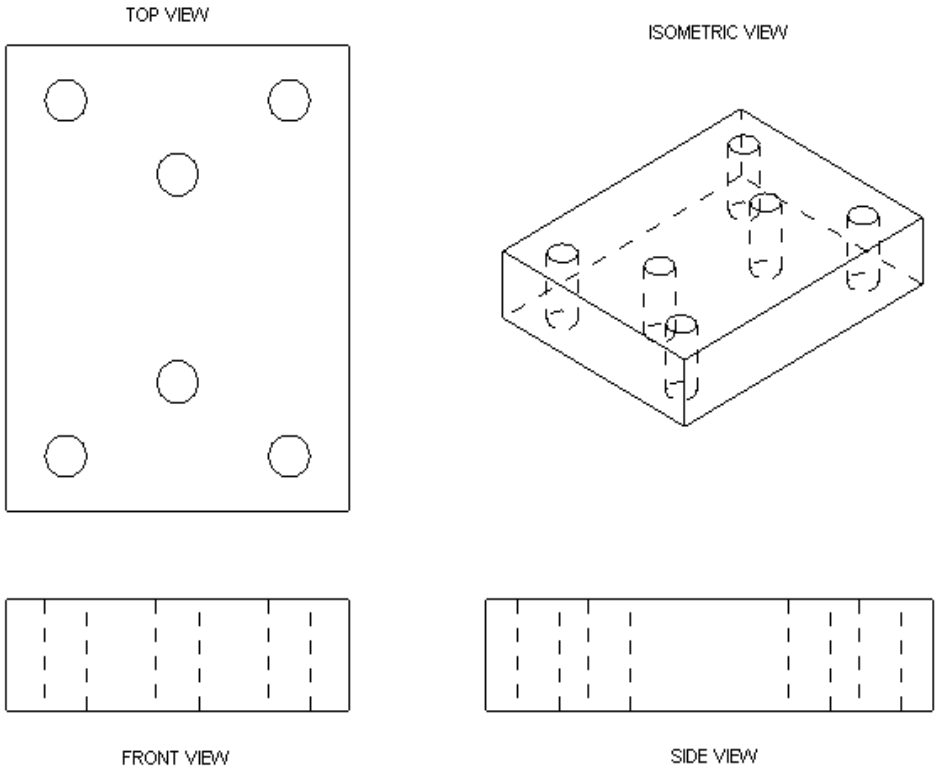

Top View
30.000
31.905
17.000
R 60.000
30.605
30.000
4.000
Front View

CHAPTER 5 – DRAFTING

The NX9.0 Drafting application lets you create drawings, views, geometry, dimensions, and drafting annotations necessary for the completion as well as understanding of an industrial drawing. 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 to 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 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.
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.

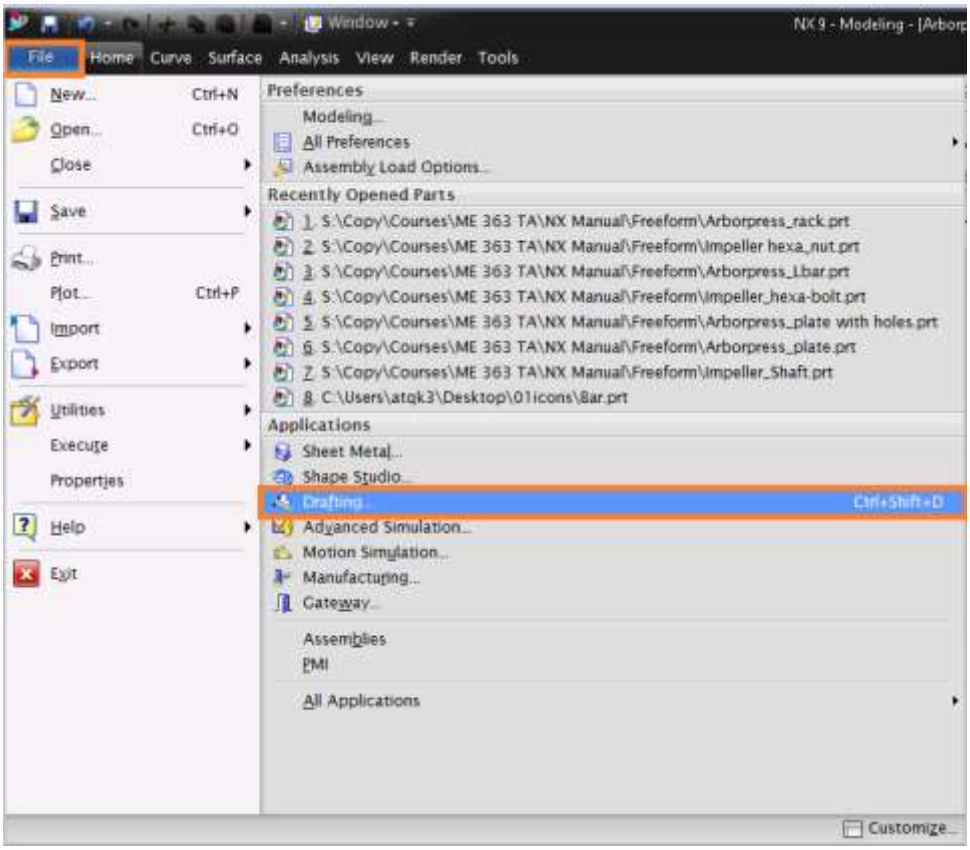
We will see how views are created and annotations are used and modified in the step-by-step examples.

5.2 DRAFTING OF MODELS

We will draft some models that have already been drawn. We will go through the drafting options step-by-step to make them easier to understand.

5.2.1 Drafting

- Open the file **Arborpress_rack.prt**
- From the NX9.0 Interface Choose **FILE → APPLICATIONS → DRAFTING** as shown



*** Note:** All other applications such as ***Modeling***, ***Manufacturing, Assembly,*** etc. can be opened in a similar fashion.

When you first open the ***Drafting*** Application, a window pops up asking for inputs like the *Template*, *Standard Size* or *Custom Size*, the units, and the angle of projection.

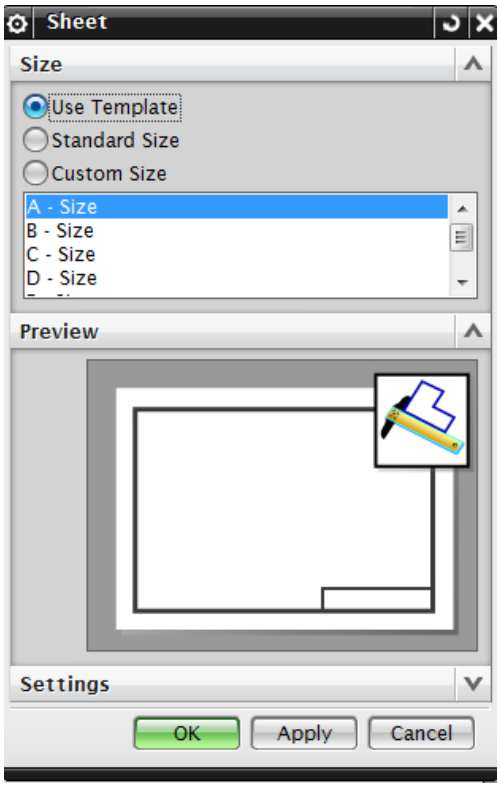
Size
Size allows you to choose the size of the Sheet. There are standard Templates that you can create for frequent use depending upon the Company Standards. There are several Standard sized Sheets available for you. You can also define a Custom sized sheet in case your drawings don't fit into a standard sized sheet.

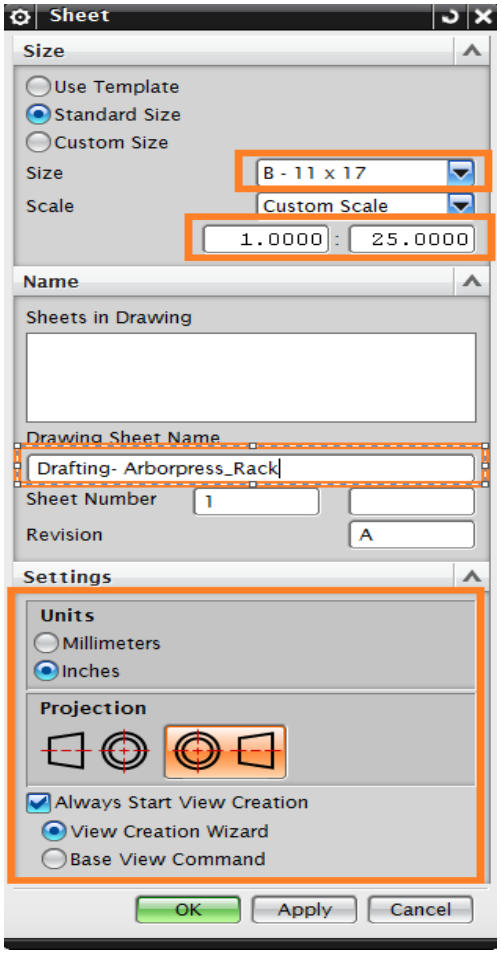
Preview
This shows the design of the Template

Units
Units follow the default units of the parent 3-D model. In case you are starting from the Drafting Application you need to choose the units here.

Projection
You can choose the projection method either first angle or third angle method.

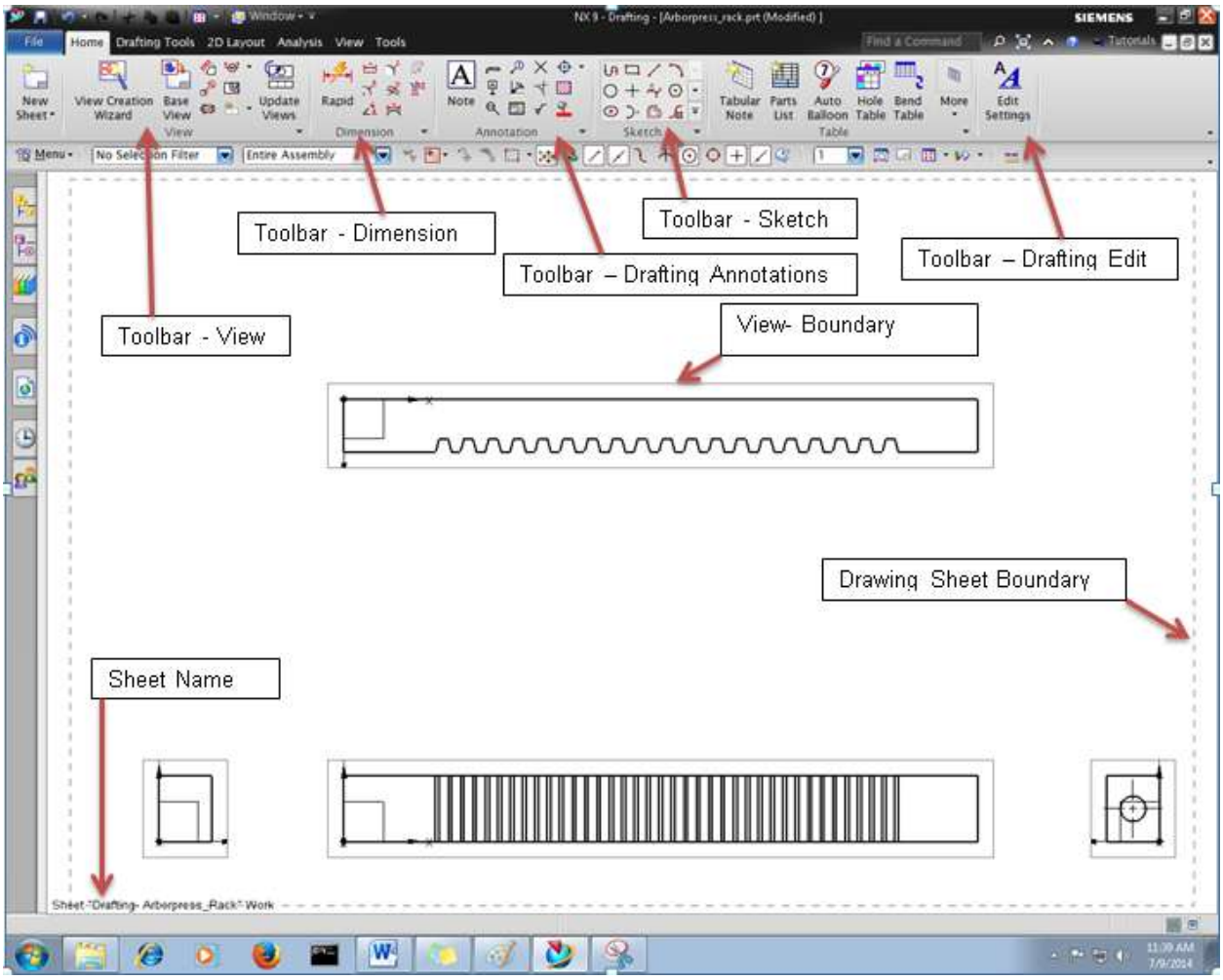
To start using the ***Drafting*** Application we will begin by creating a *Standard Sized* sheet



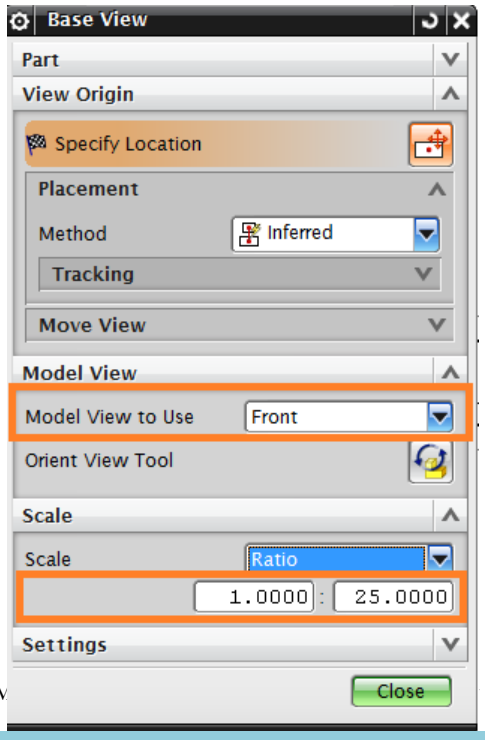


- Click on the ***Standard Size*** radio button.
- In the drop-down menu on the ***Size*** window, select sheet B, which has dimensions **11 x 17**. Then change the **scale** to **1:25** by using the drop-down menu and choosing the **Custom Scale** under the ***Scale*.**
- Click **OK**

This will open the drafting option and the following screen will be seen as below. Let us first look at the Drafting Application Interface.



After this you will see another dialog box pops-up which will define the **Base View** and its location. If you do not see the figure on right then



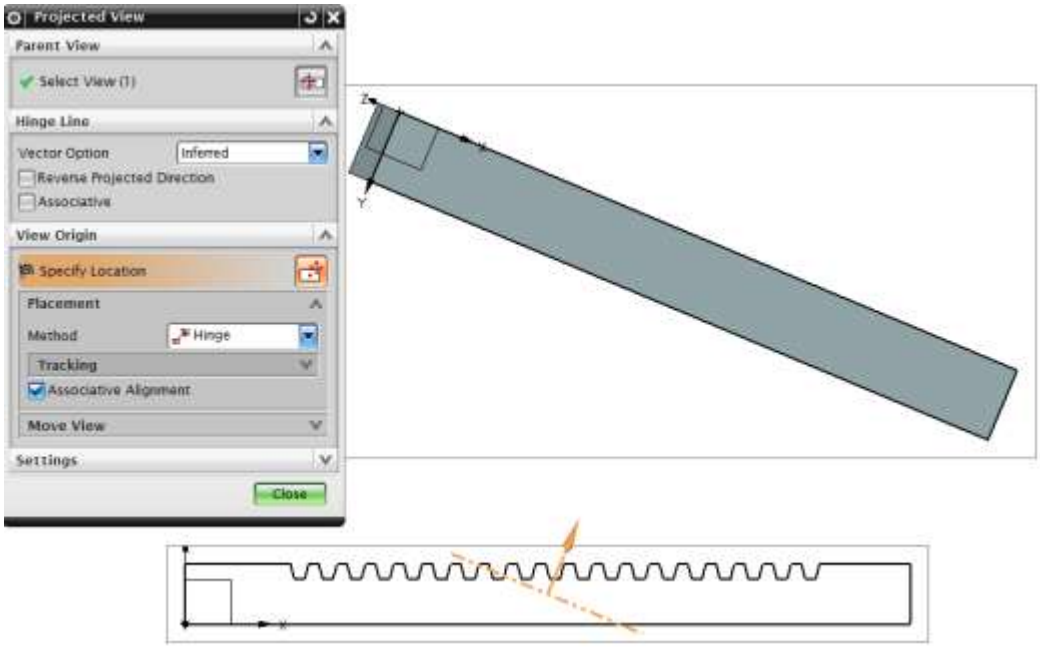
- Choose **INSERT → VIEW → BASE** or click on **BASE VIEW on TOOLBAR-VIEW**

You can find a Dialog box with the options of the **View** and the **Scale** of the view, as shown in the figure on your screen along with a floating drawing of the object.

- Choose the **View** to be **FRONT**

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

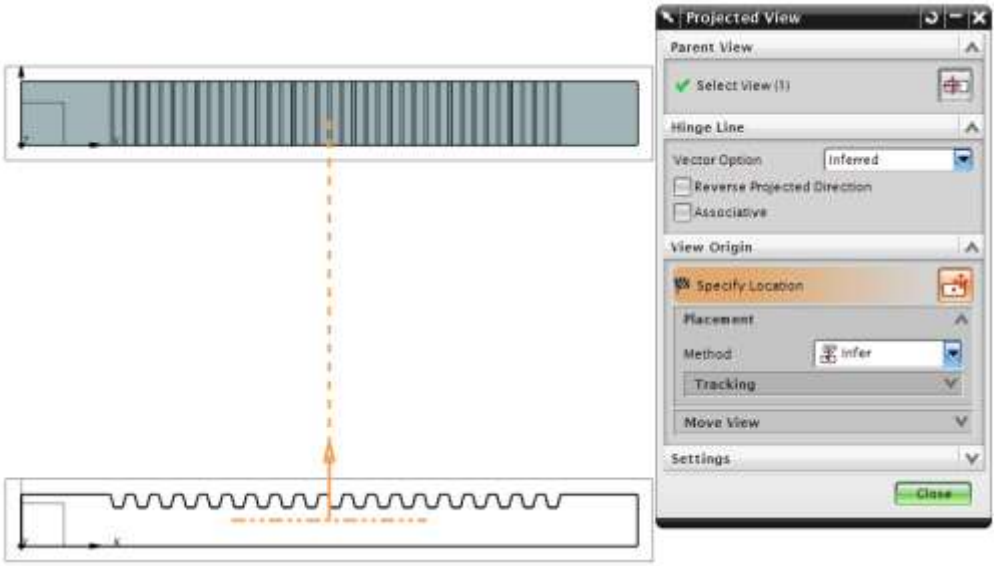
Once you set the **Front** view another dialog box will pop-up asking you to set the other views at any location on the screen within the ***Sheet Boundary***.



You can find the views by changing the cursor around the first view (FRONT VIEW). The following are some snap shots of the views seen at different location of the mouse cursor. If you want to add any orthographic views after closing this file or changing to other command modes

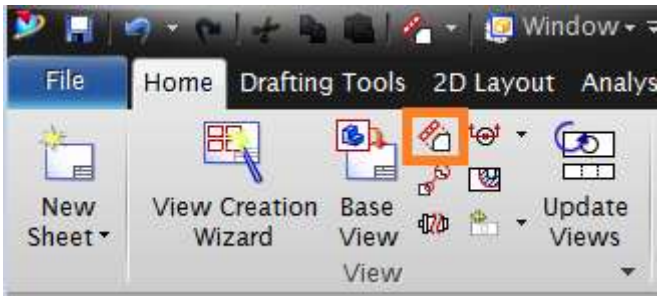
➢ Choose **INSERT → VIEW → PROJECTED VIEW**

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

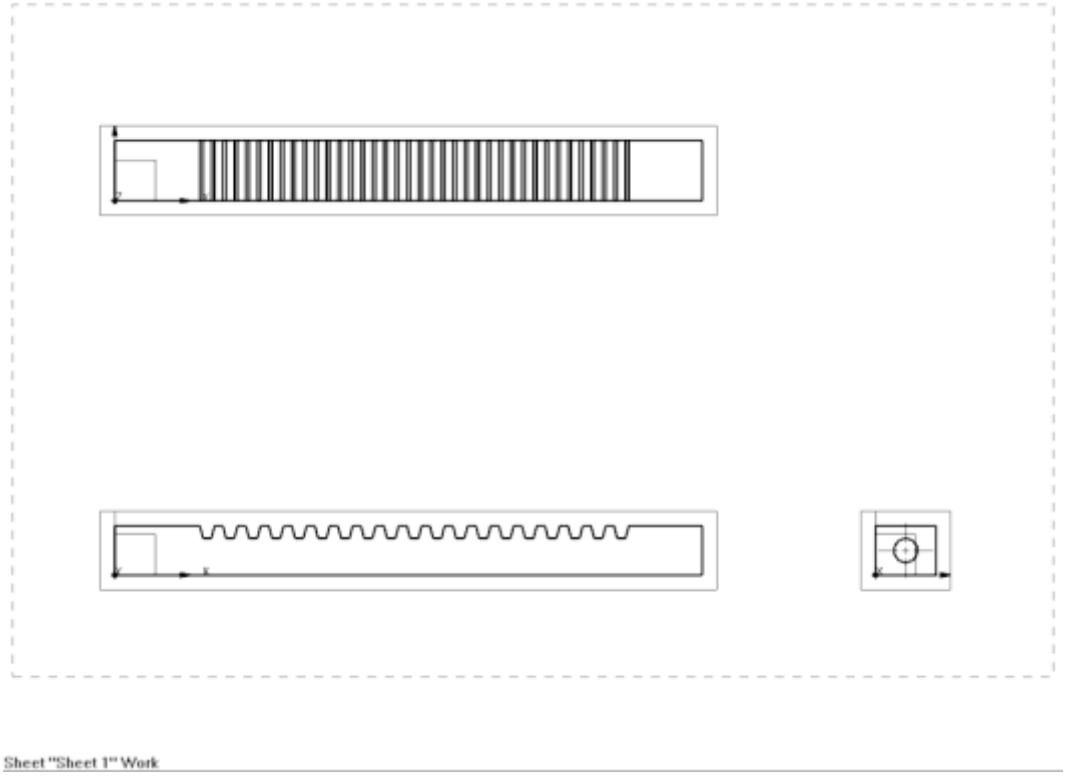


- Keep on Clicking in the Sheet to create the side view.

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



- Move the cursor to the right side and click there to get the right-side view



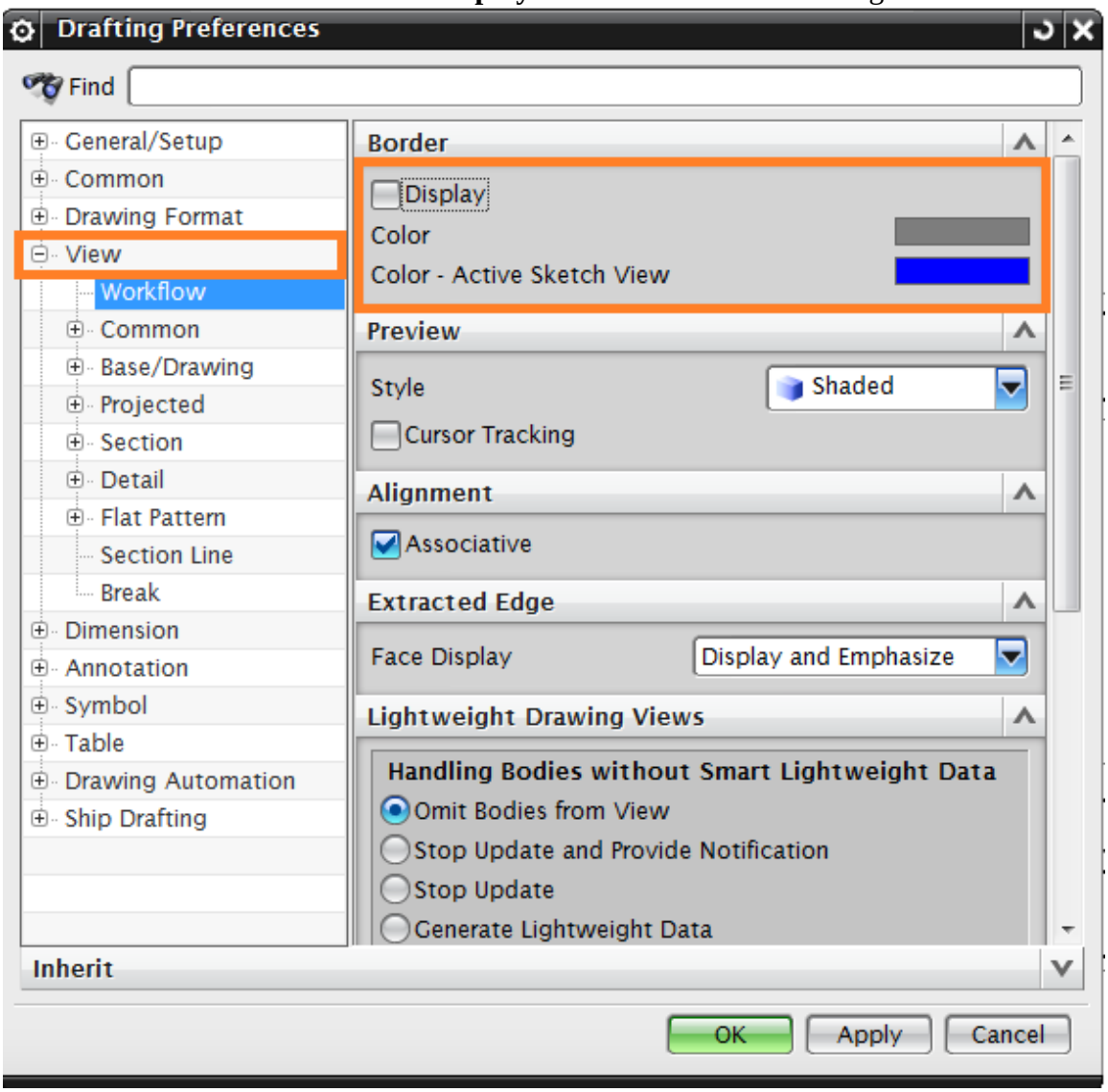
- Click **Close** on the ***Projected View*** dialog box or Press **<Esc>** key on the Keyboard to get out of the **View** creation.

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

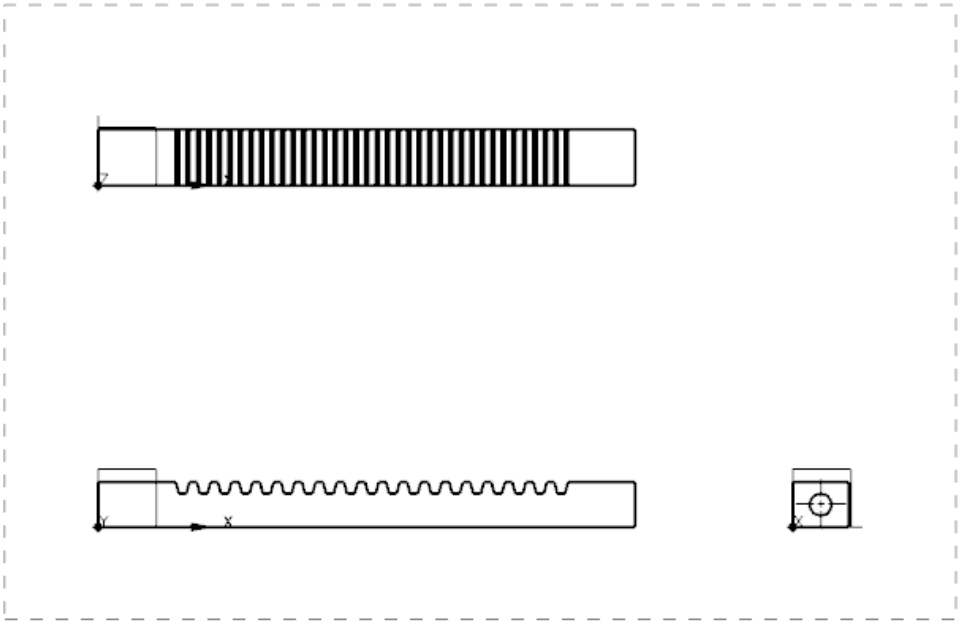
➢ Choose **MENU→ PREFERENCES→ DRAFTING**

The Drafting Preferences window will pop up.

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



Now you can find the drawing views without borders as shown below.



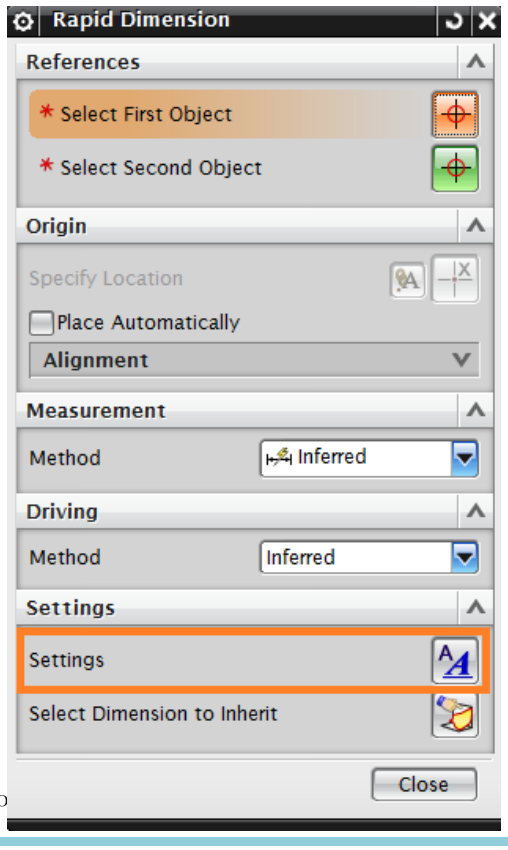
5.2.2 Dimensioning

Now we have to create the dimensions for these views. The dimensions can be inserted by either of the two ways as described below:

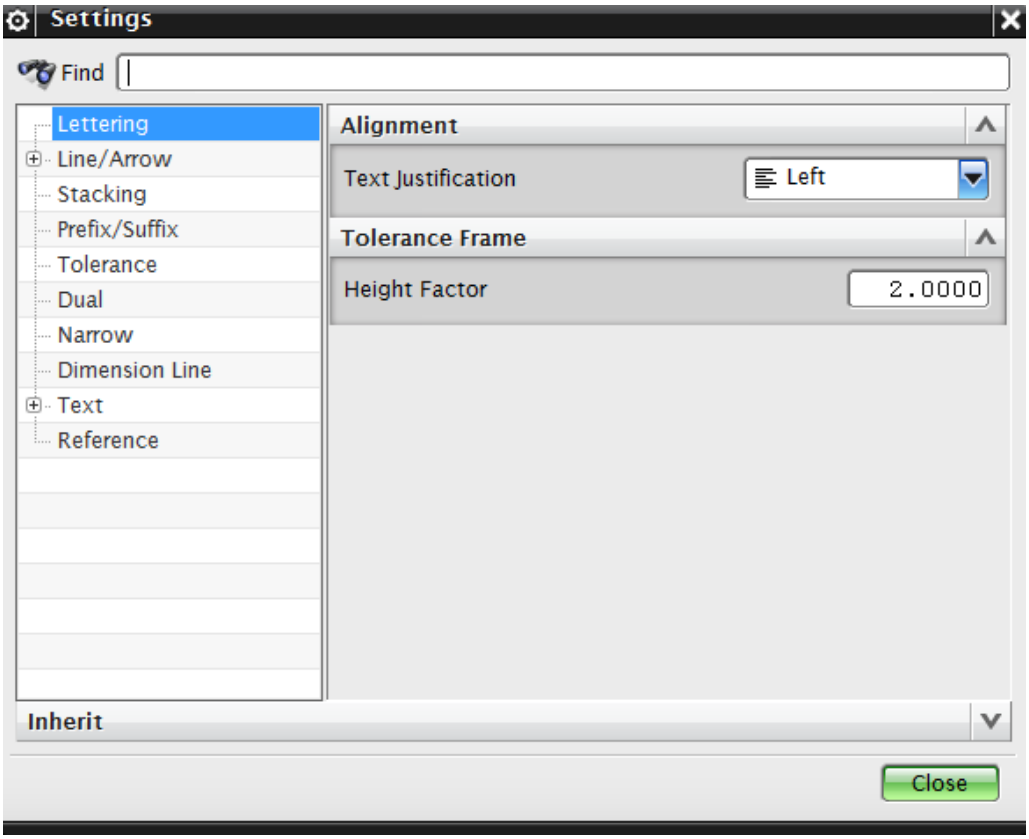
1) Choose **MENU** ⟶ **INSERT** ⟶ **DIMENSION**
   OR
2) Click on the ***Dimension Toolbar*** as shown in the following figure

➢ Choose **INSERT → DIMENSION → RAPID**

The following two option boxes will pop up. The icons on this toolbar are helpful for changing the properties of the dimensions.



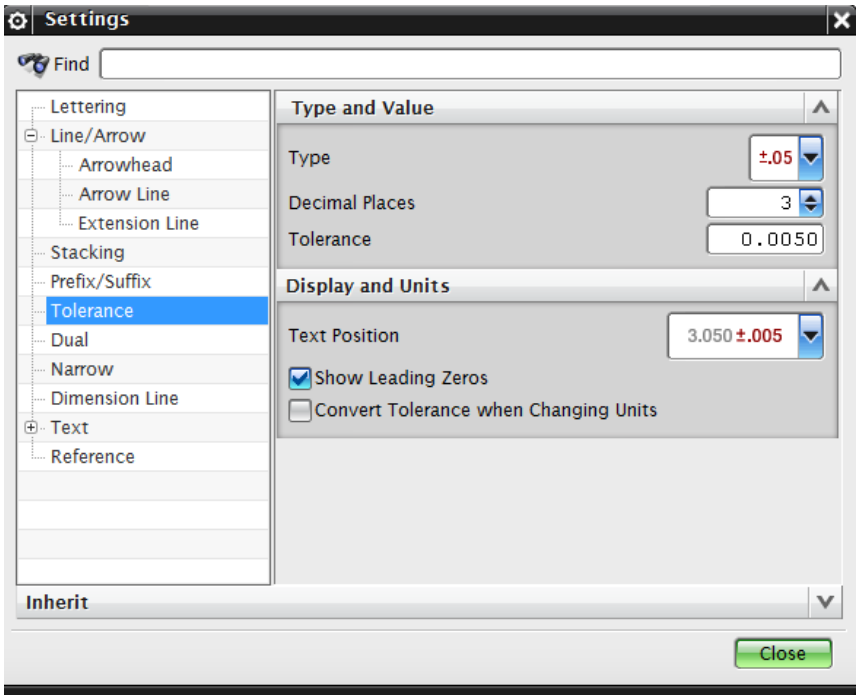
Now click on the settings button as shown in the figure. Here you will be able to modify the settings for dimensioning. A dialog appears as shown below.



The first list is for **Lettering.** This allows the user to justify and select the frame size.

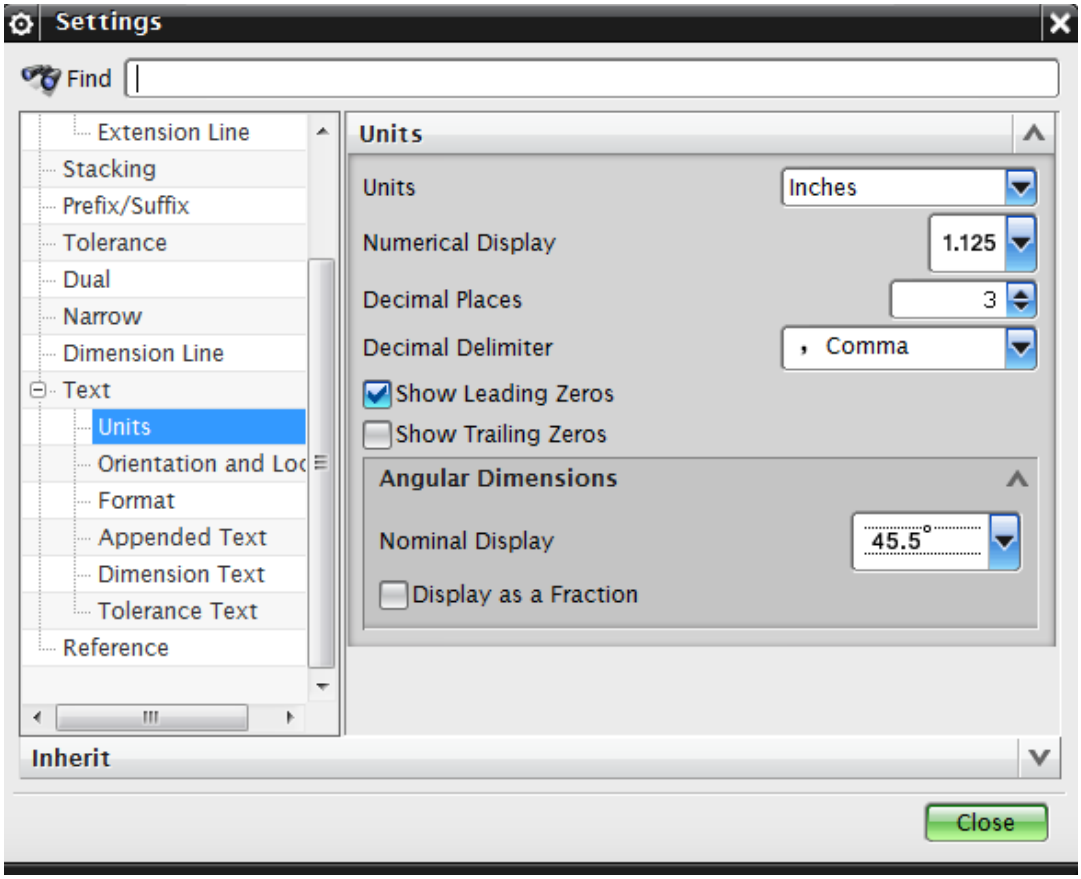
In the **Line/Arrow** section, you can vary the thickness of the arrow line, arrow head, angle format etc.

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 – such options can be modified here.

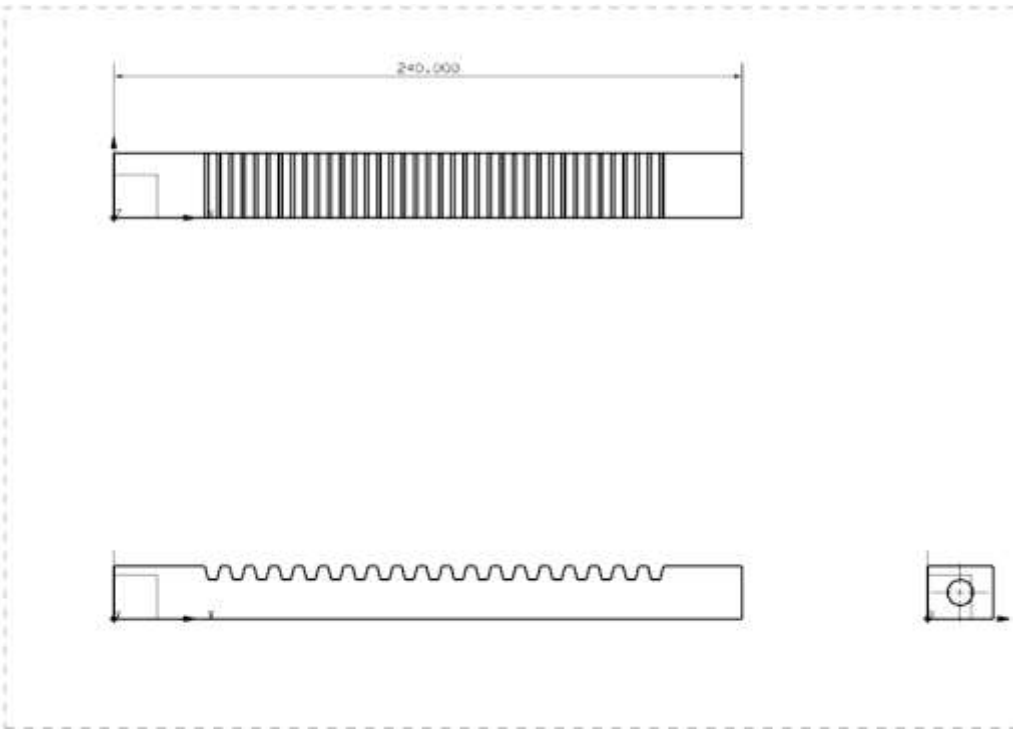
The next icon is the **Text** option, which you can use to edit the units, text style, font and other text related aspects.



- 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

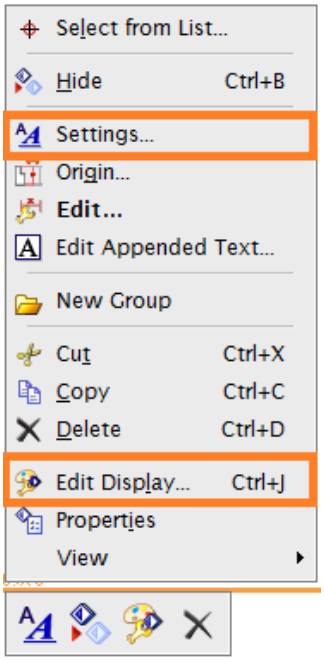
The dimension that represents the distance between these points will appear. You can put the location of the dimension by moving the mouse on the screen. Whenever you place your views in the Sheet please take into consideration that you will be placing the dimensions around it.

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

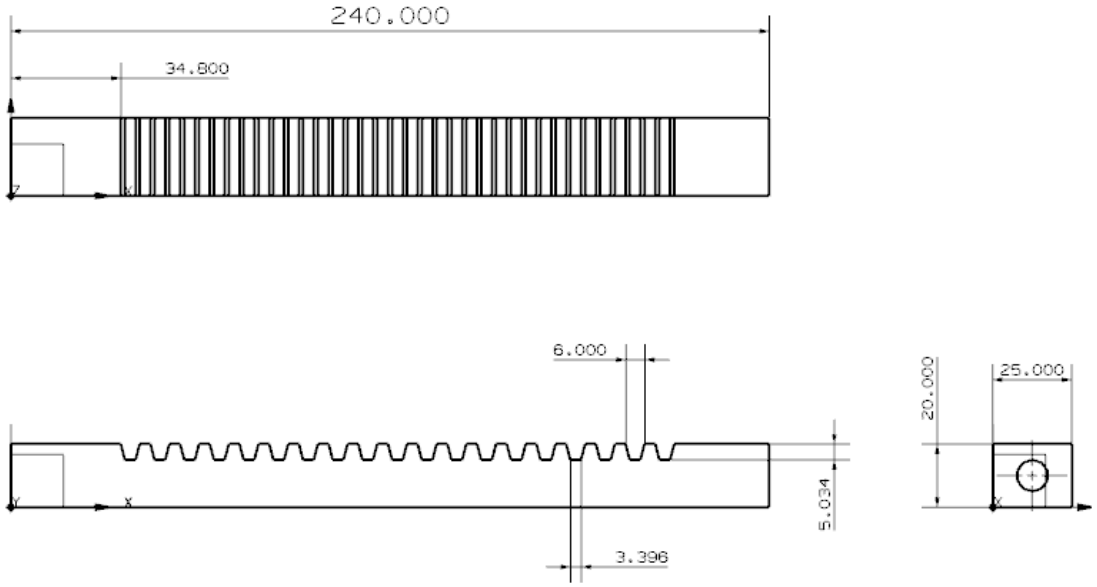


Even after creating the dimension, you can edit the properties of the dimensions.

- Right-click on the dimension you just created and Choose **SETTINGS/EDIT DISPLAY**
- You can modify font, color, style and other finer details here.



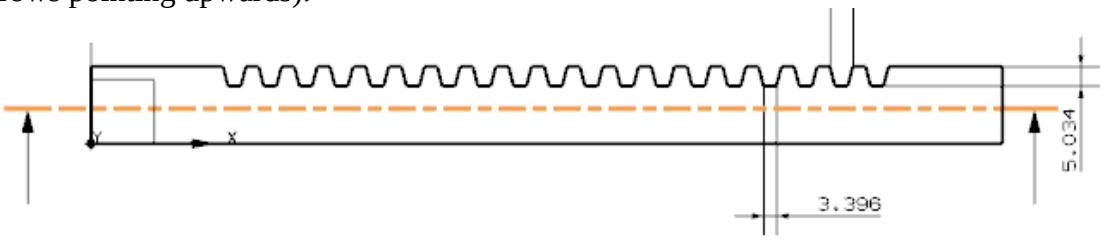
- Give dimensions to all other views as shown in the following figure



5.2.3 Sectional View

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

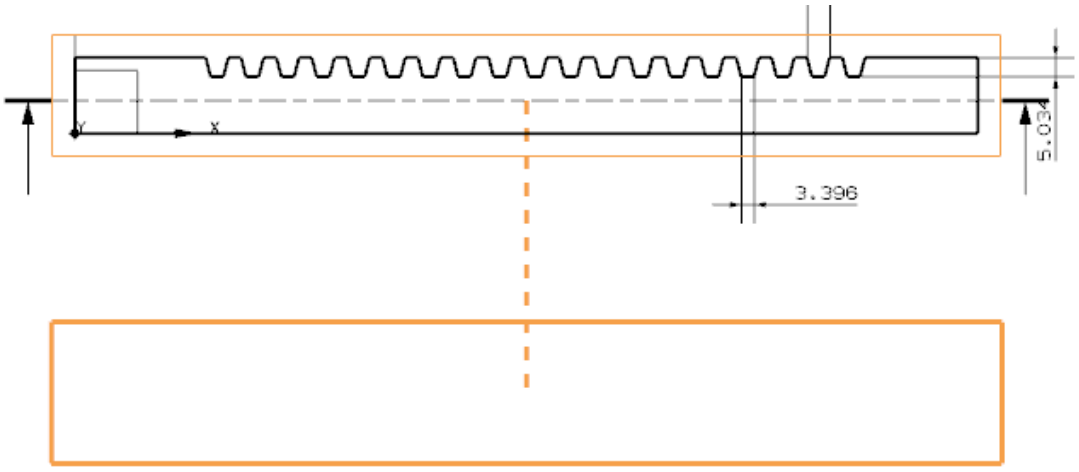
- Choose **INSERT → VIEW → SECTION→ SIMPLE/STEPPED**
- Click on the bottom of the **Base View** as Shown in the figure. This will show a **Phantom Line** with two **Arrow** marks for the direction of the Section plane (orange dashed line with arrows pointing upwards).



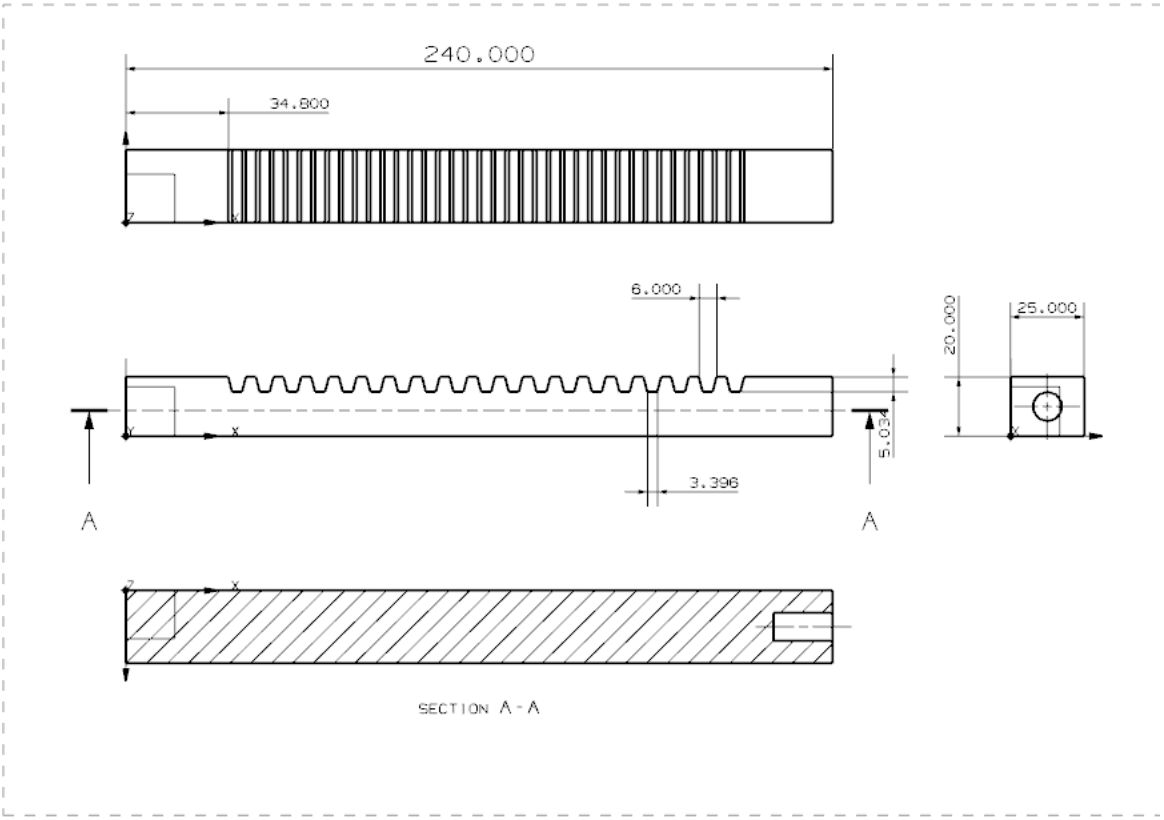
- Click on the middle of the View as shown.

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.

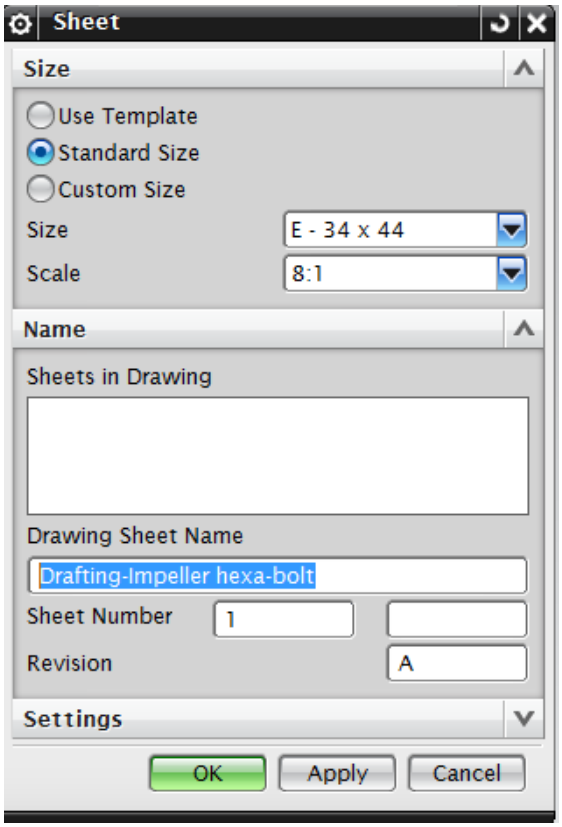


Save and close your model.

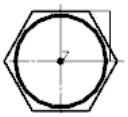
5.2.4 Drafting and Dimensioning of an Impeller hexagonal bolt

- Open the model **Impeller_hexa-bolt.prt**

- Choose **FILE → DRAFTING**
- On the New Drawing Sheet window, select sheet **E-34 X 44** and change the **Numerator Scale** value to **8.0 : 1.0**
- Choose **INSERT → VIEW → BASE VIEW**
- Add the **FRONT** view by repeating the same procedure as in the last example
- Add the **Orthographic Views**, including the right side view and top view
- Choose **PREFERENCES → DRAFTING**
- Uncheck the box next to ***Display Borders*** under **View** Tab



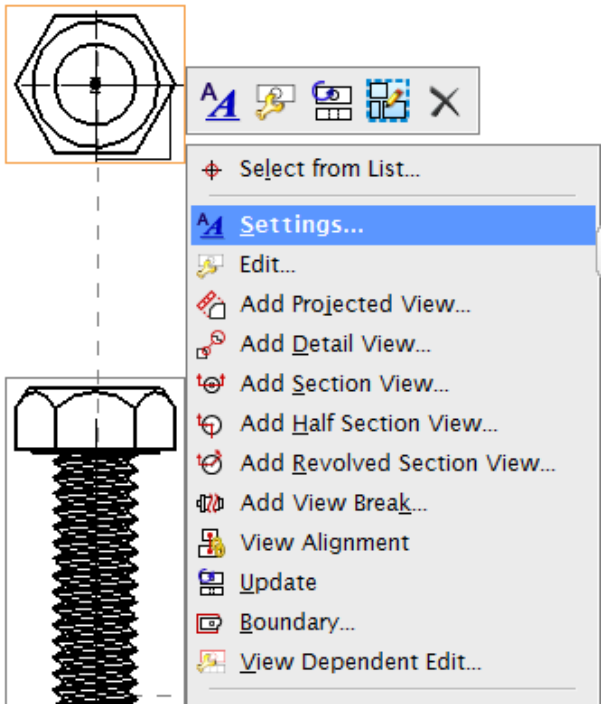
The screen will have the following three views:



There are always the hidden lines, which are not seen. To see the hidden lines

- Choose **PREFERENCES →DRAFTING→ VIEW** OR

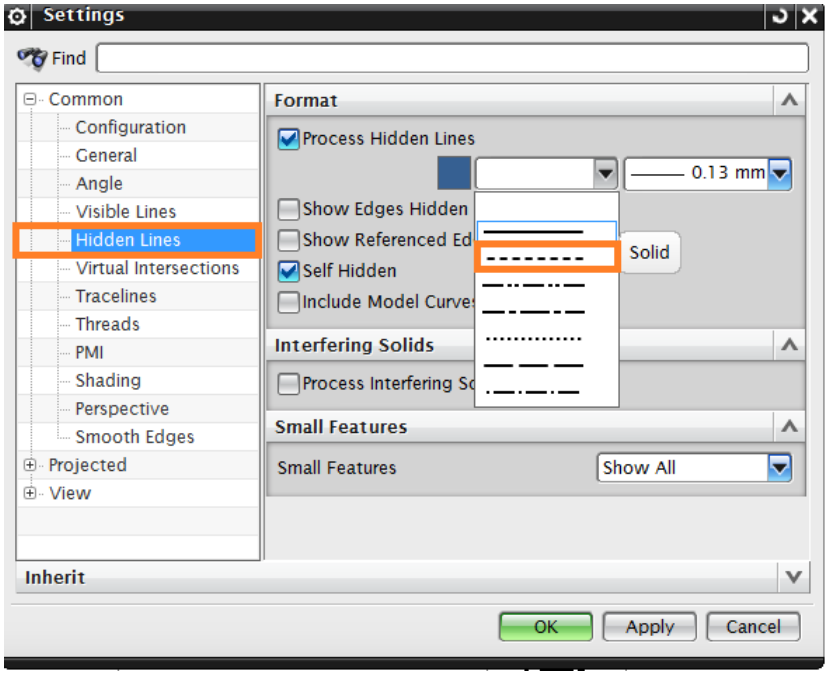
- Select the views, right-click and choose **SETTINGS** as shown below



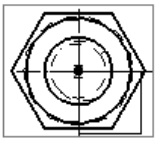
A window will pop up with various options pertaining to the views.

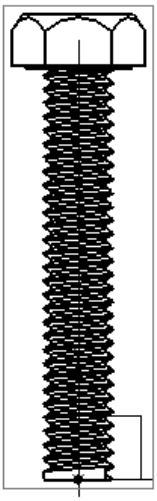
- Click on the **Hidden lines** tab

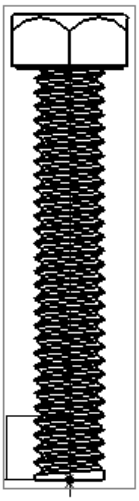
- Change **Process Hidden Lines to DASHED LINES** as shown below and click **OK**



You can see the hidden lines as shown below.

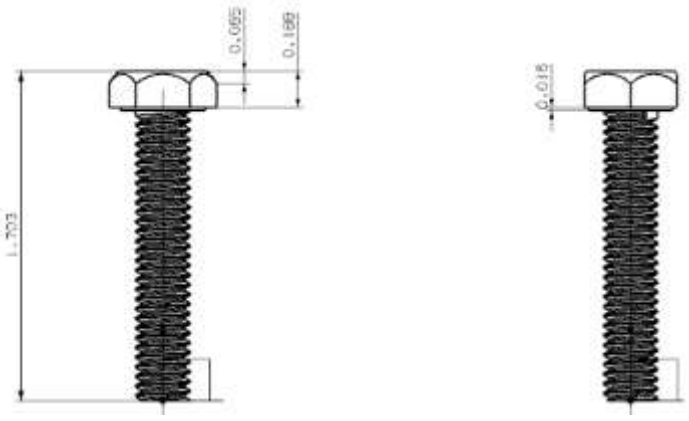




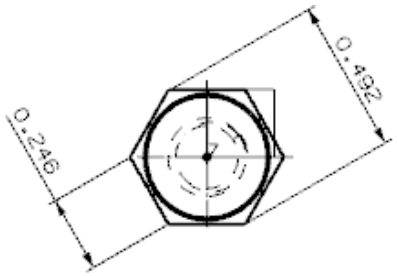


Now we will proceed to dimensioning.

- Choose **INSERT → DIMENSIONS → LINEAR**
- Give vertical dimensions to all the distances shown below.



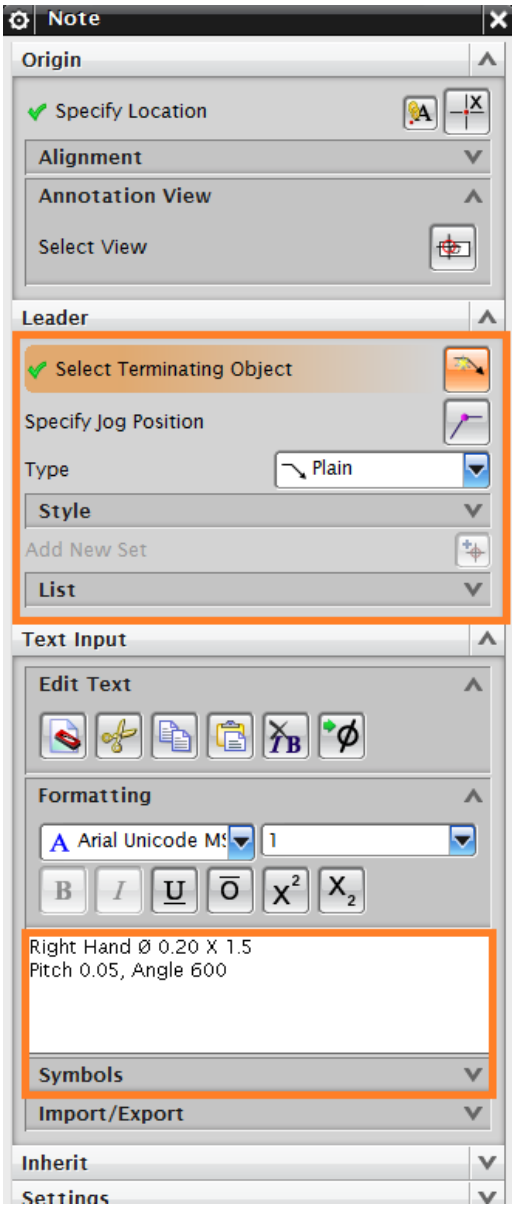
For the threading, we will use a leader line.



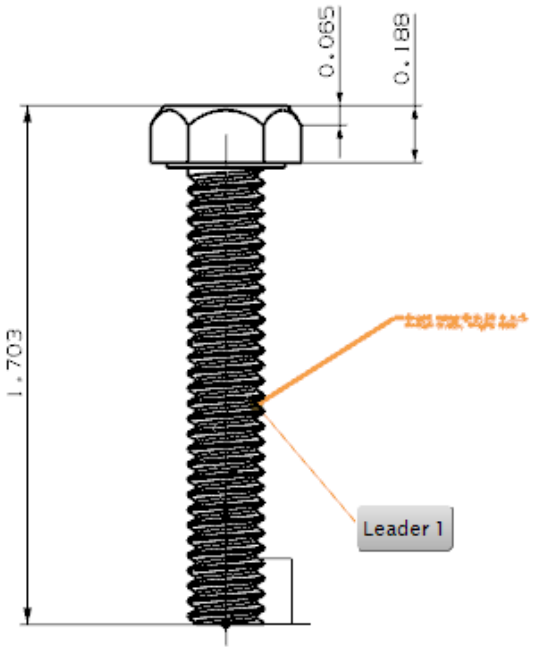
- Click on the **Note** icon shown in the Toolbar

- In the **Annotation Editor** window that opens, enter the following text exactly as shown. You can find Ø and the degree symbol on the **Symbols** tab

  Right Hand Ø 0.20 X 1.5
  Pitch 0.05, Angle 60^0



- 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**

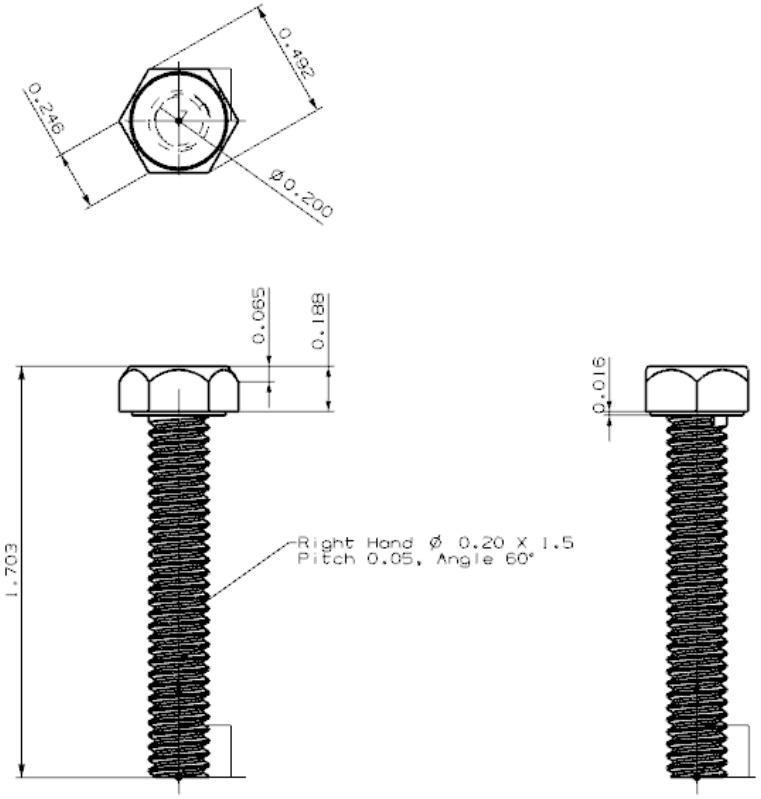
Since the height of the Lettering is small, we will enlarge the character size as well as the arrow size.

- 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.

Now we will add additional dimensions and views.

- Choose **INSERT → DIMENSIONS → RADIAL**
- Click the circle of the bolt in the top view to give the diameter dimension
- Click **INSERT → VIEW → BASE VIEW**
- Select the **ISOMETRIC** view and place the view somewhere on the screen

The final drawing is shown below. Remember to save.



5.3 EXERCISE - DRAFTING AND DIMENSIONING OF A CIRCULAR BASE

As an exercise, perform drafting and give dimensions to the circle base that you modeled in Exercise 4.4. The model of the part is displayed below.

CHAPTER 6 – SKETCHING

In this chapter, you will learn how to create and edit sketches in NX9.0. Sketching in NX9.0 version is much more user-friendly compared to its older versions. Similar to NX7, in NX9.0 you can directly create a sketch in **Modeling** application. Up to this point, the only way you have learned to create a new model is by creating and operating form features. In this second method of modeling, you will first create a sketch and then extrude, revolve or sweep the sketch to create solids. Many complex shapes that are otherwise very difficult to model using ***primitives*** or other ***form features*** 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.

6.1 OVERVIEW

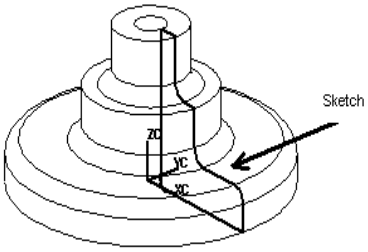
An NX9.0 sketch is a named set of curves joined in a string that when swept, form a solid. The sketch represents the outer boundary of that part. The curves are created on a plane in the sketcher. In the beginning, these curves are drawn without any exact dimensions. The solids created can be united into a single part using constraints. There are two kinds of constraints:

1) Geometric constraints
2) Dimensional constraints

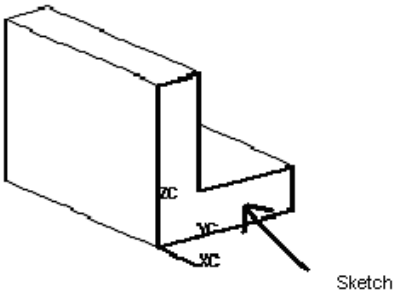
These will be discussed in detail later.

These are the different ways that you can use sketches:

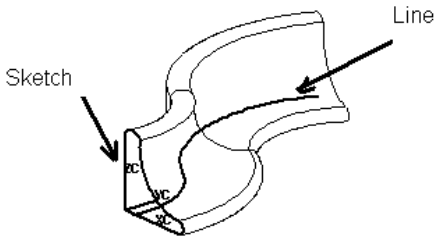
- A sketch can be revolved:



- A sketch can be extruded:



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



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

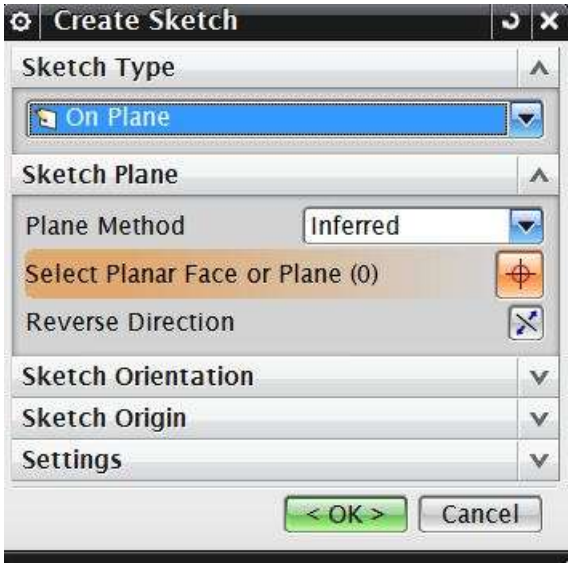
The advantages of sketching over using primitives are:

a) The curves used to create the profile outline are very flexible and can be used to model unusual shapes.
b) The curves are parametric, hence associative and they can easily be changed or removed.
c) If the plane in which the sketch is drawn is changed, the sketch will be changed accordingly.
d) 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.

6.2 SKETCHING FOR CREATING MODELS

In earlier chapters, we dealt with the ***Form Features*** and the ***Feature Operations***. In this chapter, we will model complex shapes by using sketching.

6.2.1 Model an Arbor Press Base



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

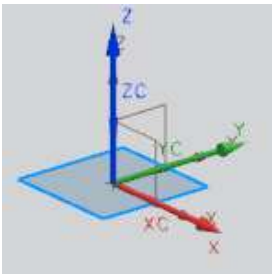
In NX9.0 you can create sketch using two ways. The first method creates the Sketch in the current environment and application. For this you will have to use

- Choose **MENU⟶ INSERT⟶ SKETCH**

In the other method you can create Sketch using

- Choose **SKETCH in HOME toolbar**

In either of the case, it pop-ups a dialog box asking you to define the ***Sketch Plane***. We will use second method of creating sketch. The screen will display the Sketch options. You can choose the sketch plane, direction of sketching and type of plane for sketching. The default sketch plane is the X-Y plane. 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.



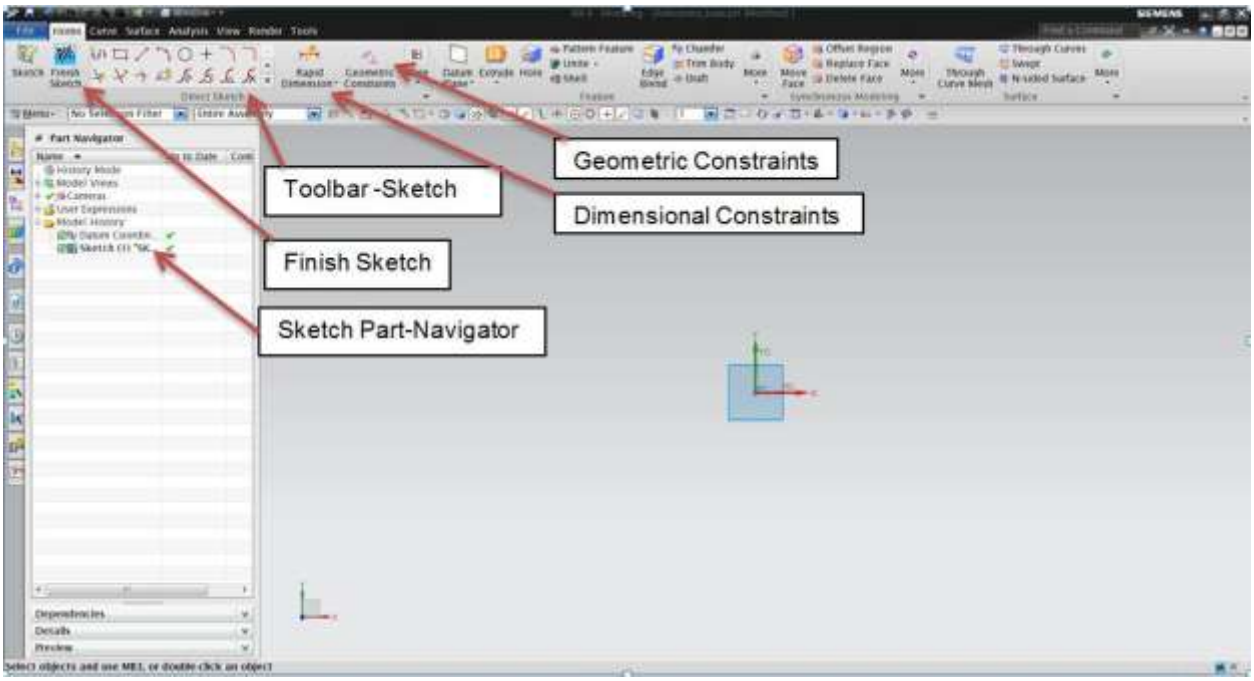
- Choose the **XC-YC** plane and click **OK**

The sketch plane will appear and the X-Y directions will be marked. This is 2D sketching.

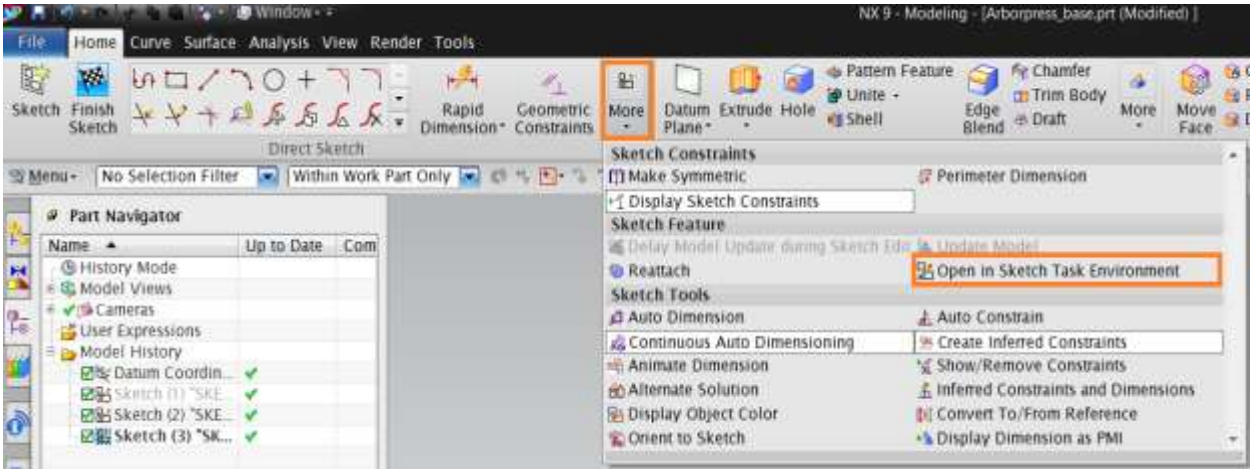
The main screen will change to the **Sketch Environment**. The XY plane is highlighted as the default plane for sketching. However, you can

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.

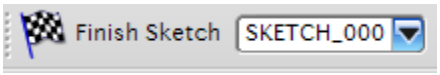
This is the basic sketch window. It can be divided into various parts, which have been labeled.



There is a special sketch task environment in NX 9.0, which displays all sketch tools in the main window.
For accessing the **Sketch Task Environment,** click the **More** option in the direct sketch tool bar area and then click on **Sketch Task Environment** as shown below.



You can change the name of the sketch in the box next to the Finish Flag.

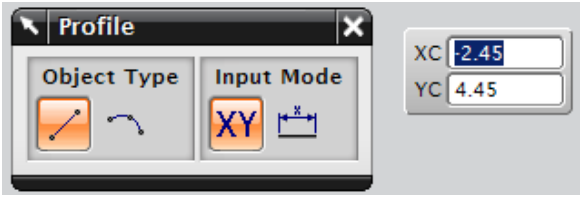


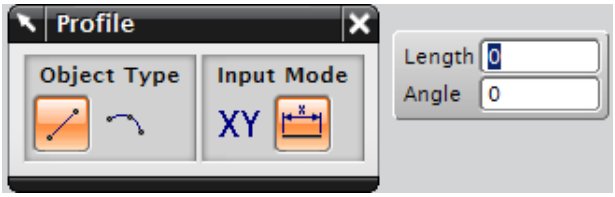
6.2.1.1 Sketch Curve Toolbar

This Toolbar contains icons for creating the common types of curves and Spline curves, editing, extending, trimming, filleting etc. Each type of curve will have different methods of selection 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.



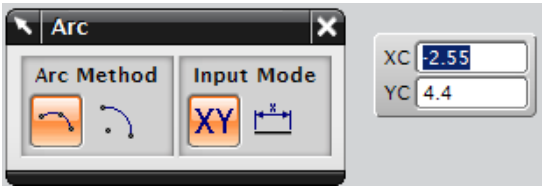


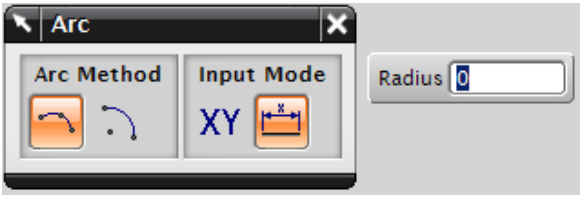
Line:

This option will selectively create only straight lines.

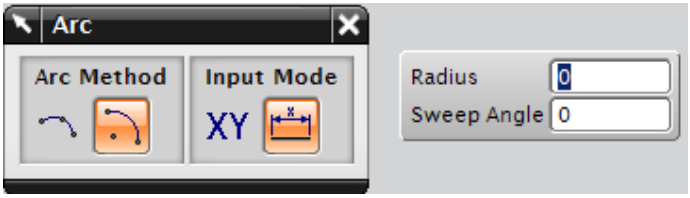
Arc:

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:

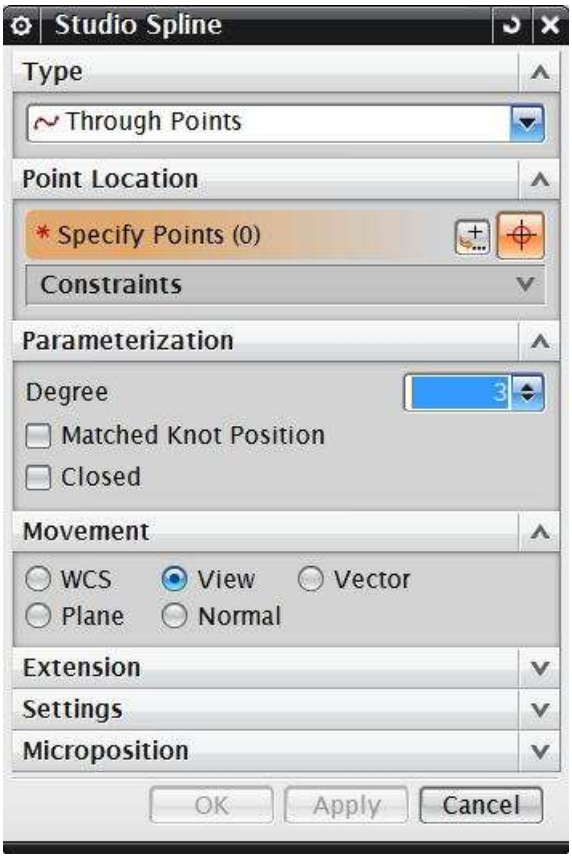
Creating a circle is similar to creating an arc, except that circle is closed unlike an arc.

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 next chapter (Freeform Features).



6.2.1.2 Constraints Toolbar

All the curves are created by picking points. For example, a straight line is created with two points. In a 2-D environment, any point will have 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 will have 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 by relating the entities directly or indirectly to the fixed entities. It can be done by giving dimensional or geometric properties like Parallelity, Perpendicularity, etc. In NX9.0 smart constraints are applied automatically, i.e. automatic dimensions or geometrical constraints are interpreted by NX9.0.

(Note: Any degrees of freedom that are not constrained are displayed in orange arrows . All these arrows should be removed by applying the constraints to follow a disciplined modeling.)

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 anytime 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 parallel, perpendicular, collinear, concentric, horizontal, vertical, equal length, etc. The software has the capability to find the set of possible constraints for the selected entities.

Show all Constraints:

Clicking this icon will show all the options pertaining to the entities in that particular sketch in white.

Show/Remove Constraints:

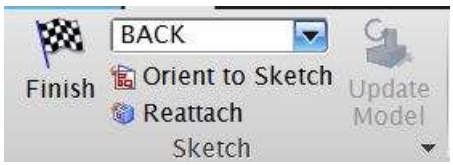
This window lists all the constraints and types of constraints pertaining to any entity selected. You can delete any of the listed constraints or change the sequence of the constraints.

6.2.1.3 Sketcher Toolbar

Besides being able to change the name of the Sketch, the sketcher toolbar also has some other highly useful features mentioned below.

Orient to 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 Sketch:

This function allows you to reattach the sketch to the desired plane without recreating all the curves, dimensions, and constraints.

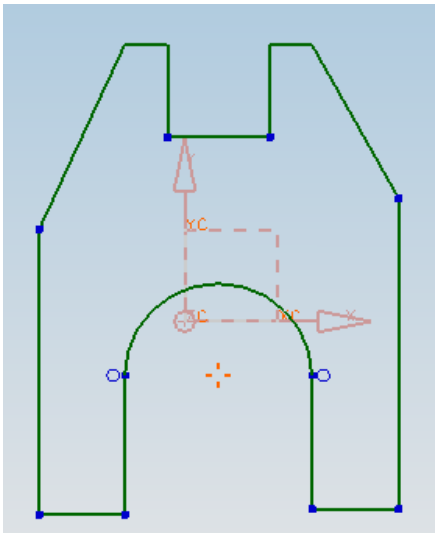
Update Model:

When you make changes in a sketch, click on this icon to see the effects of those changes without exiting the Sketch mode.

Now we will draw curves using the options discussed above.

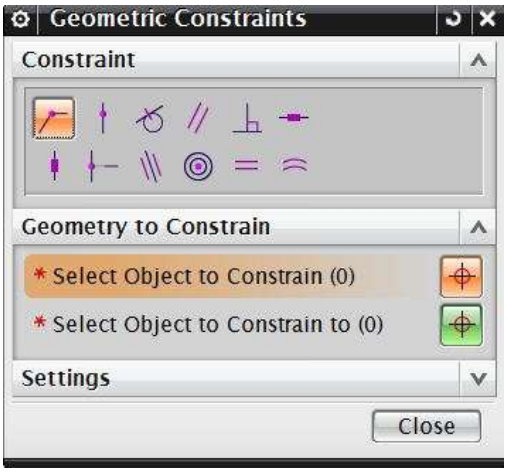
- Choose **MENU⟶ INSERT ⟶ CURVE ⟶PROFILE** if the Profile window is not already showing.

- Draw a figure similar to the one shown below. 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.)

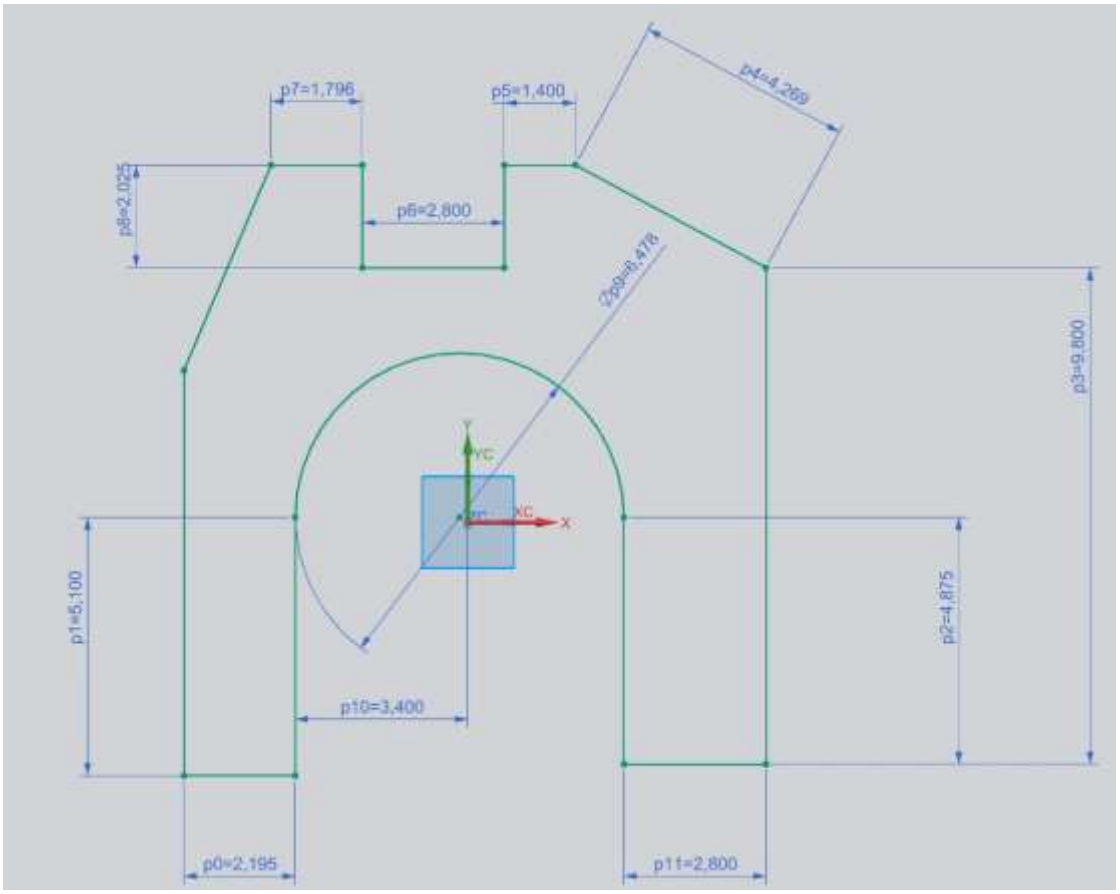


Once the sketch is complete, we will constrain the sketch. It is better to apply the geometric constraints before giving the dimensional constraints.

- Choose **INSERT → GEOMETRIC CONSTRAINTS** or click on the Constraints icon in the side toolbar

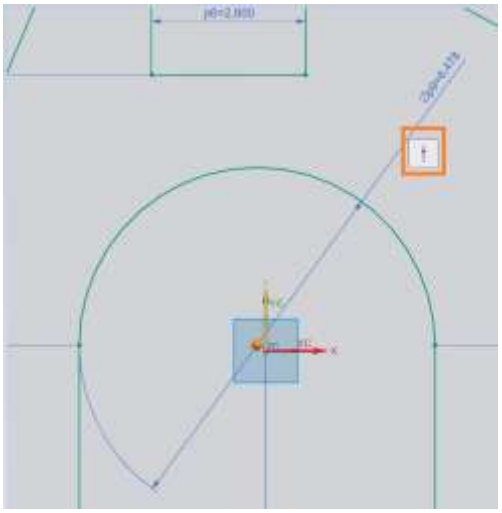


You will be able to see all the degrees of freedom on the screen represented by red arrows.



Note the automatic dimension constraints being applied to each entity. Now we will start by constraining between an entity in the sketch and the datum or fixed reference. Note that when the figure is not completely constrained it will appear light green.

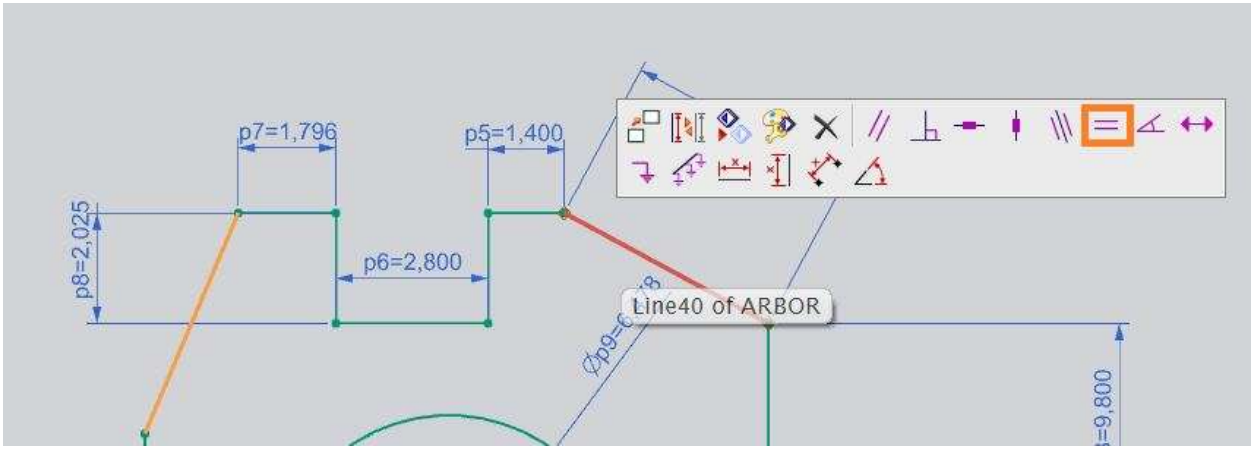
We will first place the center of the arc at the origin. This creates a reference for the entire figure. We can use the two default X and Y axes as a datum reference.



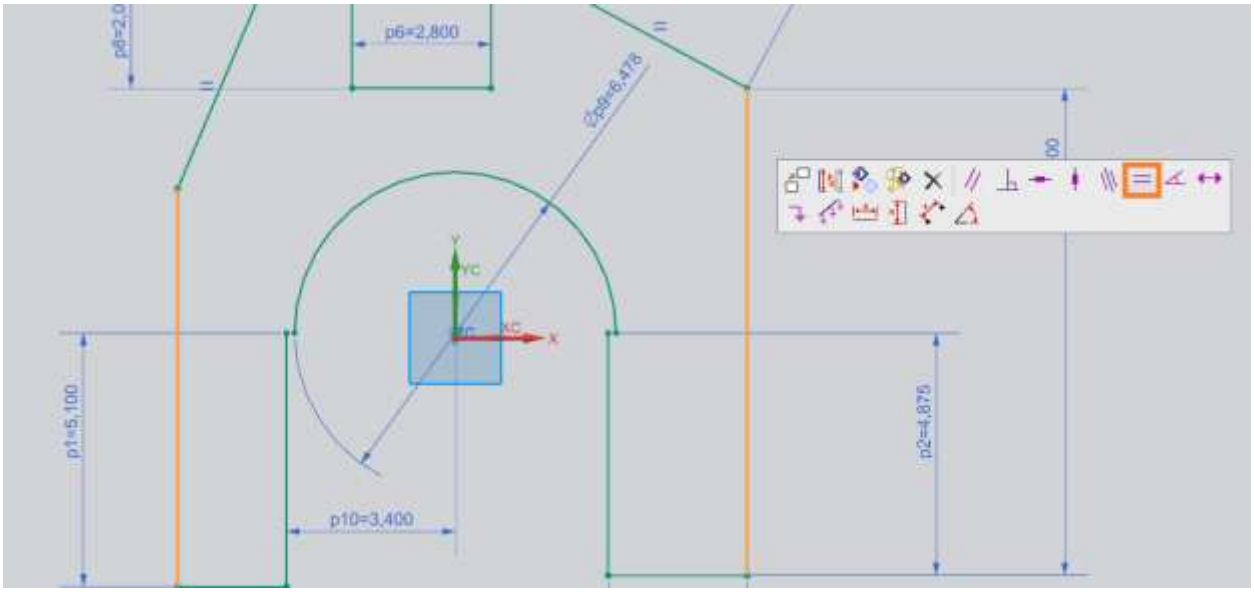
- Select the Y-axis and then the center of the arc, which is marked by the '**+**' sign. The center of the arc will be marked by a red asterisk once it has been selected.
- Click the **Point on Curve** icon
- Repeat the same procedure to place the center of the arc on the X-axis

Do not worry in case the figure gets crooked. The figure will come back to proper shape once all the constraints are applied. Note that when you initially draw the unconstrained figure, take into consideration the final shape of the object.

- Select the two slanted lines and make them equal in length

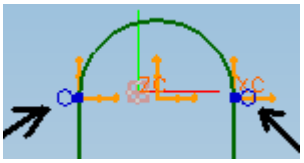


- Similarly select the two long vertical lines and make them equal in length



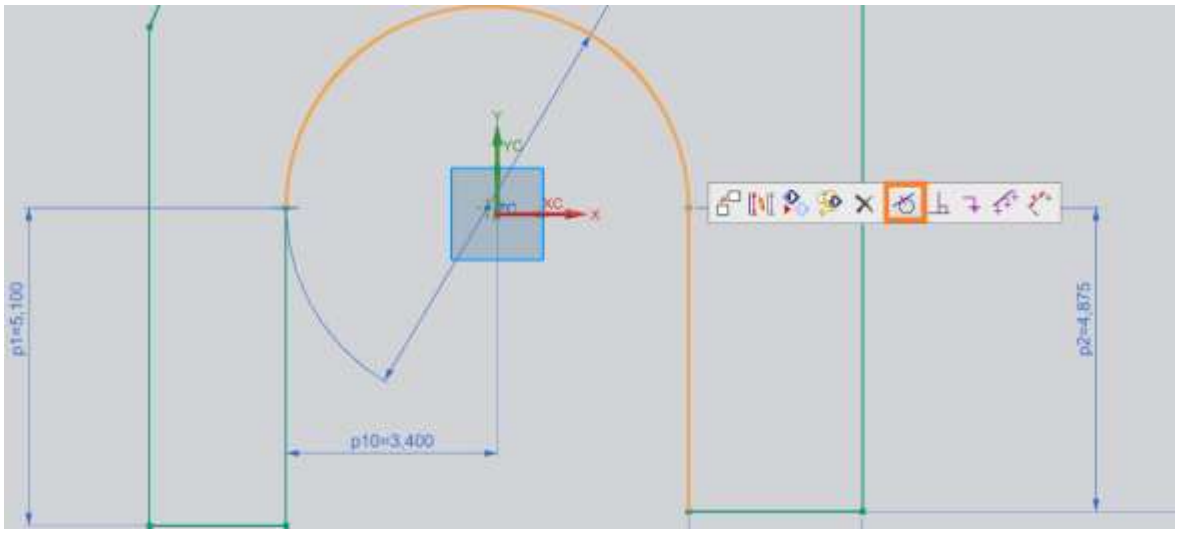
- Select the bottom two horizontal lines and make them collinear and then click on the same lines and make them equal in length

If you **DO NOT** find the two Blue circles (Tangent Constraints) near the semicircle as shown in the figure, follow the below steps. Otherwise, you can ignore this and skip down to the dimensional constraints.



- Select the circular arc and one of the two vertical lines connected to its endpoints

- Select the Tangent icon



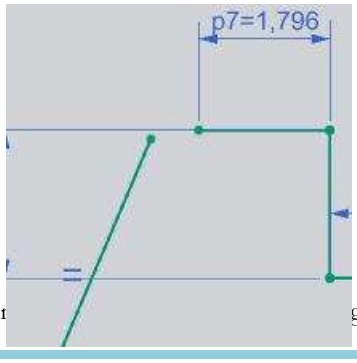
If the arc and line is already tangent to each other, the icon will be grayed out. If that is the case click on **EDIT → SELECTION →, DESELECT ALL**. Repeat the same procedure for the arc and the other vertical line.

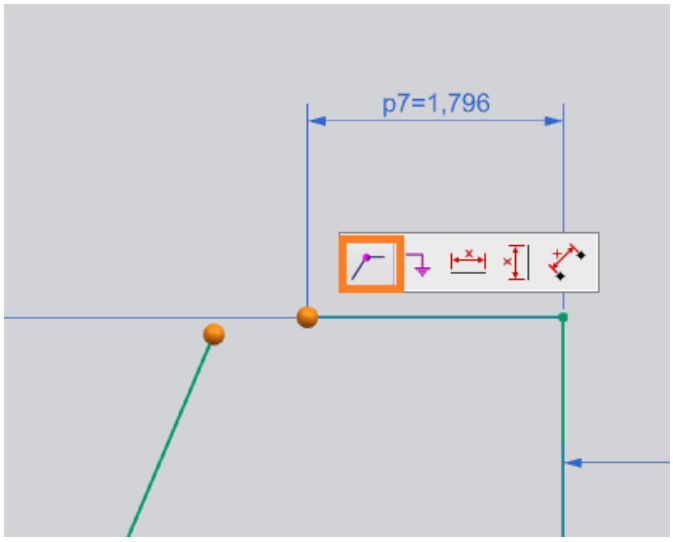
- Select the two vertical lines and make them equal

- Similarly select the two small horizontal lines and make them collinear and equal

- Similarly select the two vertical lines and make them equal

Note: At times the after applying a constraint, the geometric continuity of the sketch may be lost like shown. In such conditions, click the exact end points of the two line and click the coincident constraint as shown below



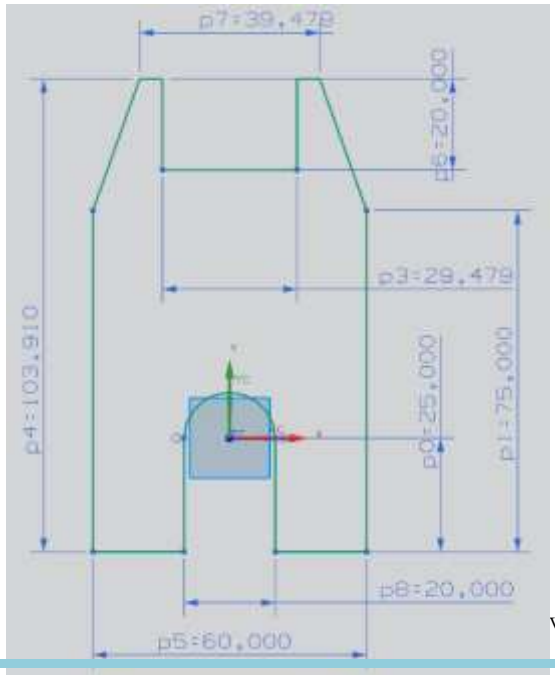


So far, we have created all the ***Geometric*** constraints. Now we have to create the ***Dimensional*** constraints. You will find that as we add on dimensions, the degrees of freedom represented by the yellow arrows will disappear. NX9.0 will not allow duplication of dimensions. This is why it is better to apply the geometric constraints first. If there is any conflict between the dimensional and geometric constraints, those entities will be highlighted in yellow.

- Choose the Inferred Dimensions icon in the Constraints toolbar

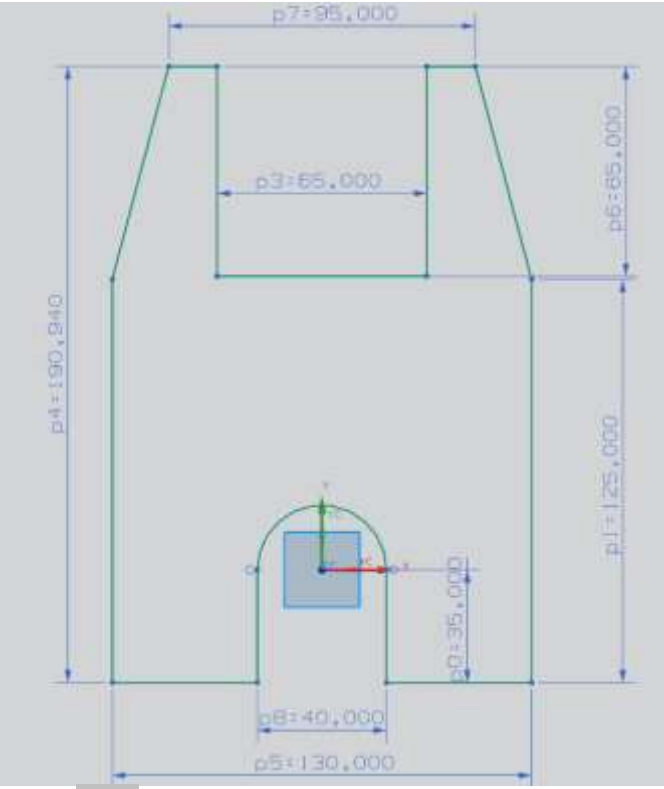
- Add on all the dimensions as shown in the following figure

For example, to create a dimension for the top two corners, you may have to click on the arrow next to the **Inferred Dimensions** icon and click on the **Horizontal** icon. Then click somewhere near the top of the two diagonal lines to select them. While dimensioning, if you find the dimensions illegible, but do not worry about editing the dimensions now. Make sure the small arrows are disappearing as constraints are placed.

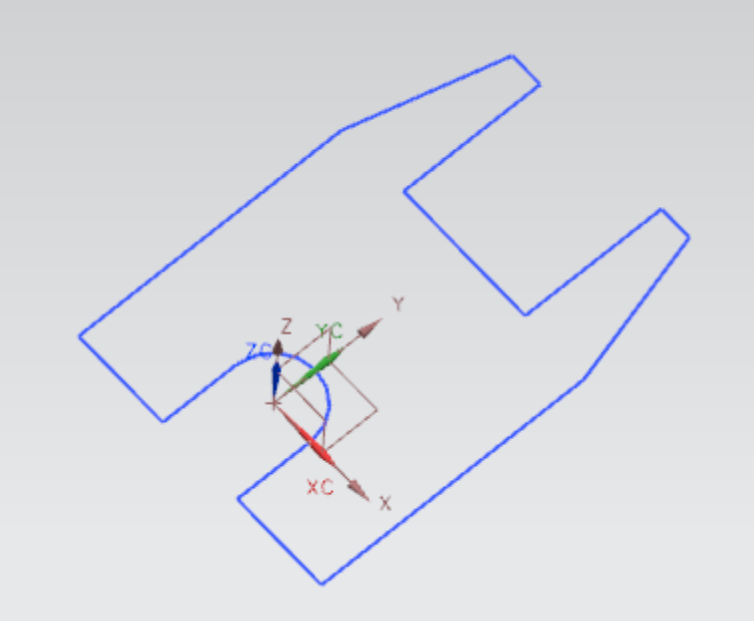


Now we will edit all the dimension values one by one. It is highly recommended to start editing from the biggest dimension first and move to the smaller dimensions.

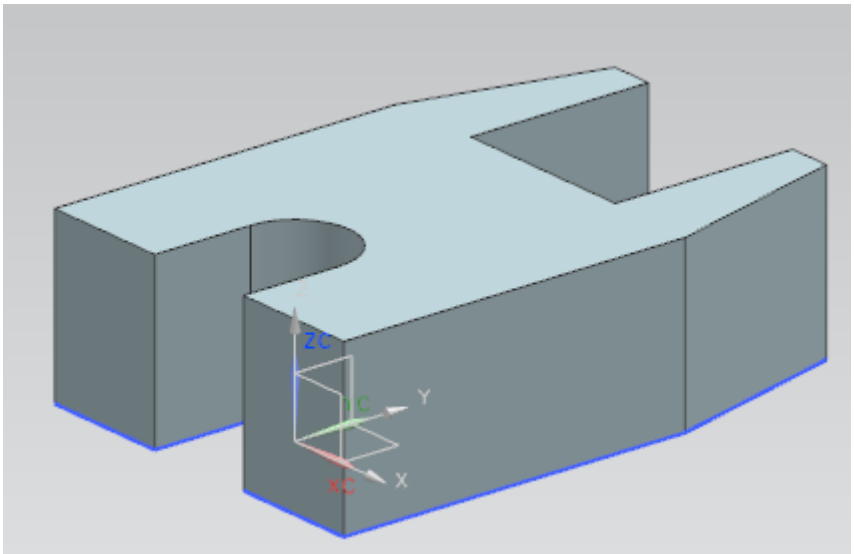
- Edit the values as shown in the figure below. Double click on each dimension to change the values to the values as shown in figure below:



- Click on the **Finish flag** on the top left corner of the screen when you are finished
- Click on the sketch and right-click
- Click **INSERT → DESIGN FEATURE → EXTRUDE**

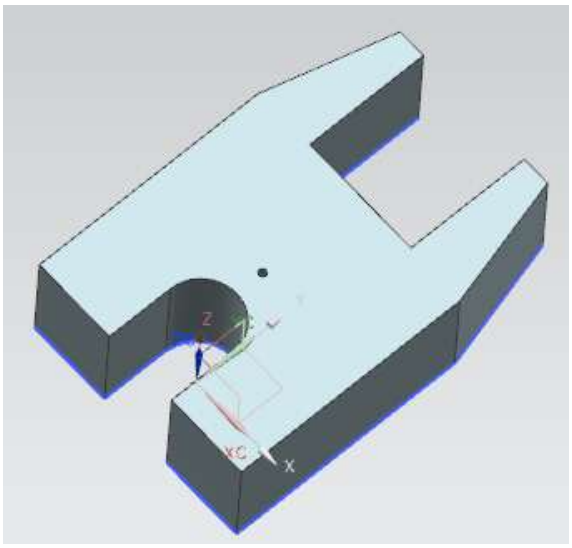


- Extrude this sketch in the Z-direction by **60** inches



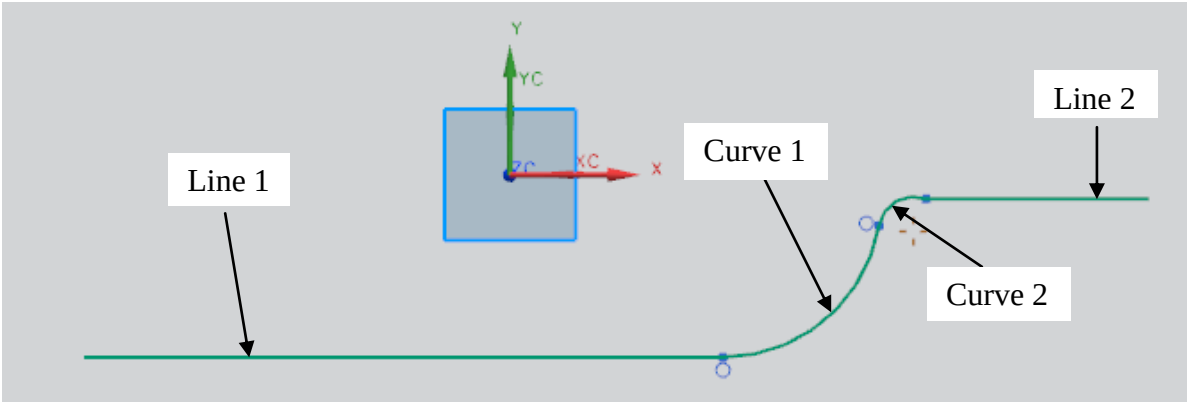
- Create a hole with a diameter of **4** inches and a height of **30** inches at the point **(0, 35, 0)** from the WCS

The final figure is shown below. Save and close the file.



6.2.2 Model an Impeller Lower Casing

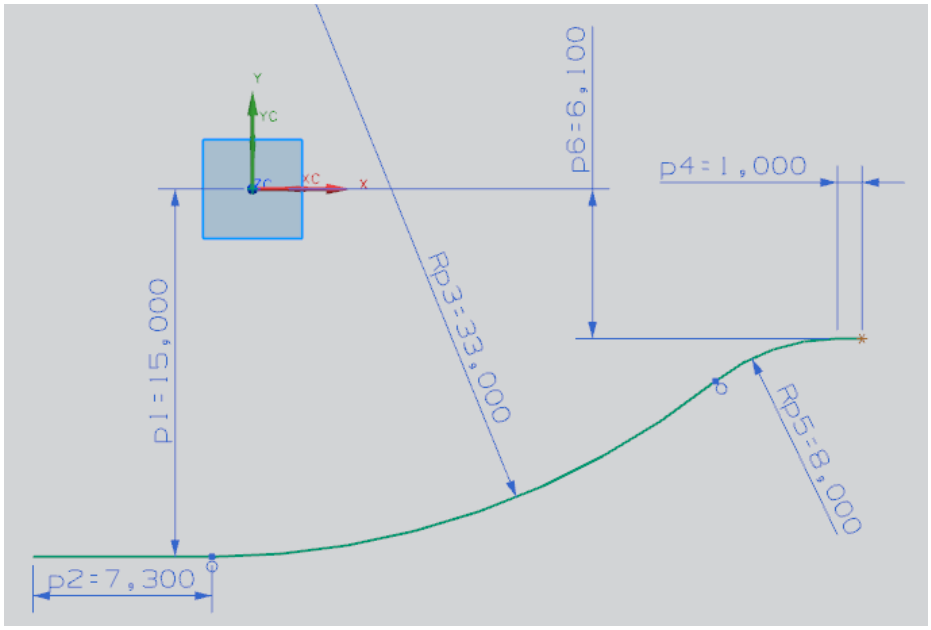
- Create a new file and save it as **Impeller_lower_casing.prt**
- Click on **INSERT → SKETCH in TASK ENVIRONMENT**
- Set the sketching plane as the **XC-YC** plane
- Make sure the **Profile** window is showing and draw the following curve



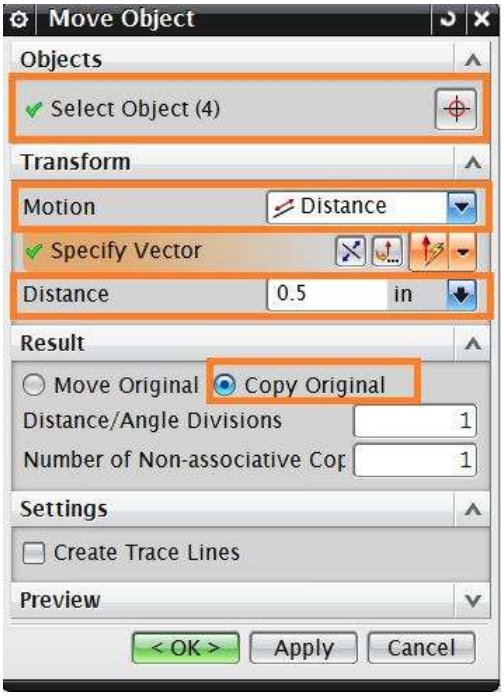
- Click **INSERT → DATUM/POINT**
- Create a point at the origin (0, 0, 0) and click **OK**
- Click **Close** to exit the **POINT CONSTRUCTOR** window

Next, we will constrain the curve.

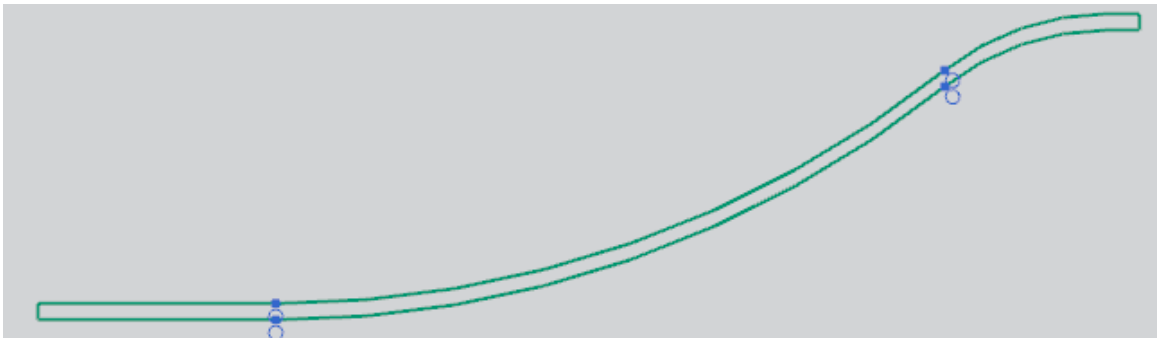
- Click on the **Geometric Constraints** icon
- Select the point at the origin and click on the **Fixed** constraint icon
- Make all of the curve-lines and curve-curve joints tangent
- Then apply the dimensional constraints as shown in the figure below:



- Select all the dimensions. Right click and **Hide** the dimensions
- Choose **EDIT → MOVE OBJECT**
- Select all the curves. You should see '4' objects being selected in ***Select Object***
- Specify the ***Motion*** to be *Distance*
- Choose **–YC-Direction** in the ***Specify Vector***
- Enter the ***Distance*** to be **0.5** inch
- In the ***Result*** dialog box make sure you the click on the *Copy Original* radio button
- Click **OK**



- Then join the end-points at the two ends using the basic curves to complete the sketch

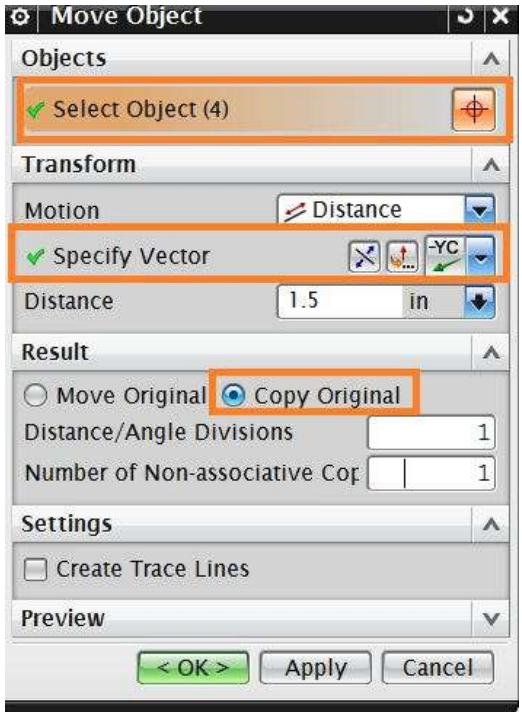


The sketch is ready.

- Choose **EDIT → MOVE OBJECT**

- Select the outer curve as shown in the figure below. Be sure to select all the four parts of the curve.

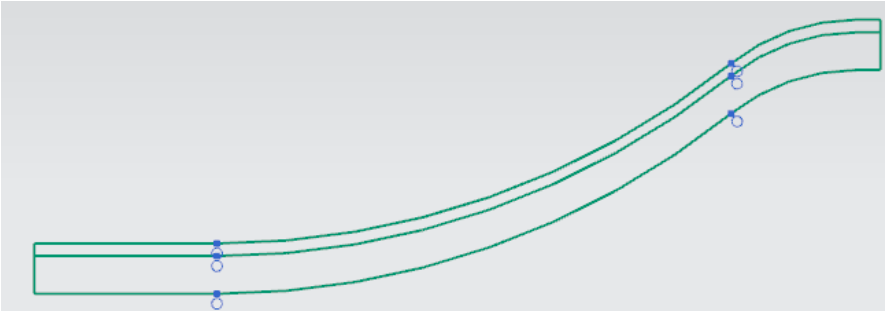
- **Move** the lower curve in the Y-direction by **-1.5** inches. This is the same as translating it in the ***negative YC-direction*** by **1.5** inches



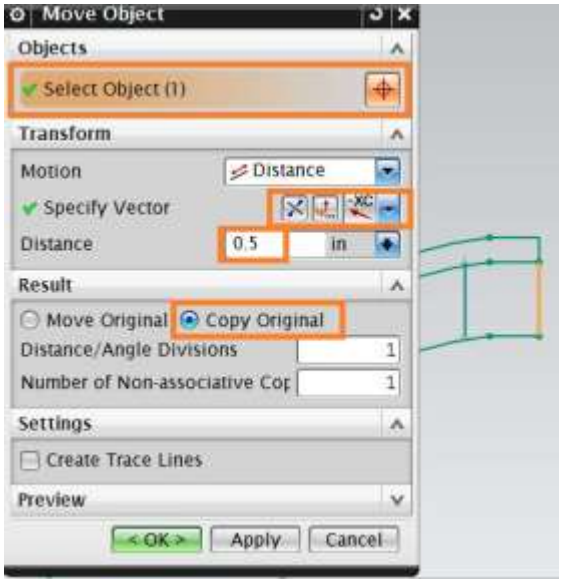
This will form a curve outside the casing.

- 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.



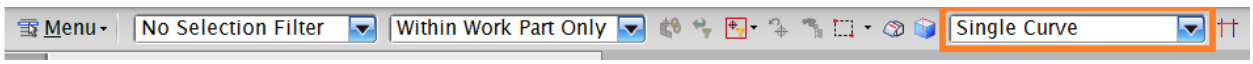
- Choose **EDIT → MOVE OBJECT**

Select the outer curve as shown in the figure on the right.

- **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 as shown in the figure on right side
- Click on the Finish Flag

- Click on **INSERT → DESIGN FEATURE → REVOLVE**

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



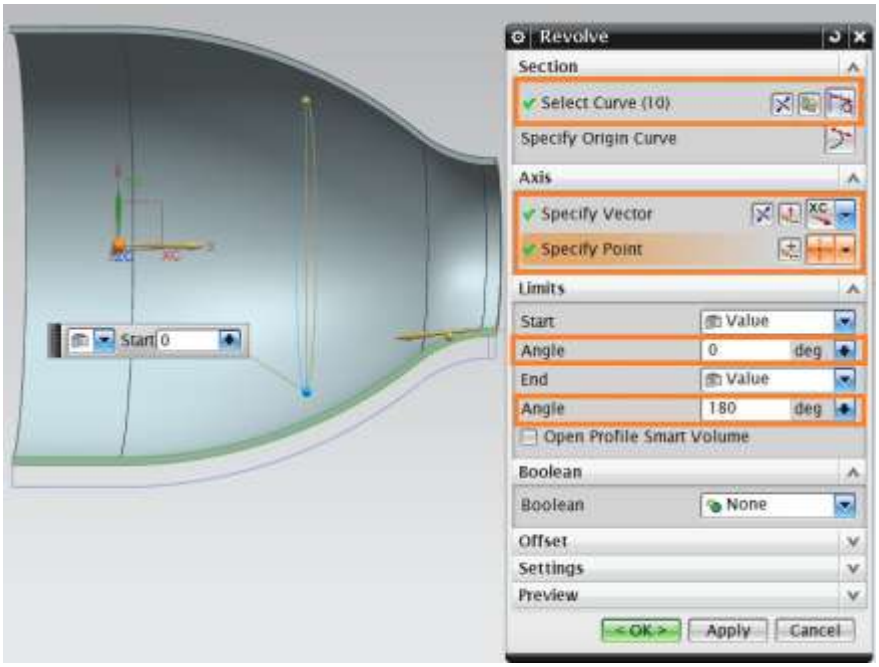
- Click on each of the 10 curves as shown

- In the **Axis** dialog box , in the ***Specify Vector*** option choose the *positive XC-direction*

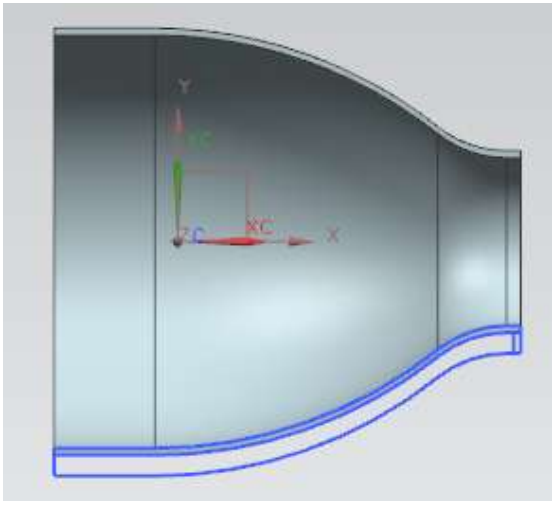
- In the ***Specify Point*** option, enter the coordinates (0, 0, 0) so the curve revolves around XC-axis with respect to the origin

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

- Click **OK**



The solid is seen as below.



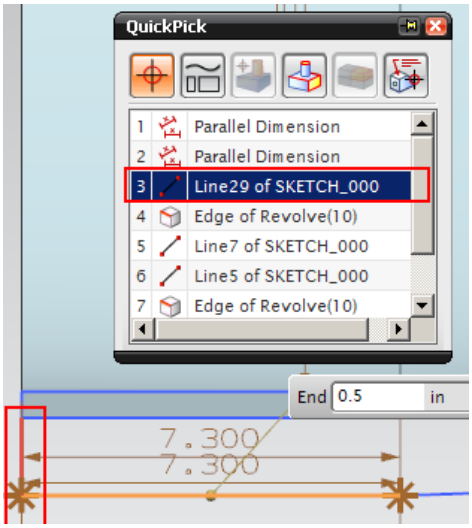
Now, we will create edges.

- Click on **INSERT → DESIGN FEATURE → EXTRUDE**

- Select the outer curve of the casing as shown in the figure below.

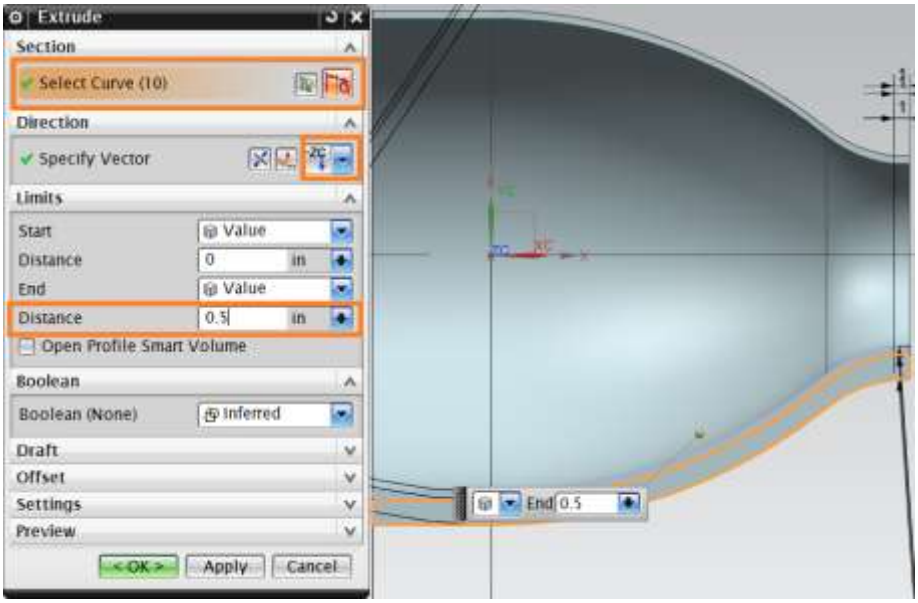
Again make sure that the ***Selection Filter*** is set to *Single Curve.*

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

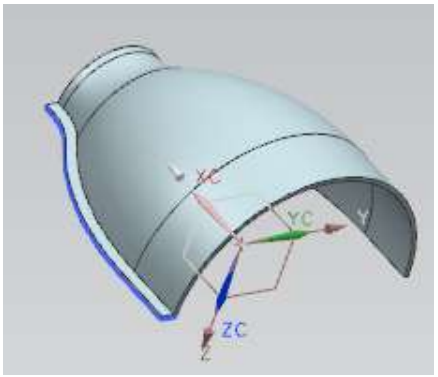


- Select the curve you just created in the second Sketch

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



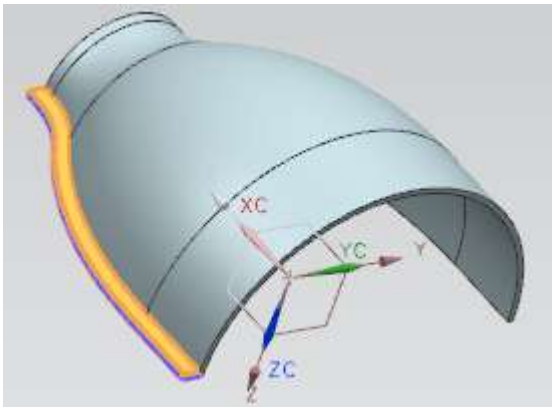
The final solid will be seen as follows.



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

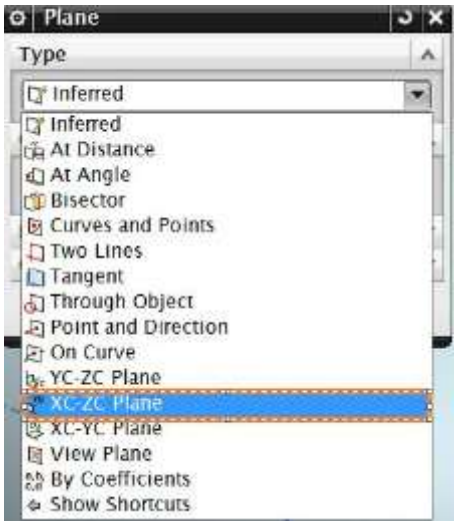
- Choose **EDIT → TRANSFORM**

- 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**

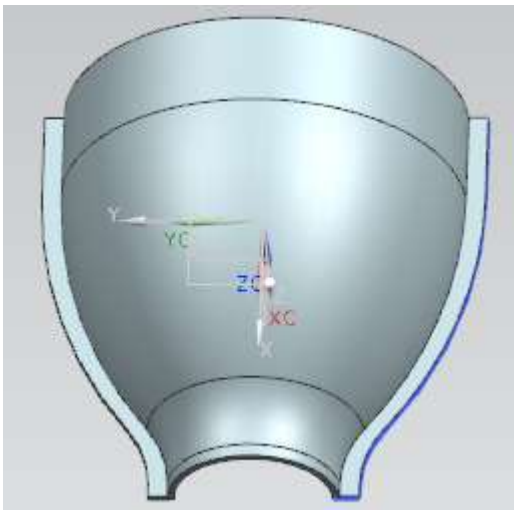
- Under Principal Planes, click on the ***XC-ZC*** icon as shown and click **OK**



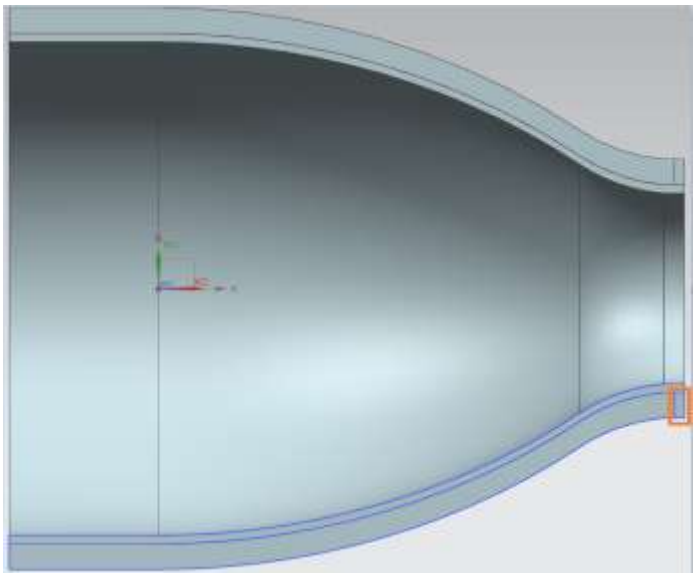
- Select **COPY**

- Click **Cancel**

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



We will create an edge at the smaller opening of the casing as shown.

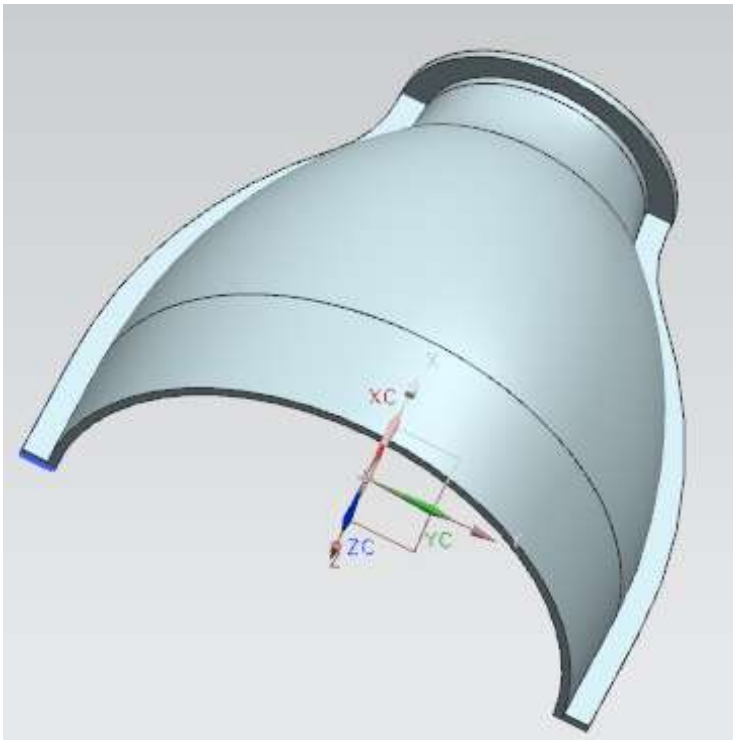


- Click on **INSERT → DESIGN FEATURE → REVOLVE**

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.

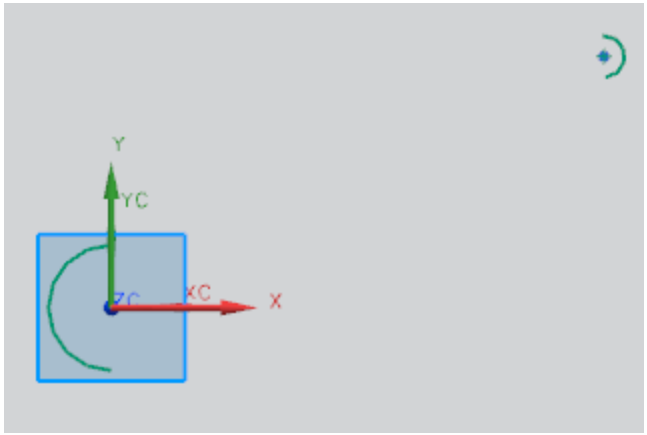


The lower casing is complete. Save the model.

6.2.3 Model an Impeller

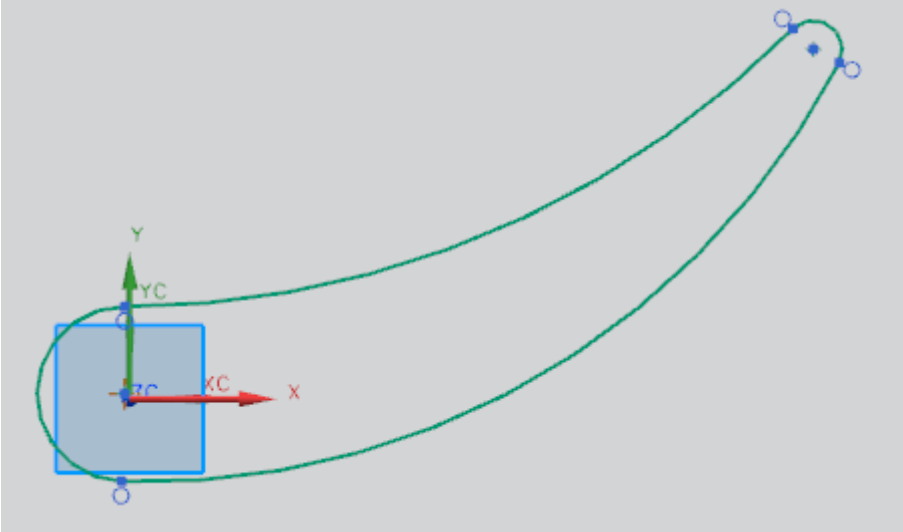
Create a new file and save it as **Impeller_impeller.prt**

- Click on **INSERT → SKETCH**
- Set the sketching plane as the **XC-YC** plane
- Click on **INSERT⟶ DATUM/POINT⟶ POINT**
- 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** 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**



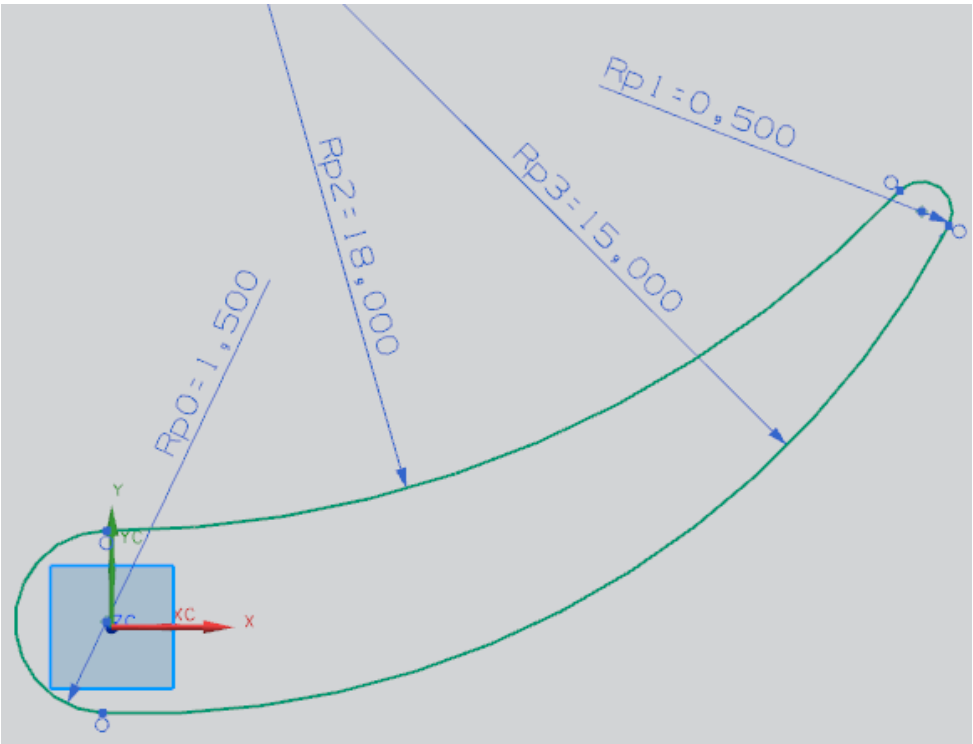
- Click on the Arc by 3 Points icon in the pop-up toolbar
- Select the top endpoints of the two arcs you just created and click somewhere in between to create another arc that connects them. Do the same for the bottom endpoints
- Click on the **Constraints** icon in the side toolbar and make sure that all the arcs are tangent to one another at their endpoints
- Click on the point at the origin and click on the **Fixed** icon

The sketch should look like the following.



- Then click on the **Inferred Dimensions** 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:

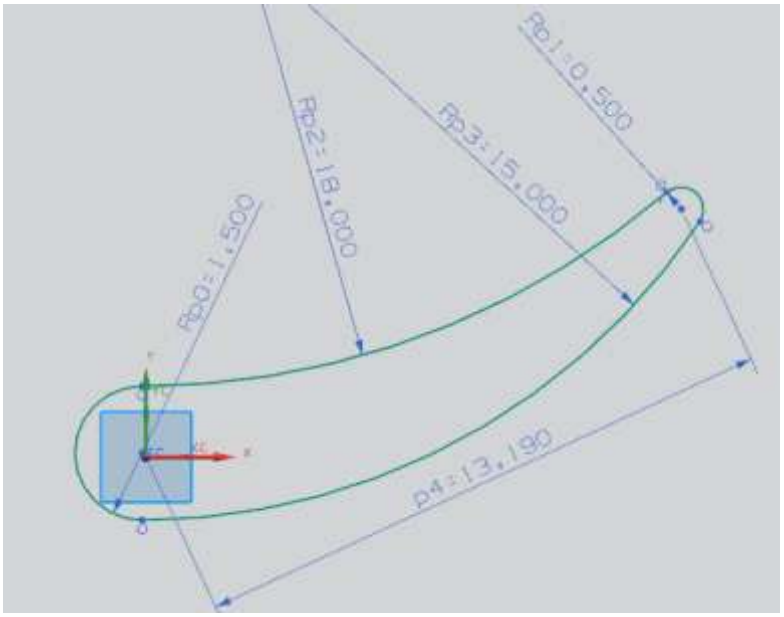


- Select the **Parallel** dimensioning option from the **Dimensions** drop-down menu

- Create a dimension giving the distance between the origin point and the other point and edit the distance to be **13.19** inches

- Click on the **Finish Flag**

Now the sketch is ready as shown below.



Now let us model a cone.

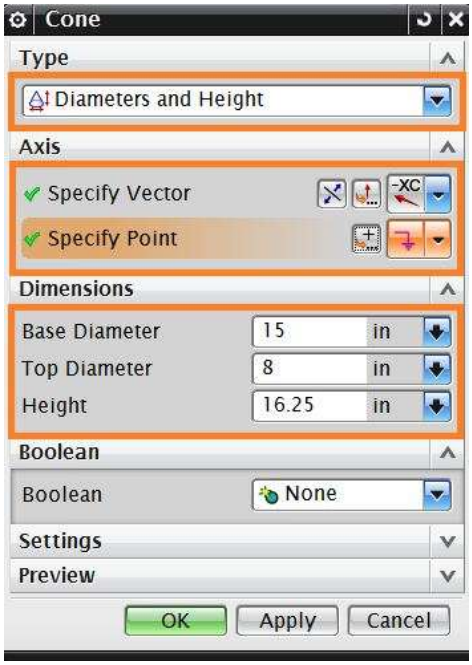
- Choose **INSERT → DESIGN FEATURE → CONE**

- Select **DIAMETERS, HEIGHT**

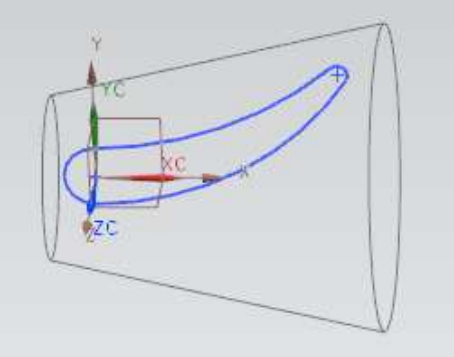
- Select the **–XC-Direction** in the ***Specify Vector*** dialog box

- In the **Point Constructor**, enter the coordinates **(14, 0, 0)** in the ***Specify Point*** dialog box

- Enter the following dimensions:
  Base Diameter = **15** inches
  Top Diameter = **8** inches
  Height = **16.25** inches

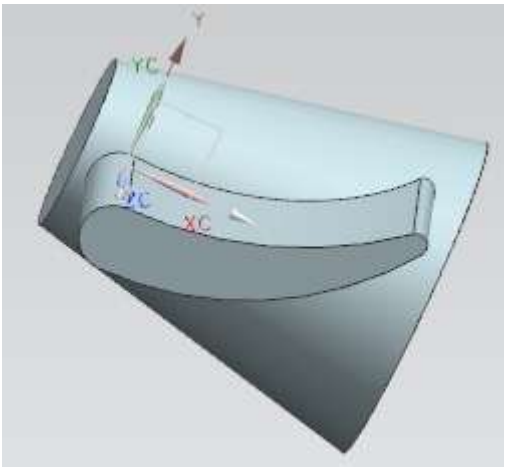


The cone will be seen as shown below.

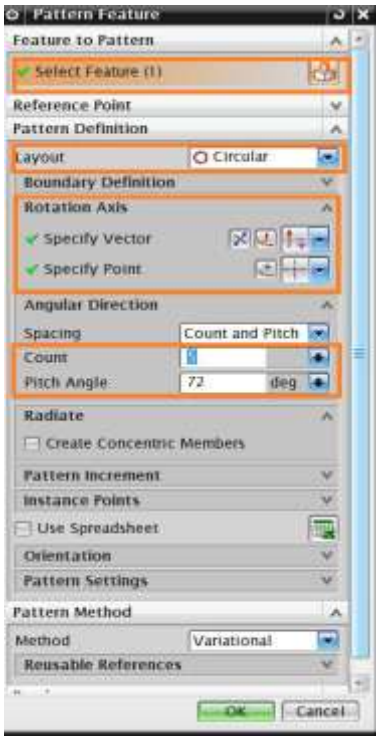


- **Extrude** the aerofoil curve in the **Z**-direction by **12** inches. **Unite** the two solids in the ***Boolean*** operation dialog box

The model will be as follows.

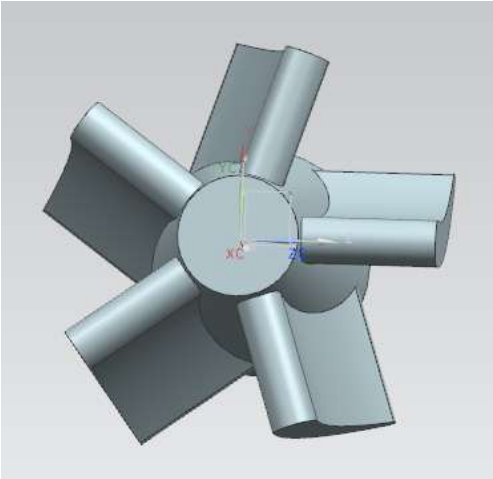


Now let us create five instances of this blade to make the impeller blades.



- Click on **INSERT → ASSOCIATIVE COPY → PATTERN FEATURE**
- Select **CIRCULAR** layout
- Select **EXTRUDE**
- For ***Count***, type in **5** and for ***Pitch Angle***, enter **72**.
- Select the **XC**-Direction for the ***Specify Vector*** and the **Origin** for the ***Specify Point***
- Click **OK**

The model will be seen as follows.

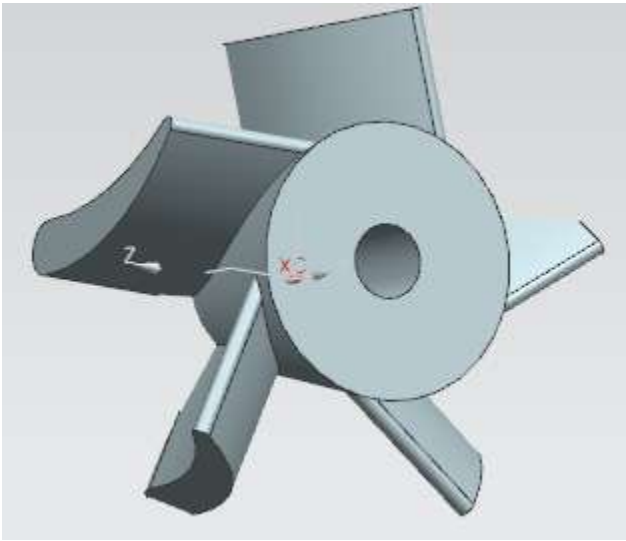


Now let us create two holes in the cone for the shaft and the locking pin. Note that these holes can also be created by **HOLE** menu option.

- **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 as shown

- **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

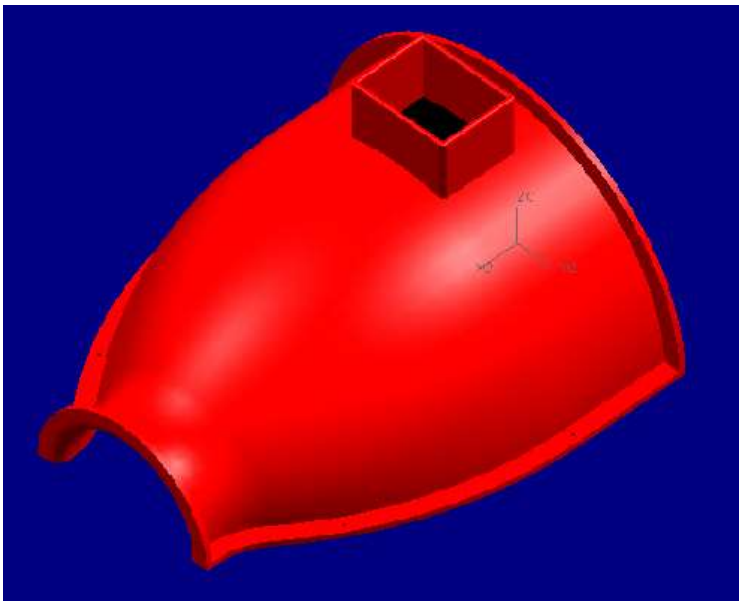
The final model will look like the following. Save your work.



6.3 EXERCISES

Exercise 1 - Model an Impeller Upper Casing:

As an exercise, model the upper casing of the Impeller as shown below.



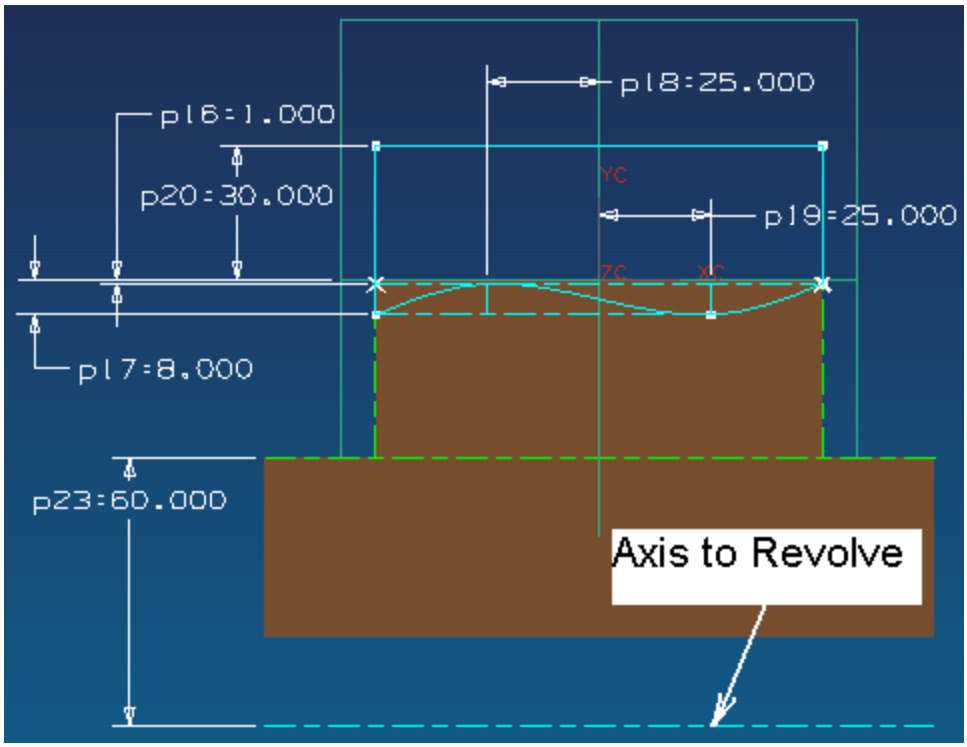
The dimensions of the upper casing are the same as for the lower casing, which is described in the previous exercise in detail. The dimensions for the manhole should be such that impeller blades can be seen and a hand can fit inside to clean the impeller.

Exercise 2 - Modeling a Die-Cavity:

Model the following part to be used for the Chapter 9 Manufacturing Module. Create a new file '**Die_cavity.prt'** with **units in *mm*** not in inches. Create a rectangular Block of 150, 100, 40 along X, Y and Z respectively with the point construction value of (-75,-50,-80) about XC, YC and ZC.

Create and Unite another block over the first one with 100, 80 and 40 along X, Y and Z. and centrally located to the previous block.

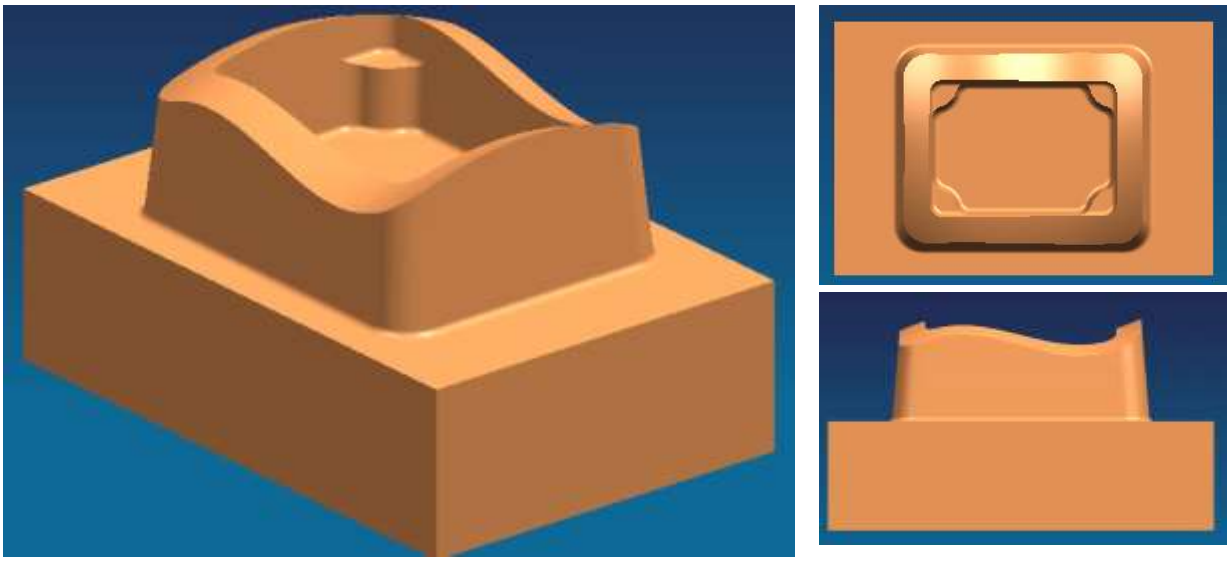
Create a sketch as shown below including the spline curve and add an Axis line. Dotted lines are reference lines. While sketching, create them as normal curves. Then right click on the curves and click convert to reference. Give all the constraints and dimensions as shown in the figure below.



Revolve the curves about the dashed axis as shown above, and subtract the cut with start angle and end angle as -45 and 45.

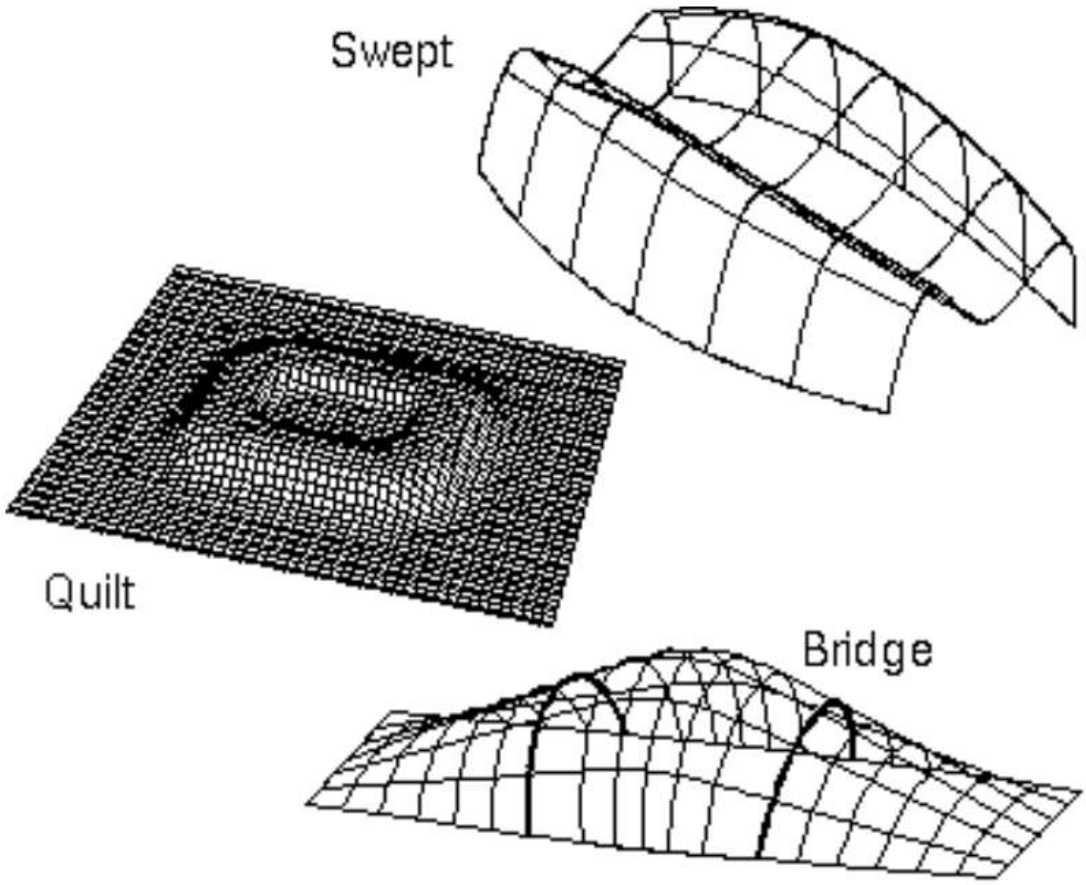
Subtract a block of 70, 50, and 30 to create a huge cavity at the centre. Create and Unite 4 cylinders at the inner corners of the cavity with 20 inches diameter and 15 inches height.

Add edge blends at the corners as shown in the final Model below. Keep the value of blend as 10 radii for outer edges and 5mm radii for the inner edges.



CHAPTER 7 – FREEFORM FEATURE

In this chapter, you will learn how to create freeform models in NX9.0. Up to this point, you have learned different ways to create models by using ***Form Features*** or by ***Sketch***. Freeform modeling involves creating solids in the form of surfaces particularly the B-surface. Because of their construction techniques and design applications, these surfaces are usually stylistic. A few freeform features are shown below.



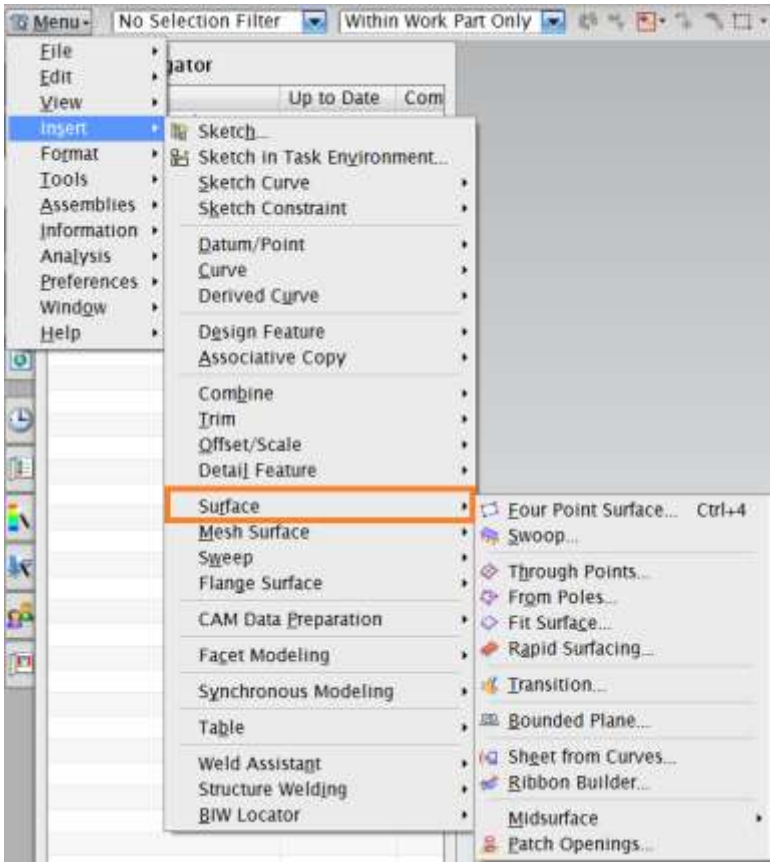
To create **Freeform Features**, you must first need a set of points, curves, edges of sheets or solids, faces of sheets or solids, or other objects. The following topics will cover some of the methods that you can use to create solids using some of the freeform features.

7.1 OVERVIEW

The Freeform Features in NX9.0 are grouped under various menus and located in the **INSERT** menu. There are a lot of ways in which you can create **Freeform Features** from the existing geometry you have like points, edges, curves, etc. A few of the menus are discussed below.

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 may be able to use one of these three options to build the feature from the given points.

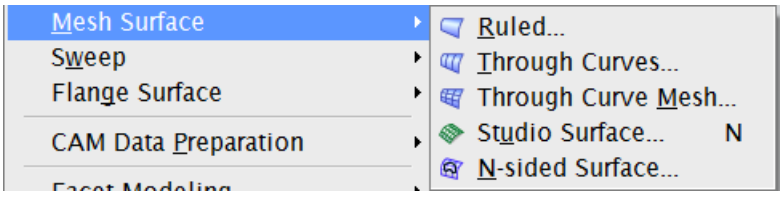


➢ Click on **INSERT → SURFACE**

- **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.

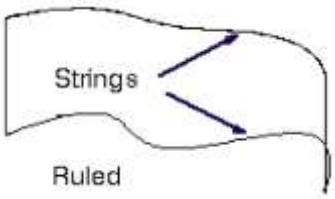
7.1.2 Creating Freeform Features from Section Strings

If construction geometry contains strings of connected objects (curves and edges), you may be able to use one of these two options to build the feature.

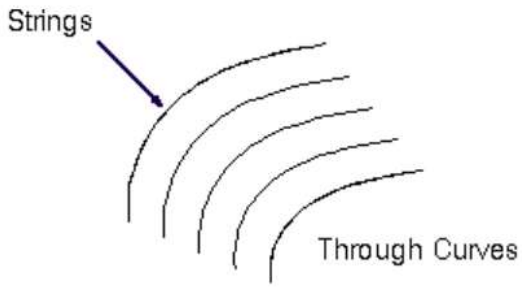


➢ Click on **INSERT → MESH SURFACE**

- **Ruled** – Used if the two strings are roughly parallel.

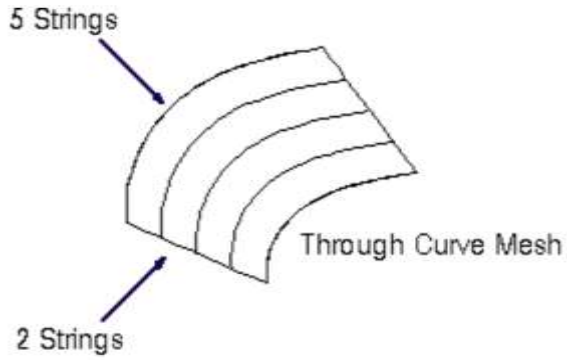


- **Through Curves** – Used if the three or more strings are roughly parallel.



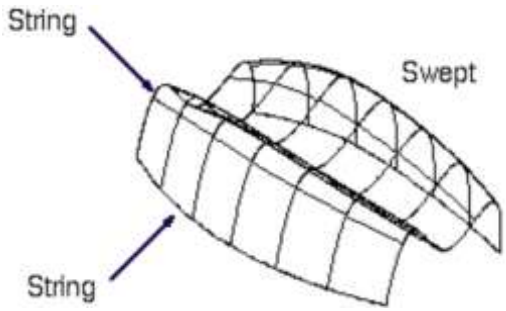
If construction geometry contains two or more strings (curves, faces, edges) that are roughly parallel to each other, and one or more section strings that are roughly perpendicular to the first set of curves (guides), you may be able to use one of these following options to build the feature.

- **Through Curve Mesh** – Used if at least four section strings exist with at least two strings in each direction (parallel and perpendicular).



If the two sections are perpendicular then choose **INSERT → SWEEP**

- **Swept** – Used if at least two section strings are roughly perpendicular.

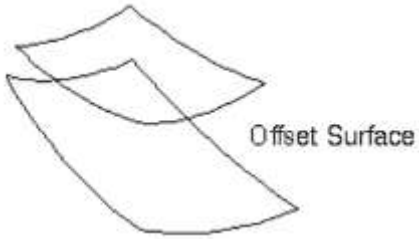


7.1.3 Creating Freeform Features from Faces

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

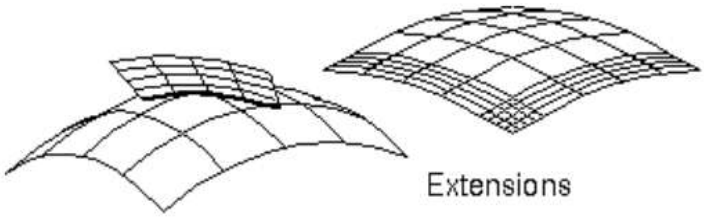
- Click on **INSERT → OFFSET/SCALE**

- Offset Surface – Use this option if you have a face to offset.



- Click on the **INSERT → FLNAGE SURFACE → EXTENSION**

- Extension – Use this option if you have a face and edges, edge curves, or curves on the face.



7.2 FREEFORM FEATURE MODELING

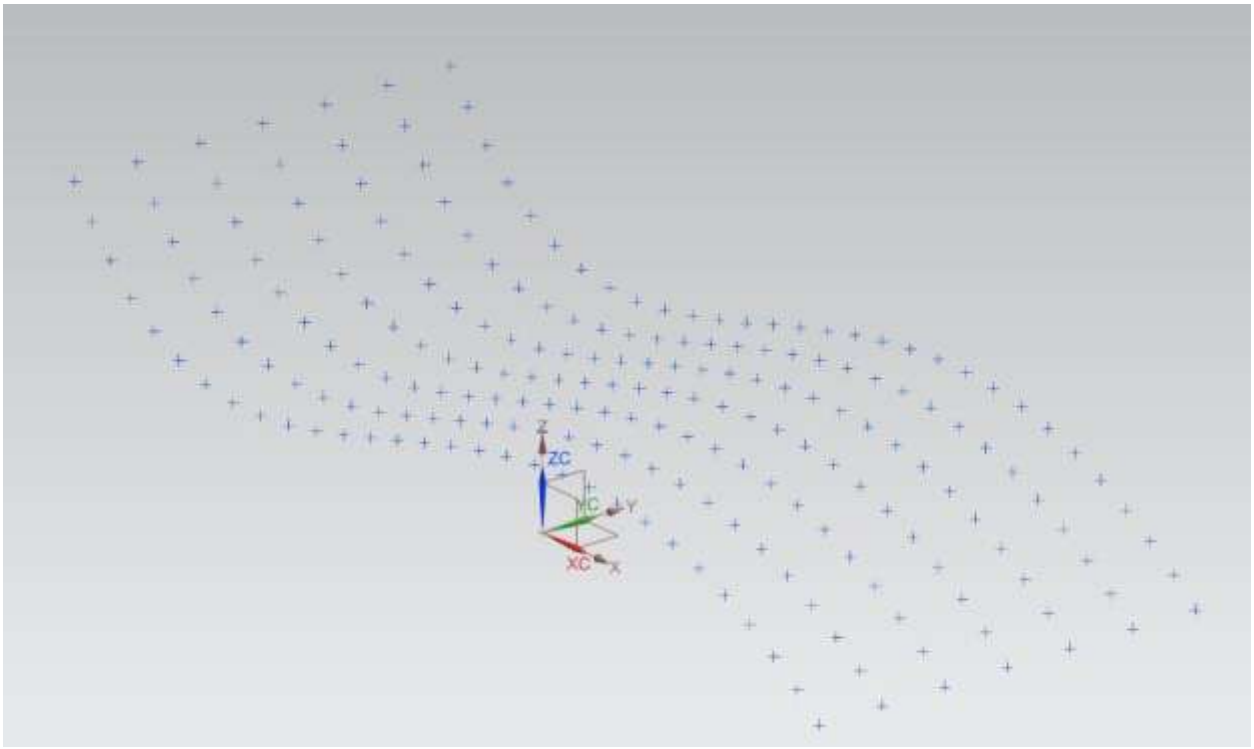
Let us do some freeform modeling on structured points, a point cloud, curves and faces. Structured points are a set of point's defined rows and columns. A point cloud has a set of scattered points that form a cloud.

7.2.1 Modeling with points

- ***Open*** the file **freeform_thrupoints.prt**

- Right-click on the **Toolbars** and make sure the **SURFACE Toolbar** is checked

You will see seven rows with many points.

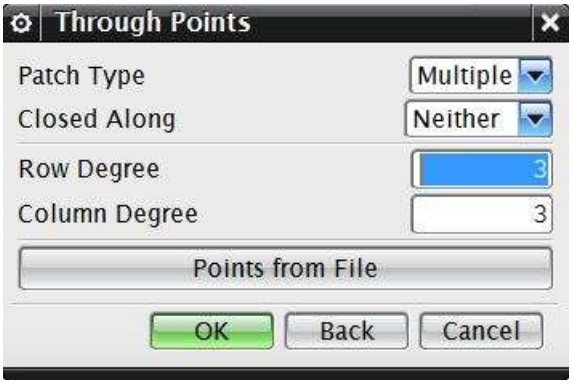


- Choose **INSERT → SURFACE → THROUGH POINTS**

OR

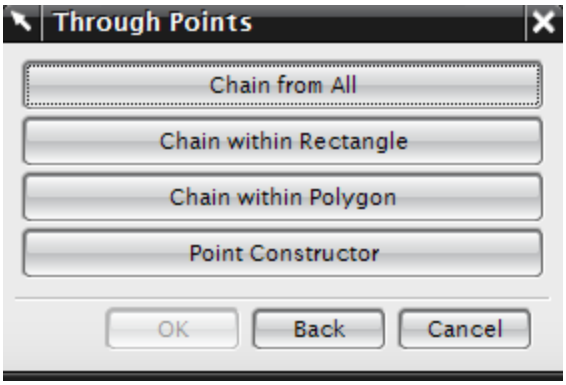
- Click on the Icon in the Toolbar

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



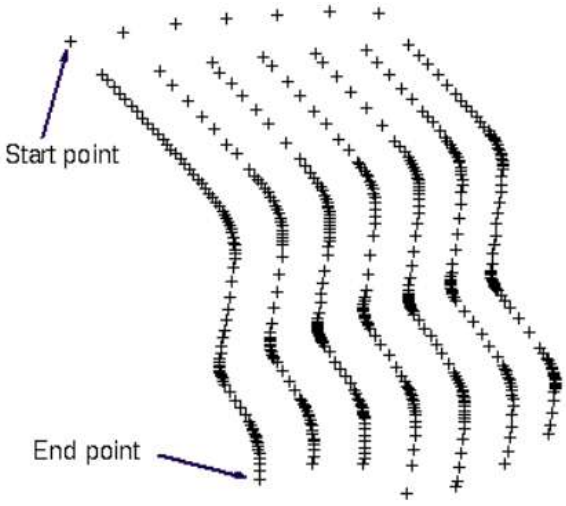
- For **Patch Type**, select ***Multiple***
- For **Closed Along**, select ***Neither***
- For **Row Degree** and **Column Degree**, enter **3.**
- Click **OK**

The next dialogue box will be as shown.

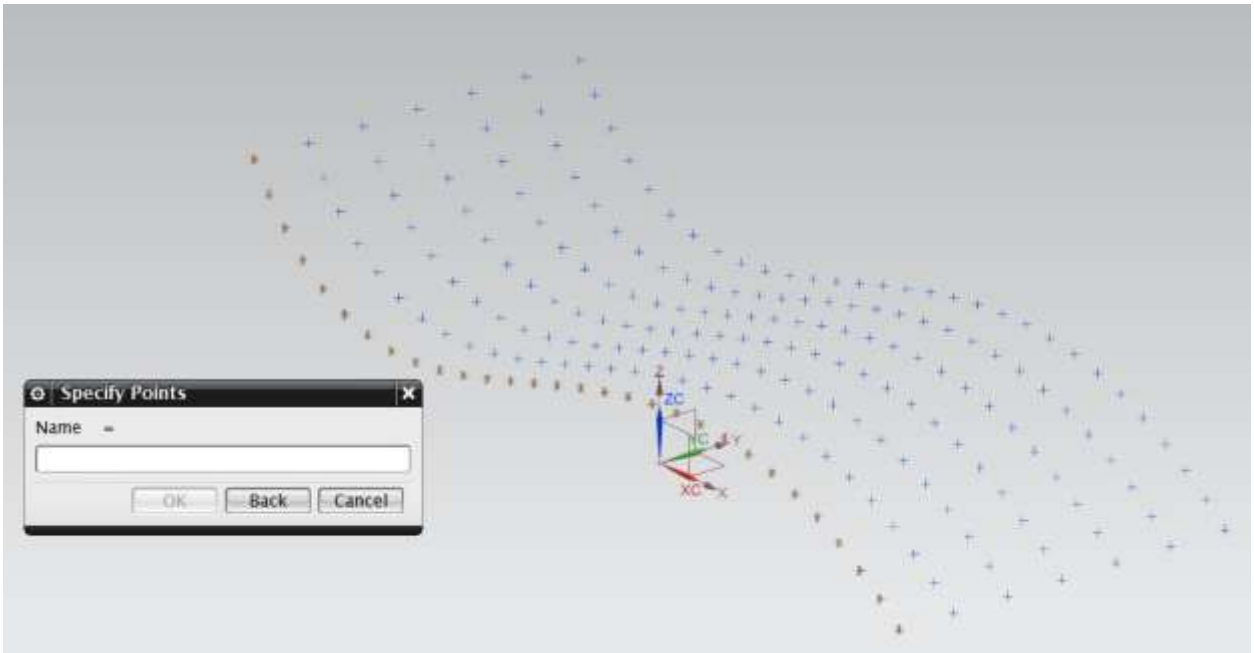


- Click **CHAIN FROM ALL**

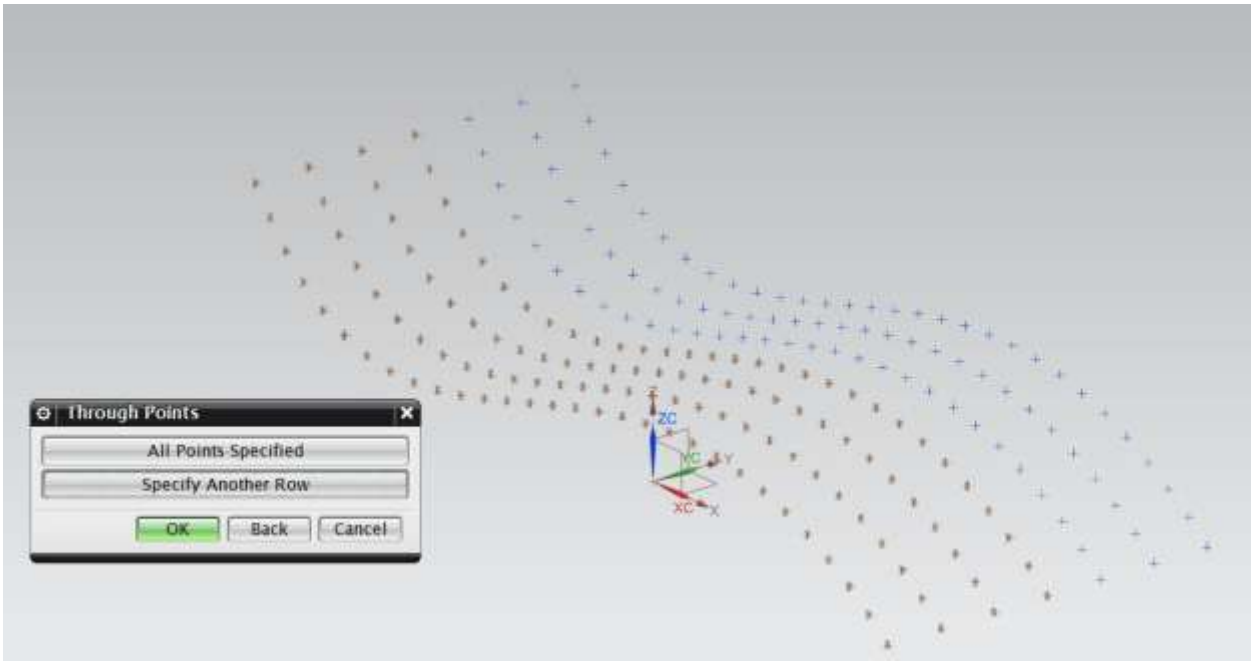
- Select the top starting point and the bottom ending point of the left most row as shown in the following figure



The first row of points will be highlighted.



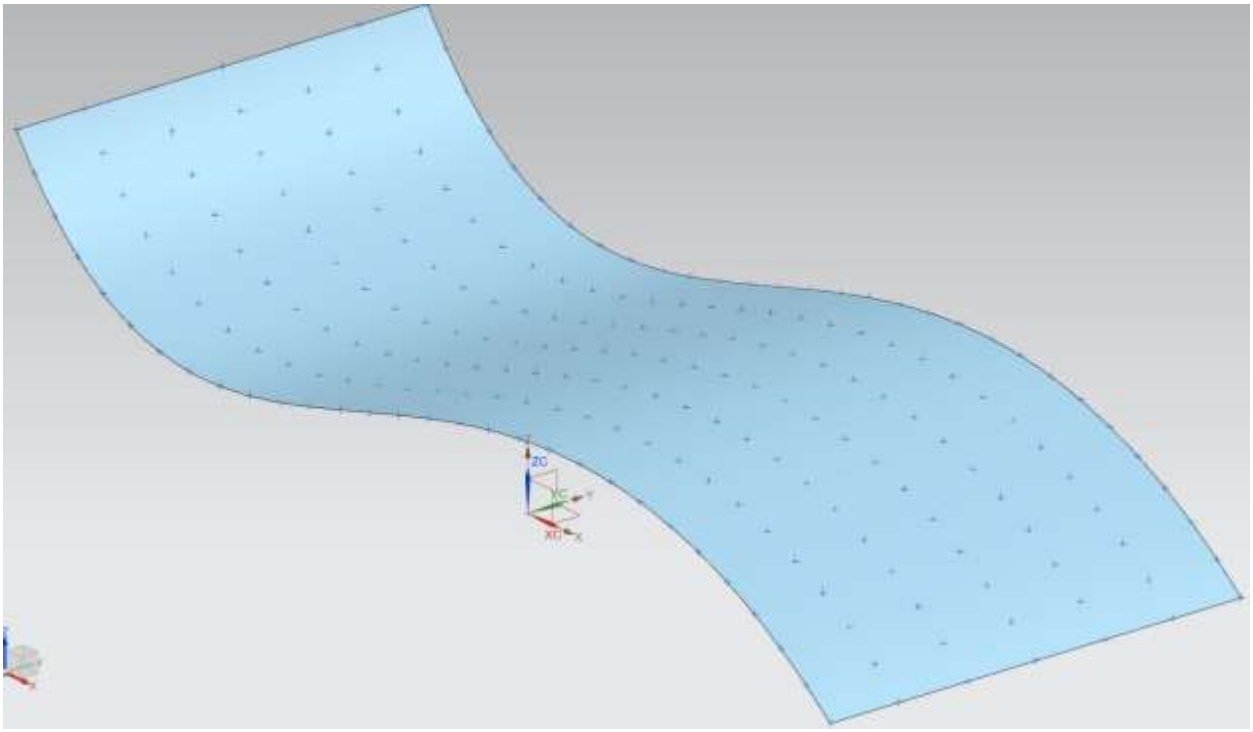
Repeat the same procedure to select the first four strings of points. After that, a window should pop up asking if all points are specified or if you want to specify another row.



- 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

- Click on the **Shaded** icon

You will see the surface as shown below.

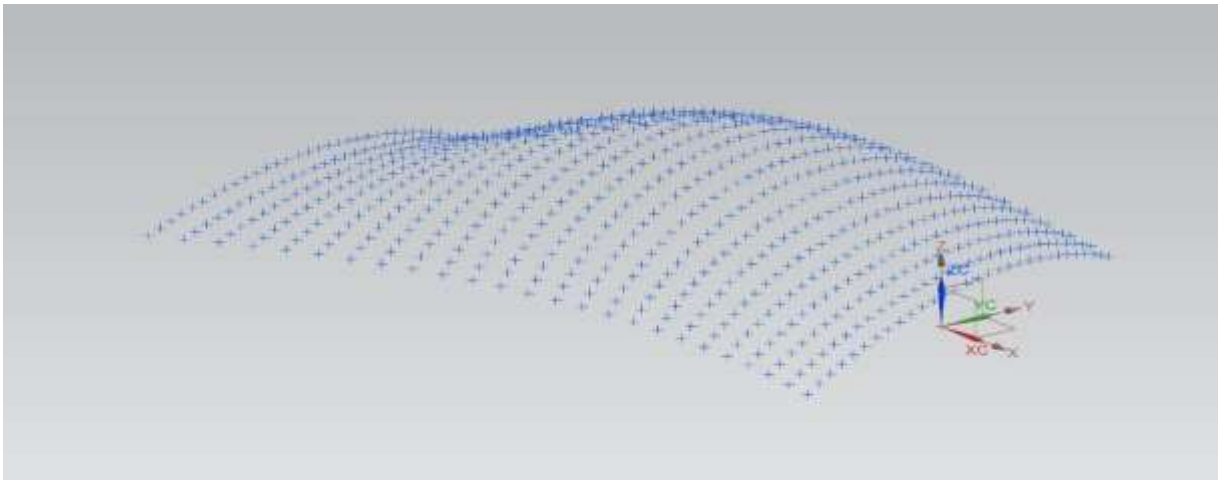


Do NOT save these files.

7.2.2 Modeling with a point cloud

- **OPEN** the file named **freeform_throughcloud.prt**

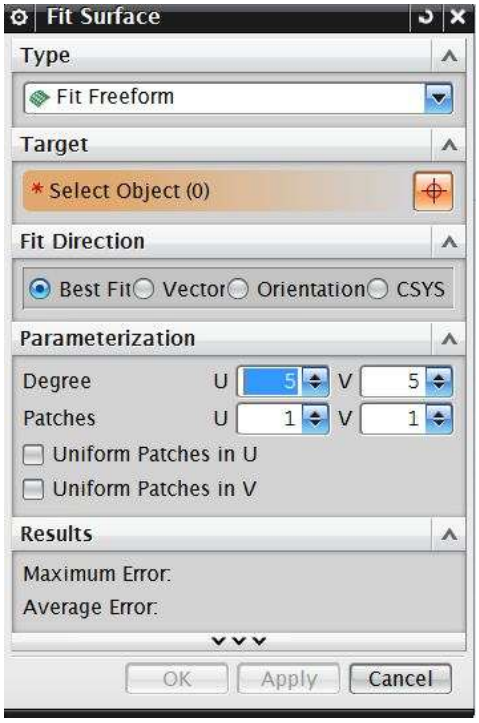
The point cloud will be seen as follows.



- Choose **INSERT → SURFACE → FIT SURFACE**

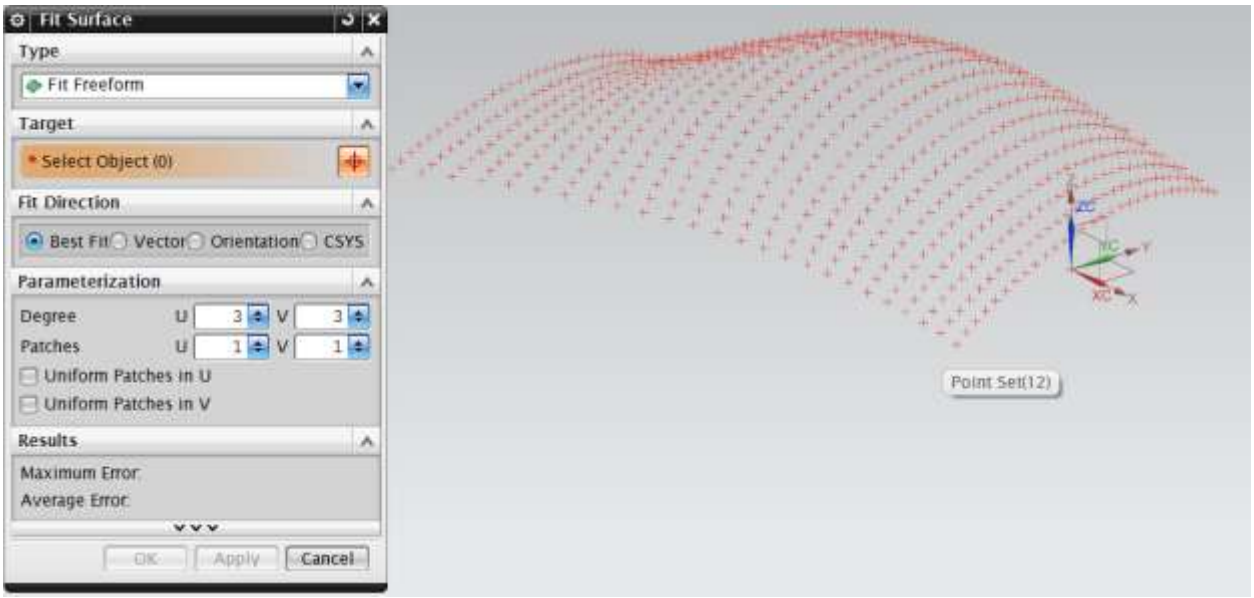
  or click on this icon on the **Surface Toolbar**

The following dialogue box will appear.



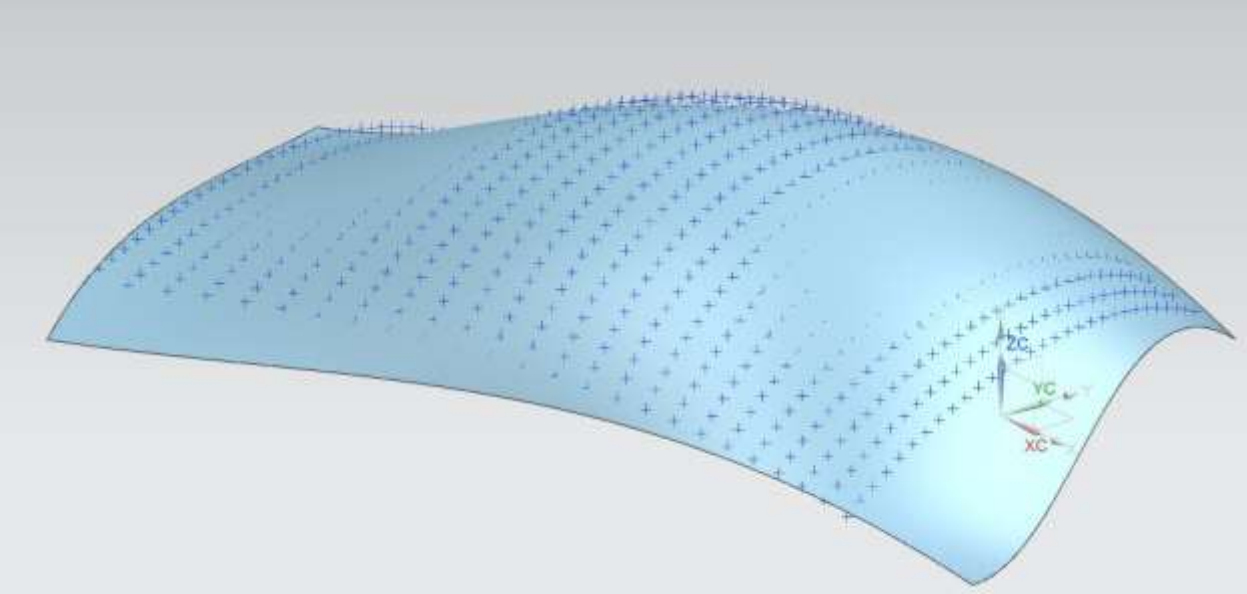
- Select all the points on the screen by clicking on the point cloud.

After you have selected the points, the screen will look like the following.



- In the **Coordinate System** drop-down menu, choose ***BEST FIT*** for the **Coordinate System**. This matches the point cloud coordinate system with original system
- Keep the default values for **U and V Degree as 3**
- Click **OK**
- Change the **VIEW** to **Shaded** to see the model as a solid

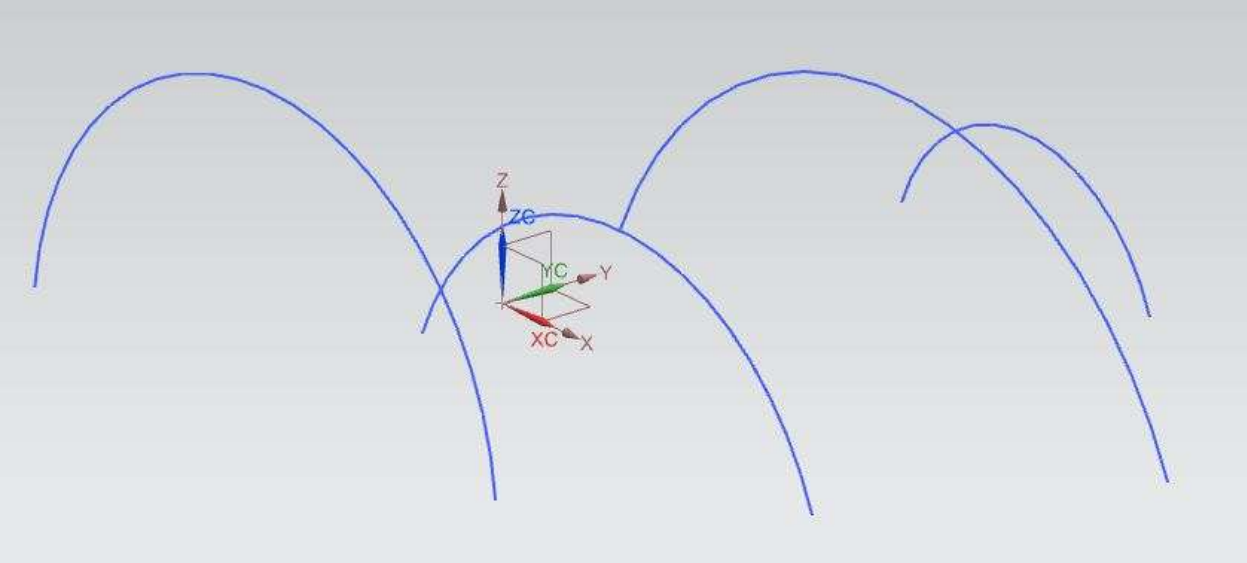
The final sheet will look like the following. Again, do **NOT** save these files.



7.2.3 Modeling with curves

- **OPEN** the file named **freeform_thrucurves_parameter.prt**

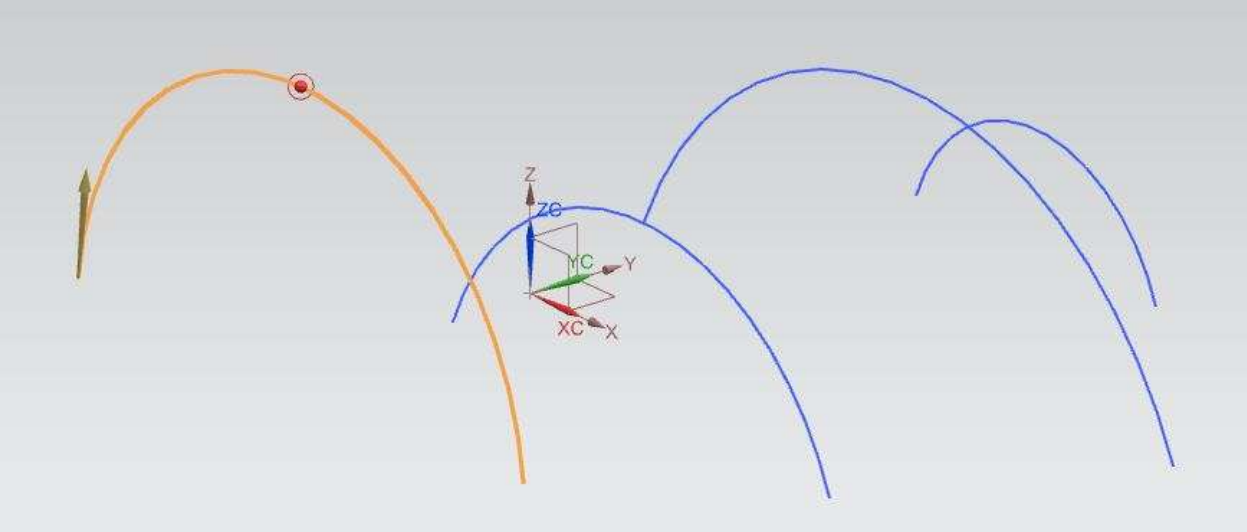
The curves will be seen as in the figure on the right.



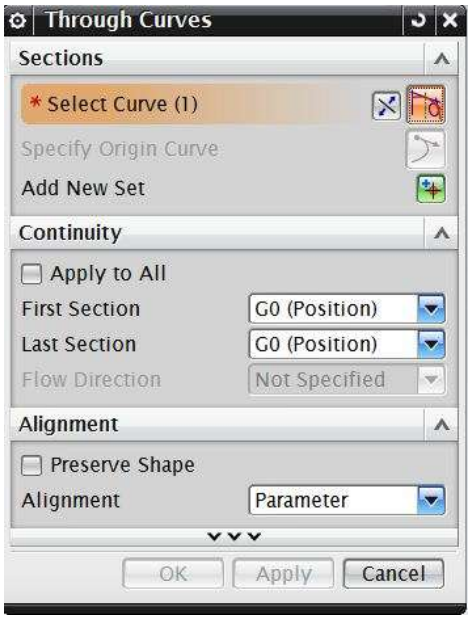
- Choose **INSERT → MESH SURFACE → THROUGH CURVES** or click on this Icon on the Toolbar

- 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.

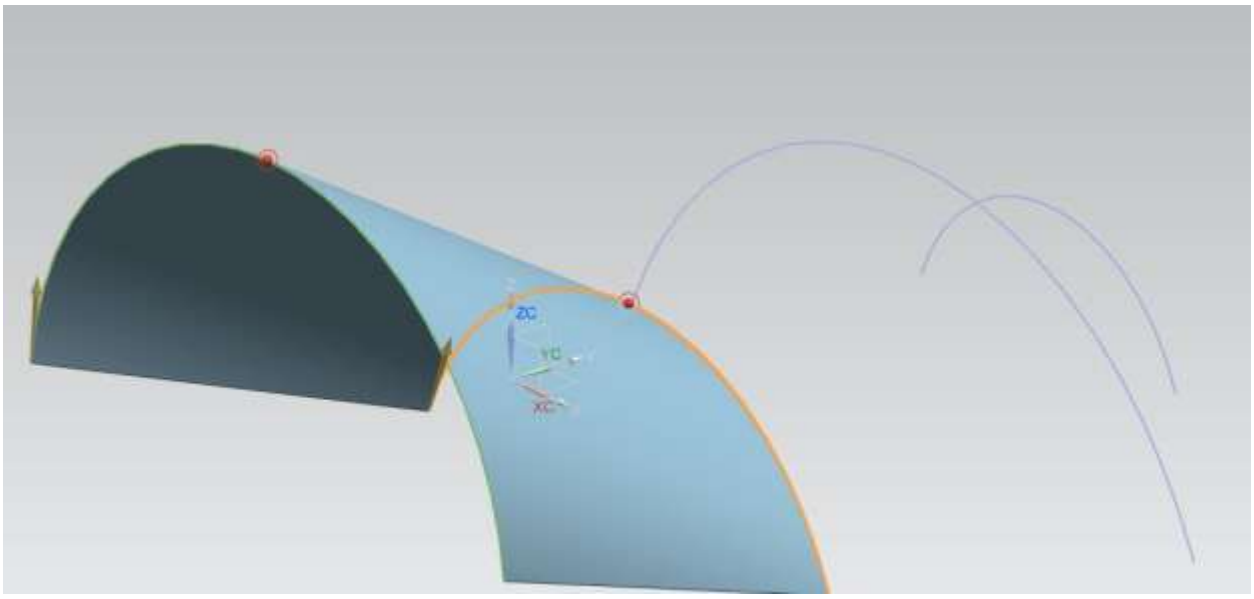


You should see the screen as shown below.

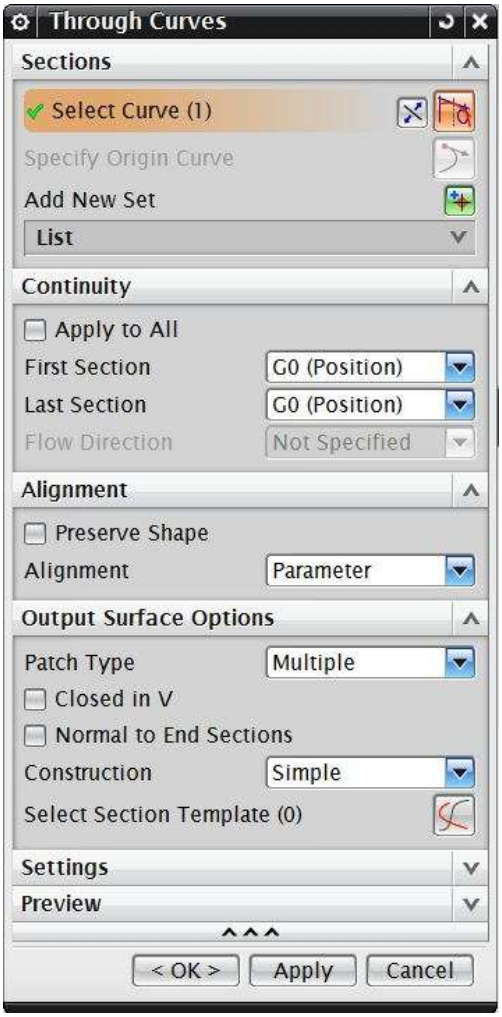


- Click the middle mouse button **MB2**

- 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** after selecting each curve.



In the **Alignment** and **Output Surface** dialog box, choose the following:

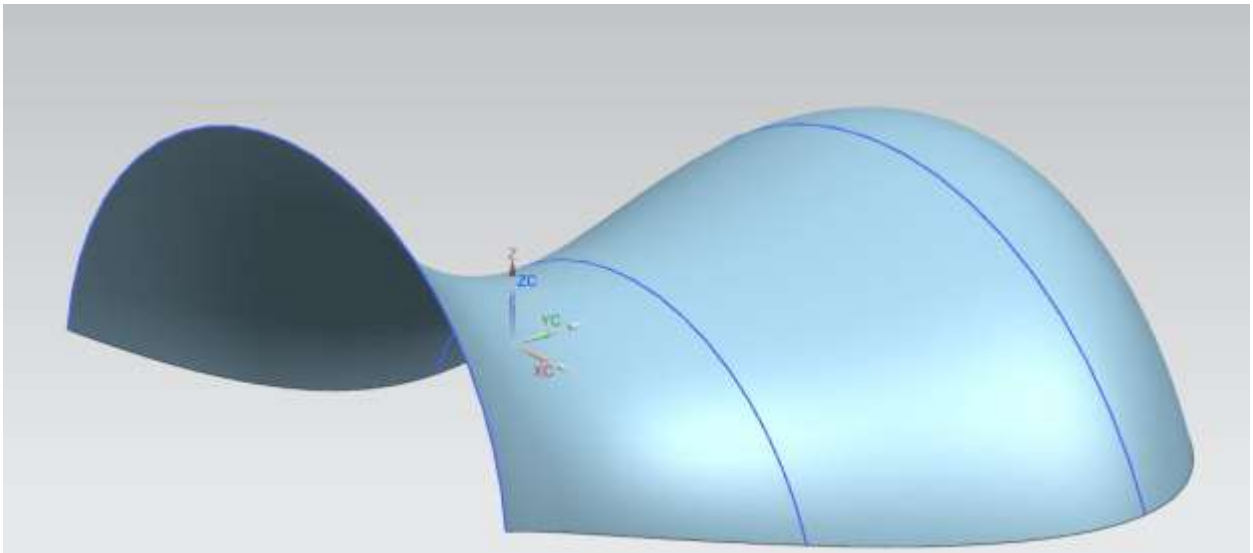
- For **Patch Type**, choose ***Single***
- For **Alignment**, choose ***Parameter***
- For **Construction**, choose ***Simple***

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

- Click **OK**

If you are not able to see the surface then click on the Shade icon on the toolbar

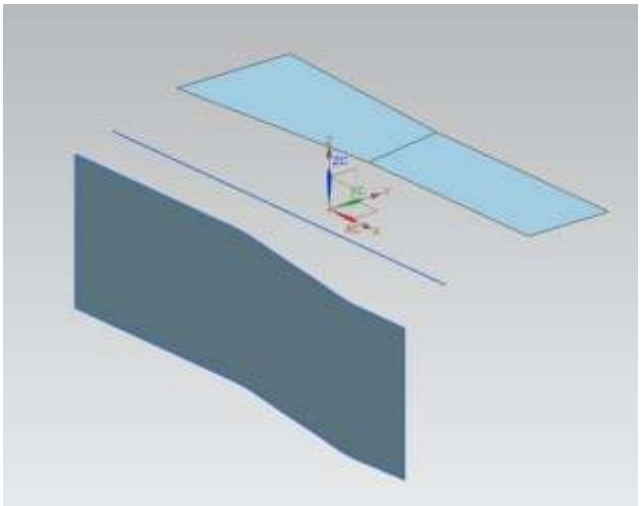
The following curved surface will be generated. Again, do not save the file.



7.2.4 Modeling with curves and faces

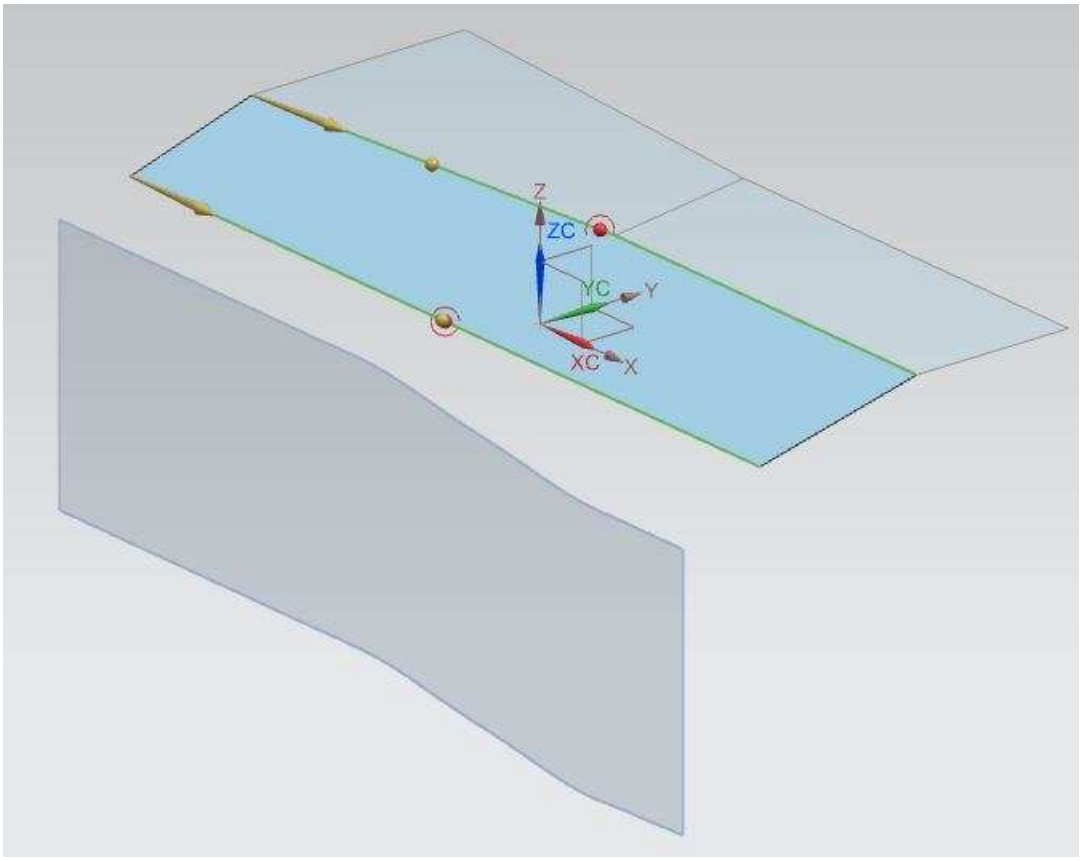
Open the file named **freeform_thrucurves_faces.prt**

The curve and faces will be seen as follows.



- Choose **INSERT → MESH SURFACE → THROUGH CURVES**
- Select the left edge of the top plane and click **MB2**
- Now select the middle edge and click **MB2**
- Now select the line
- In the Dialog box, under the **Alignment** section, uncheck the **Preserve Shape** check box

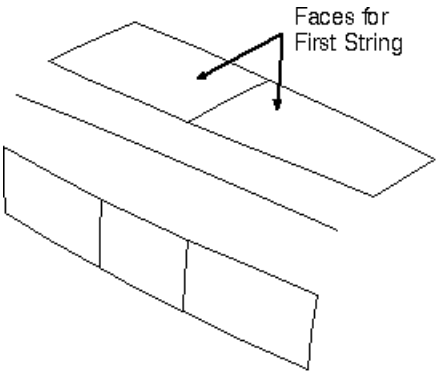
You would get the following shape displayed on screen.

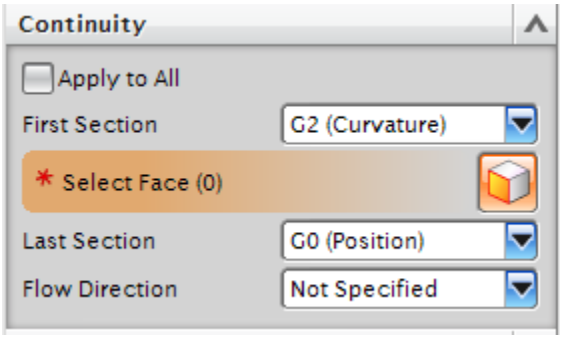


Make sure that all the arrows are pointing in the same direction. If not, click **CANCEL** and reselect the strings.

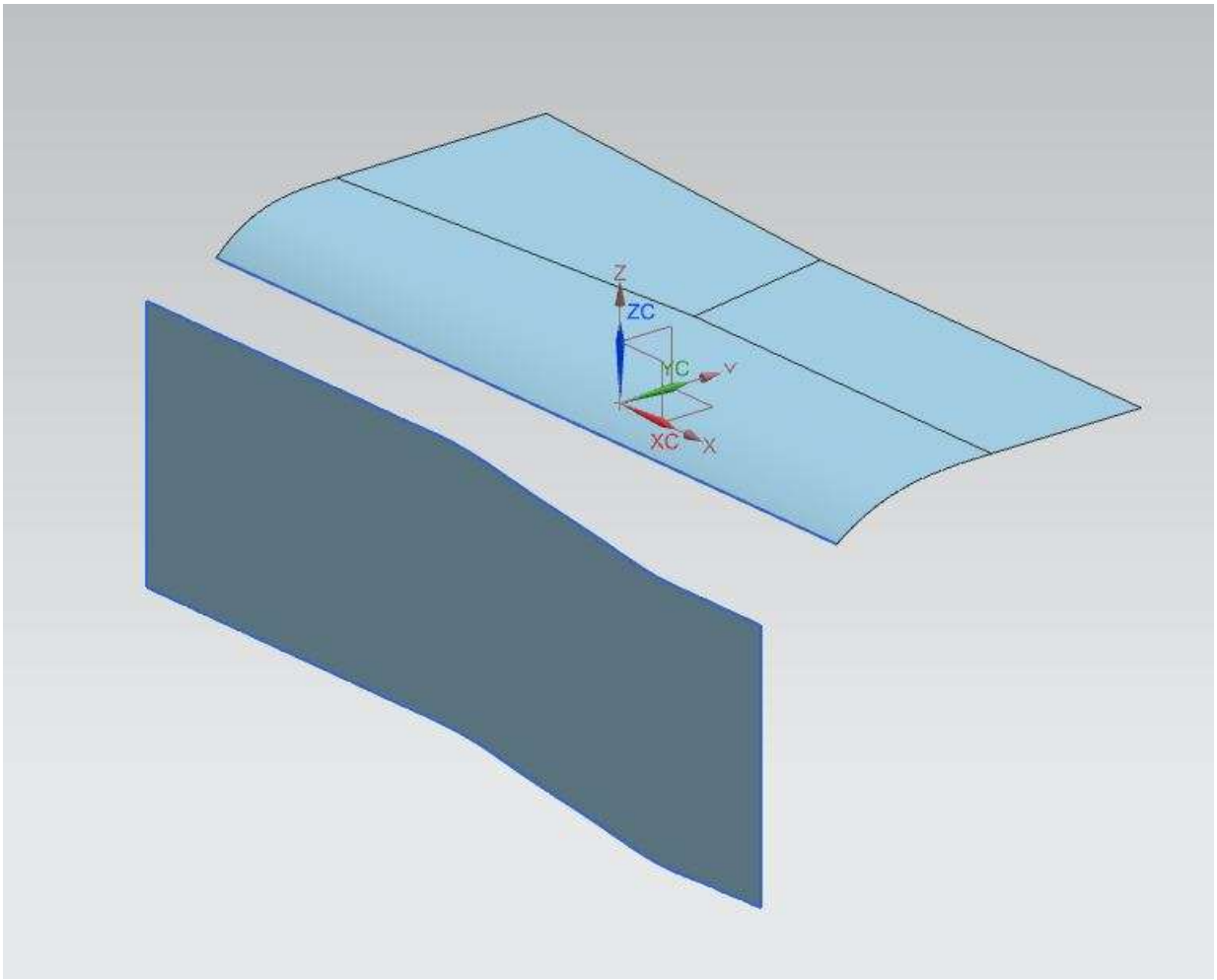
- In the **Alignment** dialog box choose **Parameter**

- In the **Continuity** dialog box select **G2 (Curvature)** option and select the two faces of the top plane as shown



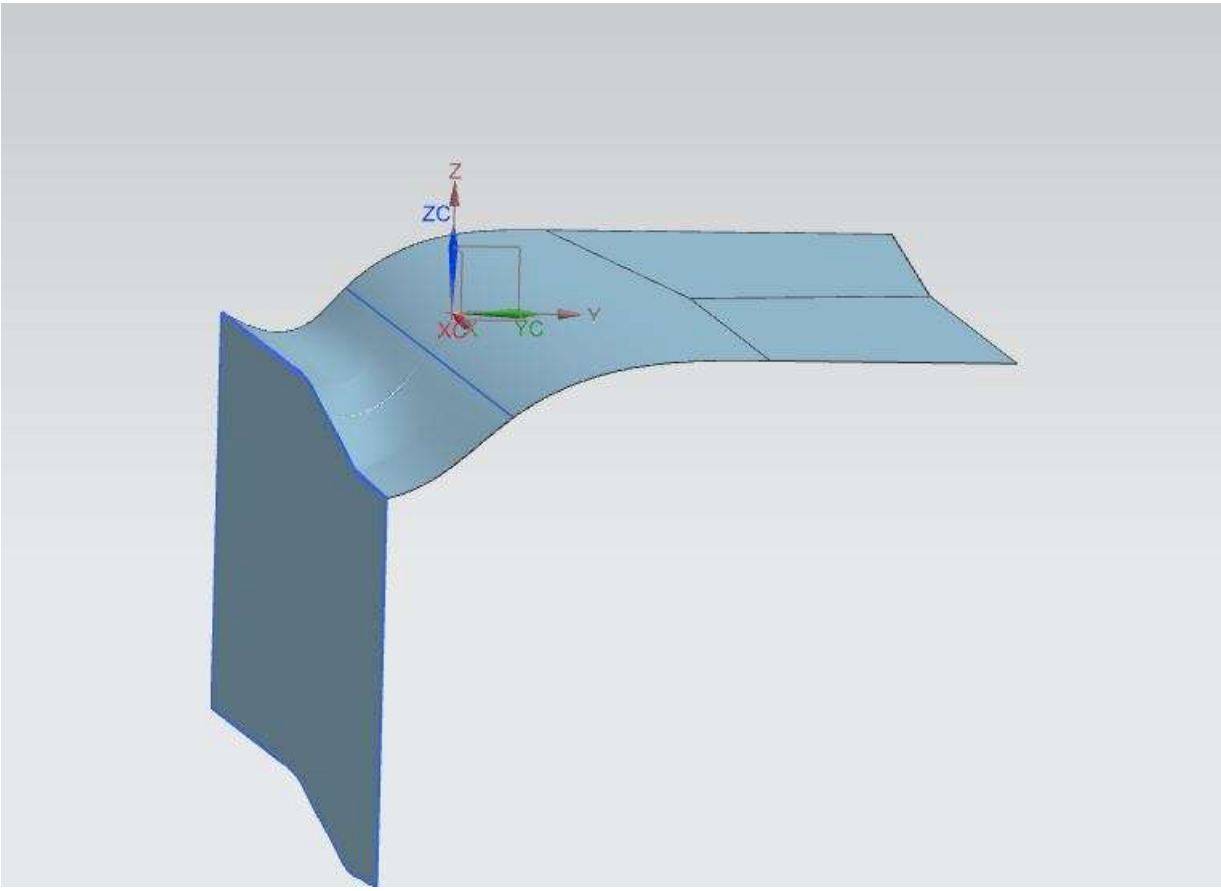


- Click **APPLY**



- Now select the middle edge and click **MB2**
- Select the edge of the lower plane and click **MB2**
- Click **MB2** to finish the curve selection
- Change the option to **G2 (Curvature)** in the **Continuity** dialog box
- Select the face of the upper surface(newly created and click **MB2**
- Select the bottom face
- Click **APPLY** and then click **CANCEL**

The final curve will be seen as shown below. Do not save the files.



7.3 EXERCISE - MODEL A MOUSE

Model a computer mouse similar to the one shown below or use your imagination to model a different mouse. As a hint, create some boundary curves on different planes and use them to form freeform surfaces. Use these quilt surfaces to create the solid. Add and subtract blocks and pads to attach the accessories like buttons.

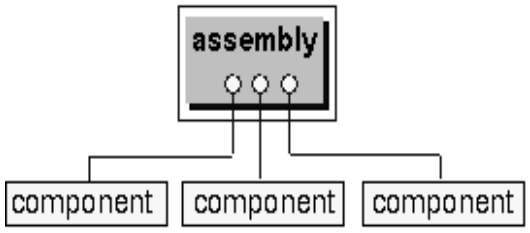
CHAPTER 8 – ASSEMBLY MODELING

This chapter introduces 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 be learning 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.

8.1 OVERVIEW

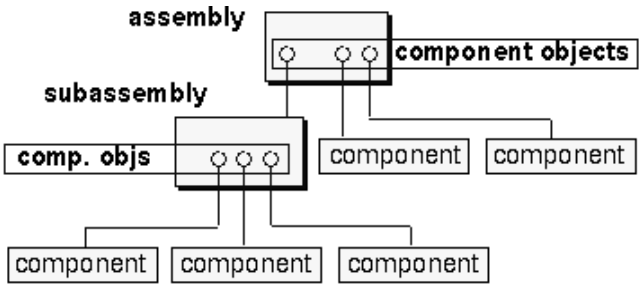
NX9.0 ***Assembly*** is a part file that contains the individual parts. They are added to the part file in such a way that the parts are virtually in the assembly and linked to the original part. This eliminates the need for creating separate memory space for the individual parts in the computer. All the parts are selectable and can be used in the design process for information and mating to insure a perfect fit as intended by the designers. The following figure is a schematic, which shows how components are added to make an assembly.



8.2 TERMINOLOGIES

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.

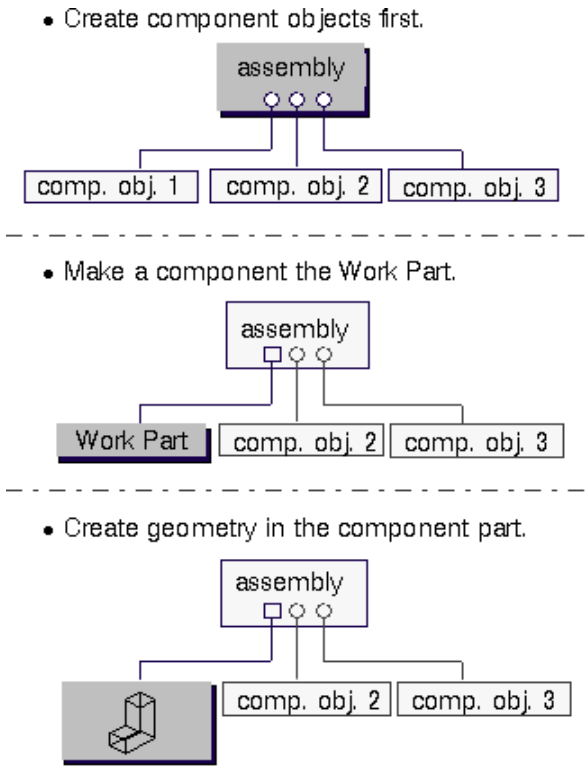
8.3 ASSEMBLY MODELS

There are two basic ways of creating any assembly model.

- Top-Down Approach
- Bottom-Up Approach

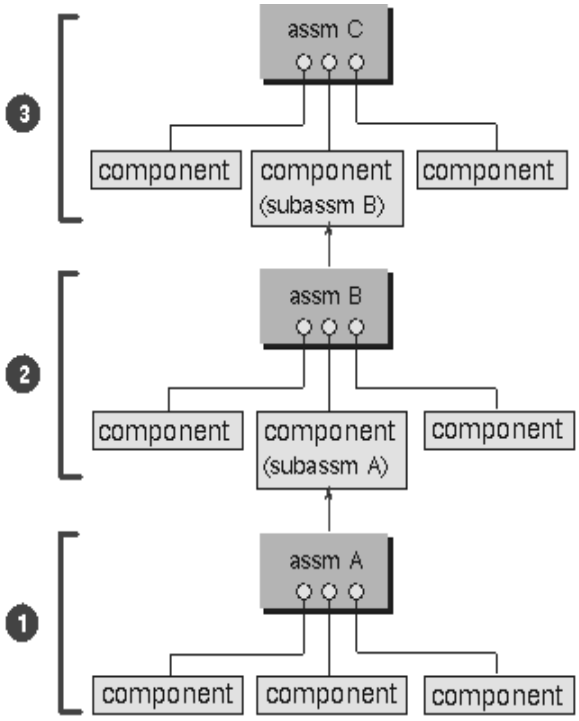
8.3.1 Top-Down Approach

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.



8.3.2 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.

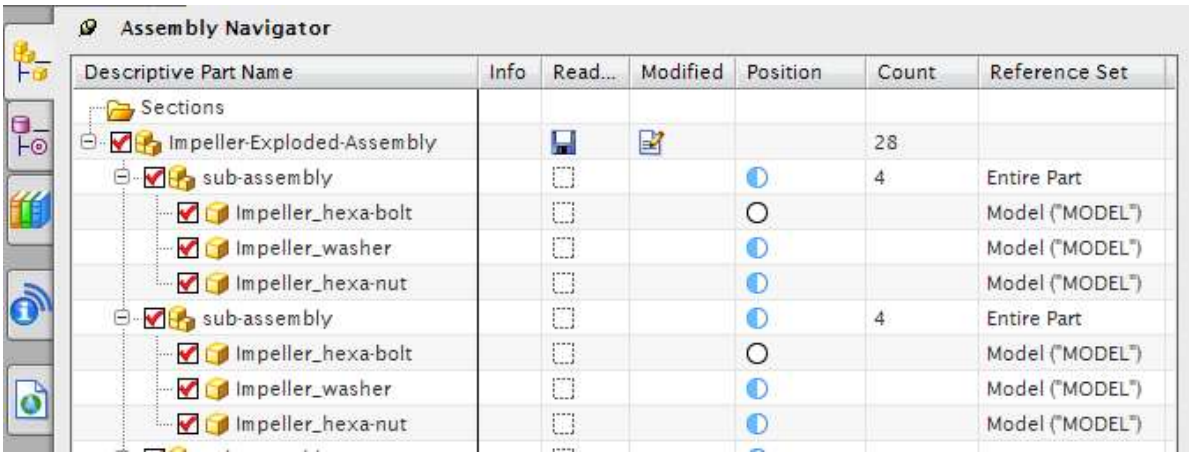


8.3.3 Mixing and Matching

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

8.4 ASSEMBLY NAVIGATOR

The ***Assembly Navigator*** is located on top of the ***Part Navigator*** in the **Resource Bar** on the left of the screen. The navigator shows 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, the position,*** which lets you know whether the part is constrained using assembly constraints or mating condition, and the ***reference set***. Following is a list of interpretation of the *Position* of the components.



● - Indicates a fully constrained component

● - Indicates a fully mated component

- (Fixed) Indicates that all the degrees of freedom are constrained

◐ - Indicates partially constrained component

◐ - Indicates partially mated component

○ - Indicates that the component is not constrained or mated

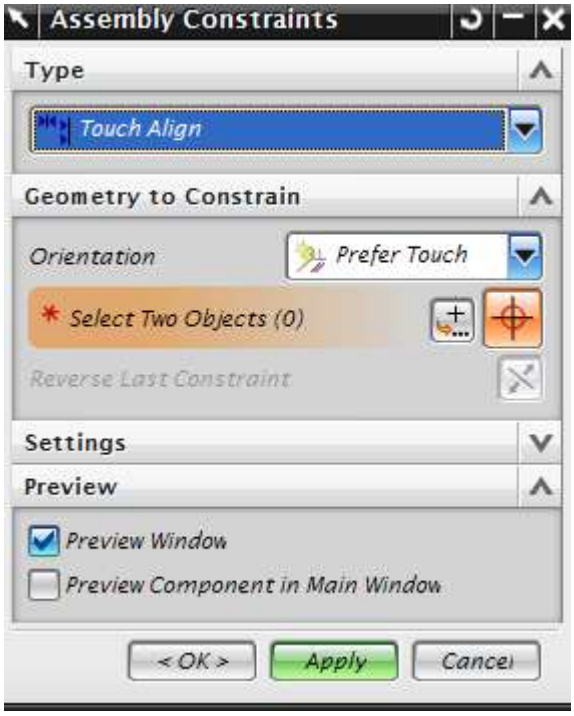
8.5 MATING CONDITIONS

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 shown below :

- **Touch/Align** – Planar objects selected to align will be coplanar but the normals to the planes will point in the same direction. Centerlines of cylindrical objects will be in line with each other.
- **Angle** – This fixes a constant angle between the two object entities chosen on the components to be assembled.
- **Bond –** Creates a weld and welds components together to move as single object.
- **Parallel** – Objects selected will be parallel to each other.

- **Perpendicular –** Objects selected will be perpendicular to each other.
- **Center** – Objects will be centered between other objects, i.e. locating a cylinder along a slot and centering the cylinder in the slot.
- **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

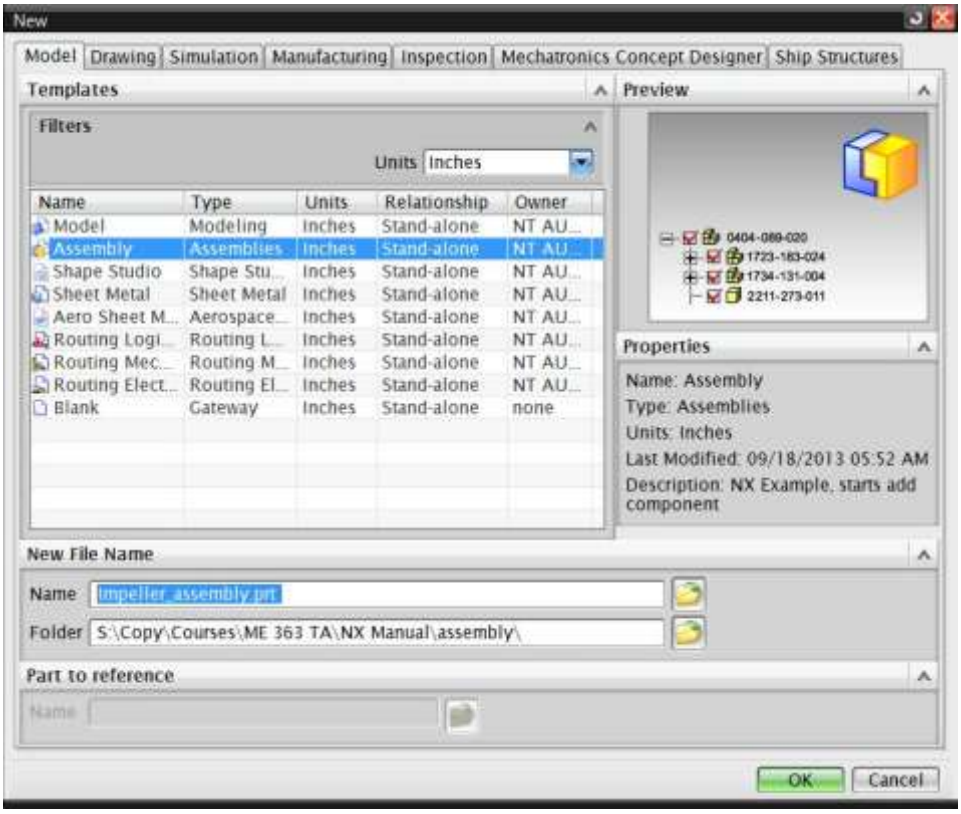
The Mating Conditions dialog box is shown on right.



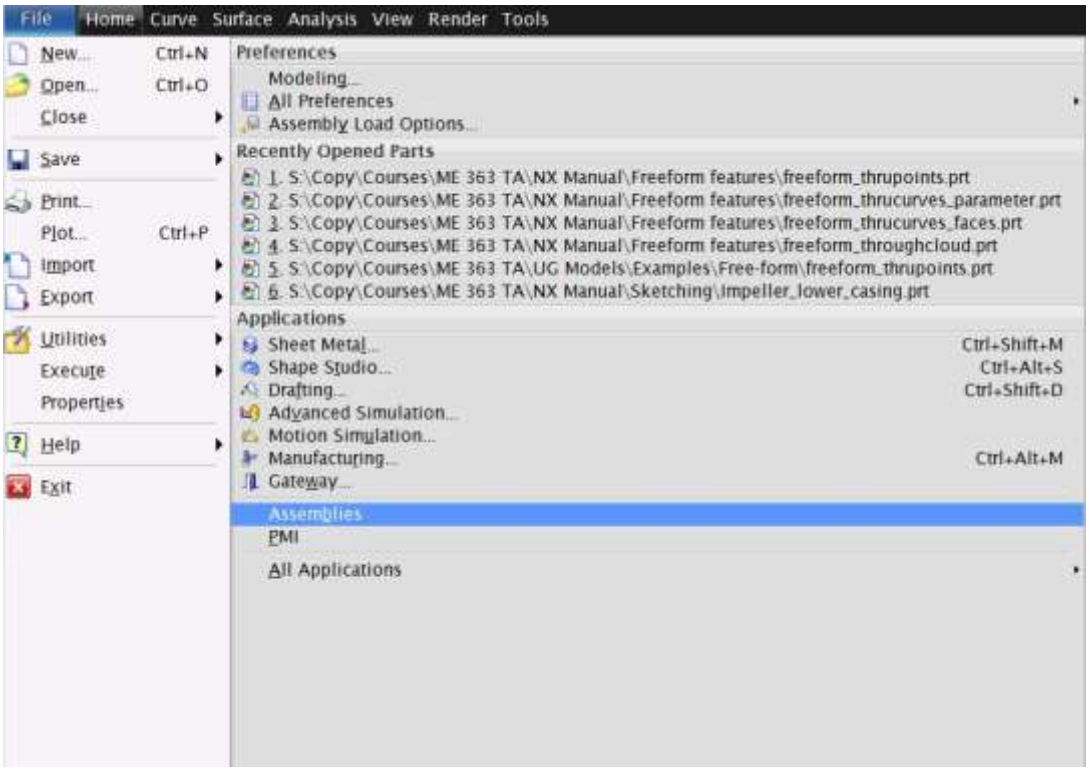
8.6 IMPELLER ASSEMBLY

We will assemble the impeller component objects. All the part files will be provided to you.

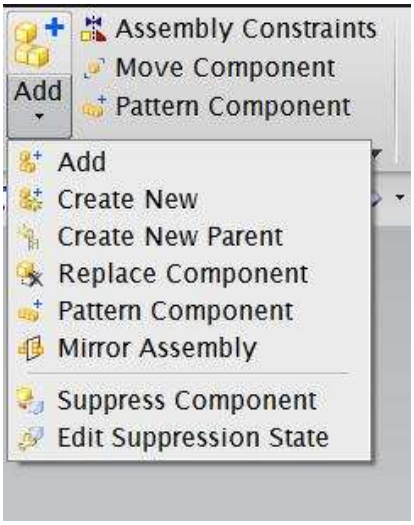
- Create a new file and save it as **Impeller_assembly.prt**
- Choose **NEW** ⟶ **ASSEMBLY**, if you just started NX 9.0 as shown below



➢ Or Click **FILE⟶ ASSEMBLIES** as shown below



➢ The **HOME** menu bar will now display tools for assembly



We will be mostly using the **COMPONENTS** option, which includes:

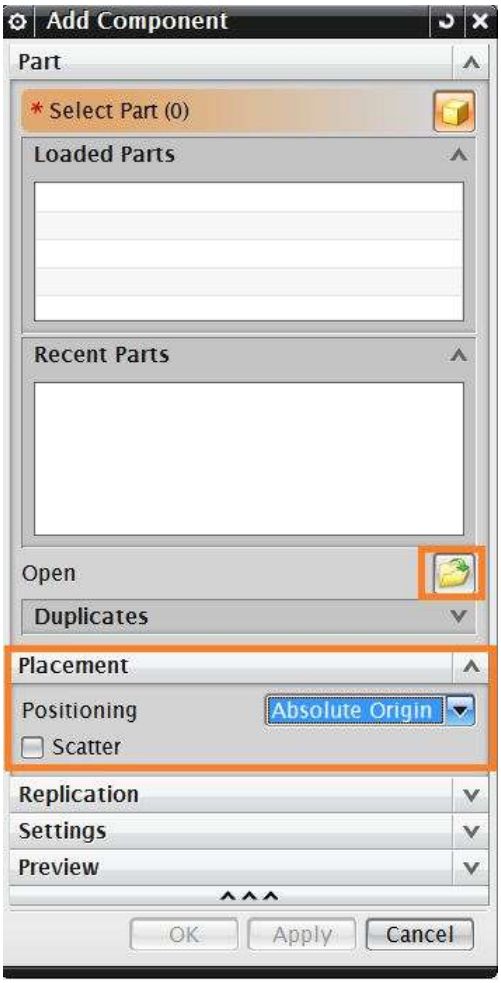
- **ADD** – This option adds new component objects whose part files are already present.

- **CREATE NEW** – This option lets you create new component geometries inside the assembly file in case you are using Top-Down approach of assembly.

The **component position** menus allow you to create assembly constraints and allow you to reposition the components wherever you want them in the assembly.

- **MOVE COMPONENT** – Allows you to move and reposition component objects.

- **ASSEMBLY CONSTRAINTS** – They are used to mate or align the component objects.



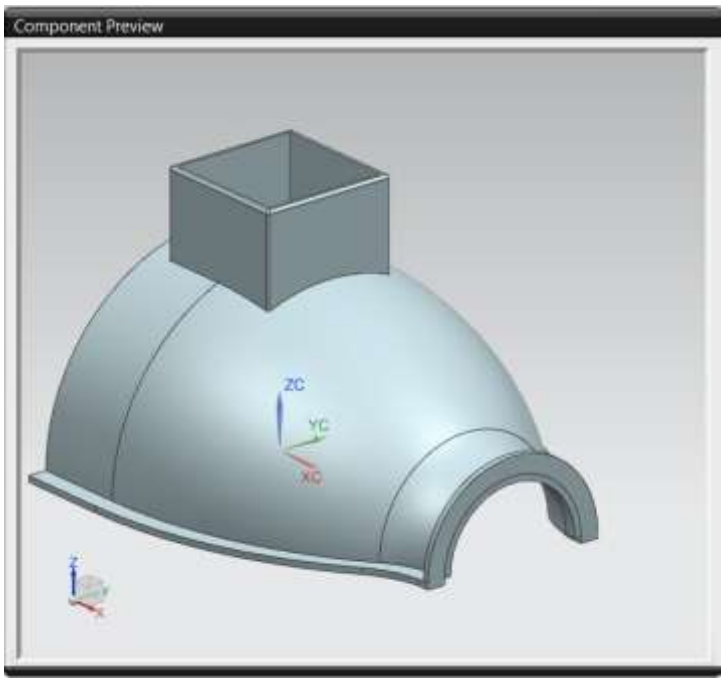
➢ Choose **ADD**

The dialogue box on the right side will pop up. You can select the part files from those existing or else 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 file **Impeller_upper-casing.prt**

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

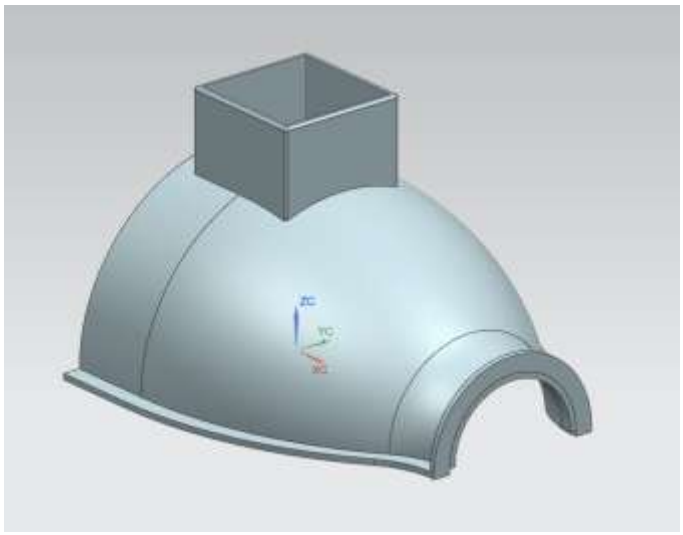
You will see that a small copy of the component object appears in a separate window on the screen as shown in the figure below.



You will need to place this figure initially at certain location. This can be done by changing the ***Positioning*** option in the **PLACEMENT** dialog box to *Absolute Origin* as shown.

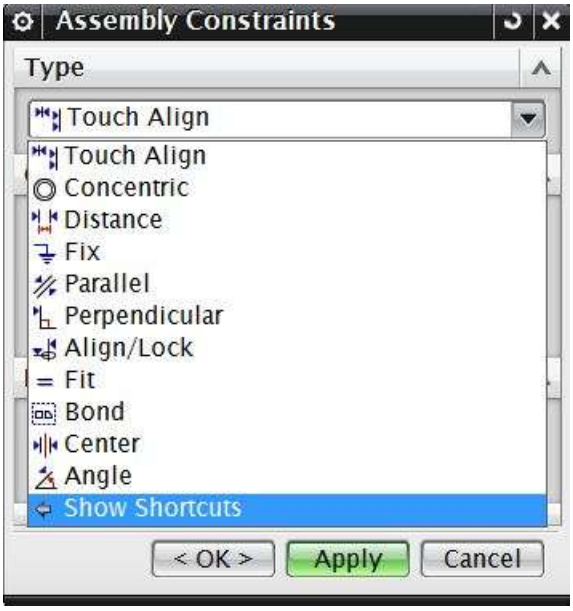
- Click **OK**

You will see the object on the screen as follows:



Now we will add the second component, the lower casing.

- Click on **ADD** in the assembly section
- Select the file **Impeller_lower-casing.prt**
- In the **POISTIONING** dialog box change the option to ***By Constraints***
- Choose **APPLY**



This will show you the added component in a **COMPONENT PREVIEW** window as before.

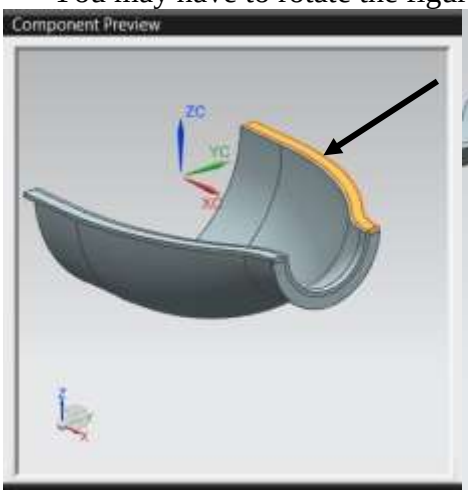
Now let us mate the upper and the lower casing. You can access all the constraints in the drop-down menu in the **Type** dialog box in the **ASSEMBLY CONSTRAINTS** menu.

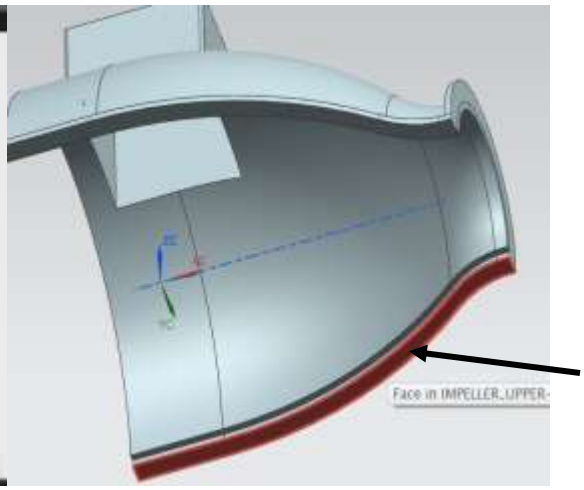
The following dialog box will appear.

Here you can see the different Mating Types, which were explained above in section 8.4.

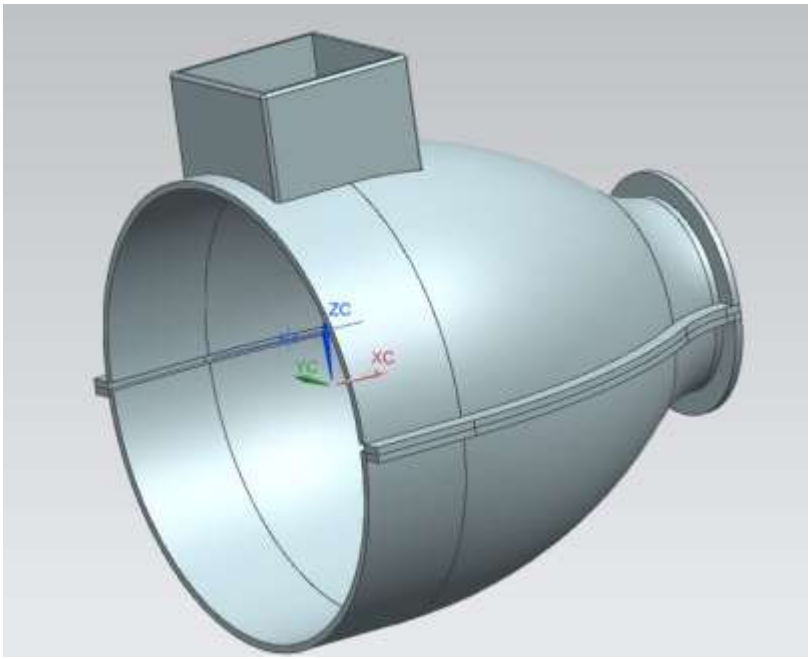
Now let us give the Mate constraint.

- Make sure the ***Touch Align*** icon is selected in the ***TYPE*** dialog box
- First, select the face that the arrow is pointing to in the **Component Preview** window as shown below in the figure on the left figure below.
- Click on the face of the Upper Casing in the main screen as shown in the figure on the right. You may have to rotate the figure in order to select the faces.





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

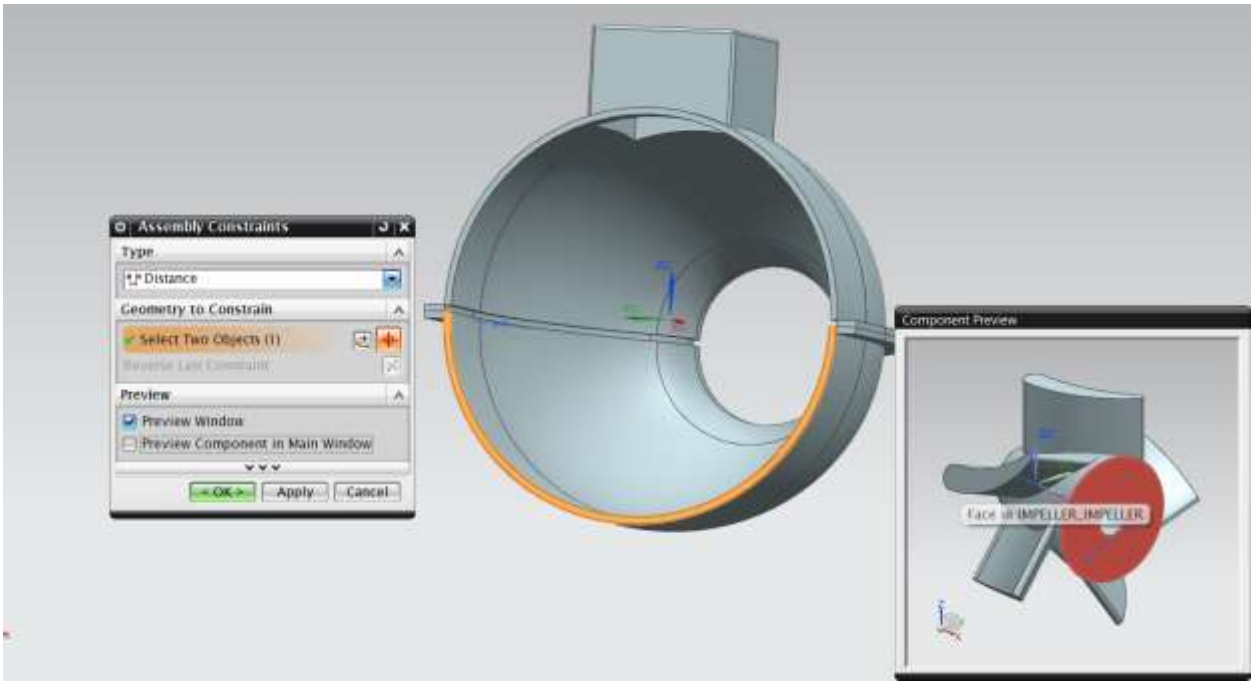


The lower casing is constrained with respect to the upper casing. Now let us add the impeller.

- Choose **ASSEMBLIES → COMPONENTS → ADD COMPONENT**
- Open the file **Impeller_impeller.prt**
- Click **OK** on the dialog box

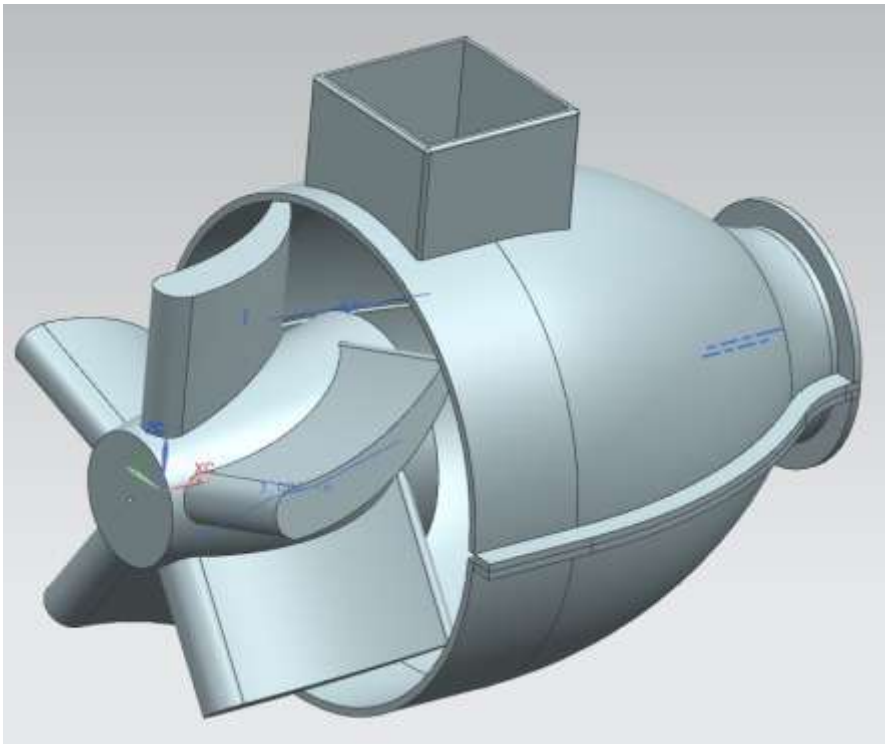
We will apply the ***Distance*** constraint.

- Click on the ***Distance*** icon in the ***TYPE*** dialog box
- Select the two faces, first on the impeller and then on the casing, as shown in the figure below
- Click **OK**
- In the *Distance* dialog box in the **Assembly Constraints** window, enter a value of **3**



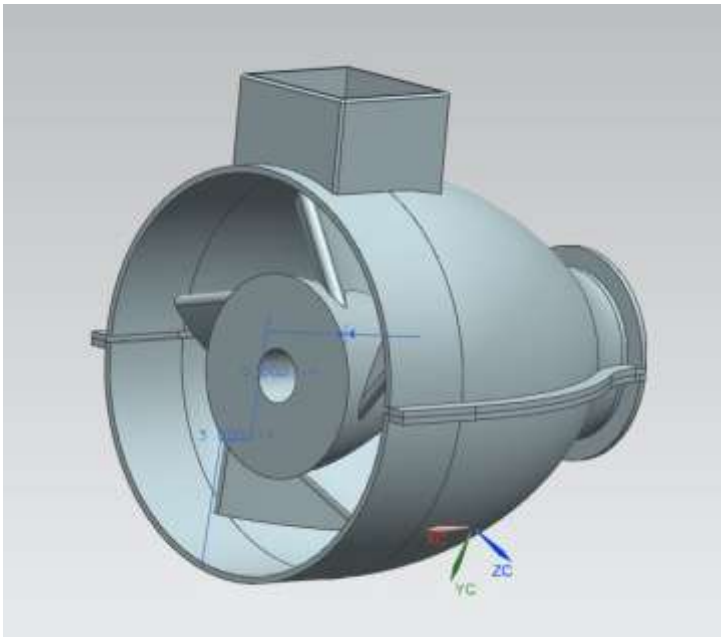
- On the **Assembly Constraints** window, unclick the ***Preview Window*** option

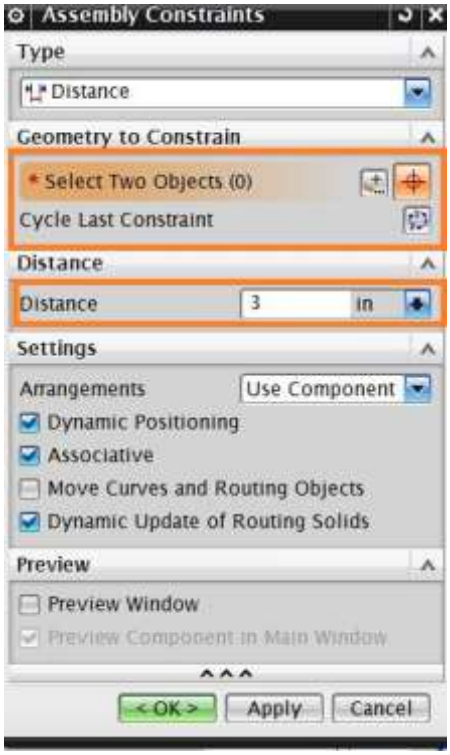
The preview will show the impeller oriented in the direction opposite to the one we want.



- On the **Assembly Constraints** window, click on the ***Cycle Last Constraint*** option in the **Geometry to Constrain** as shown in the figure on right

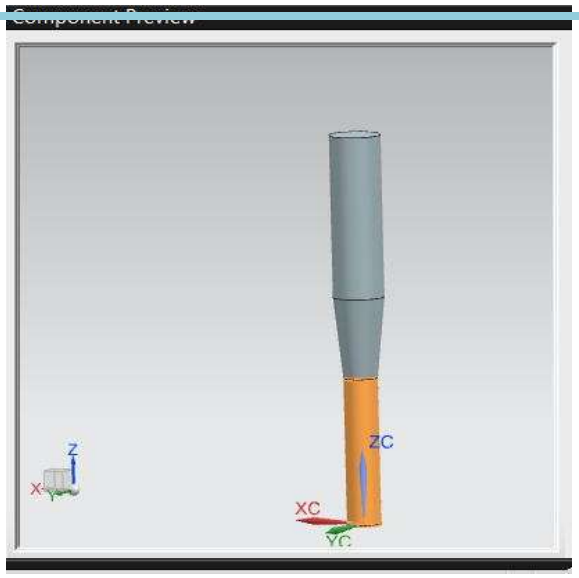
Now the impeller will be oriented in the right direction.

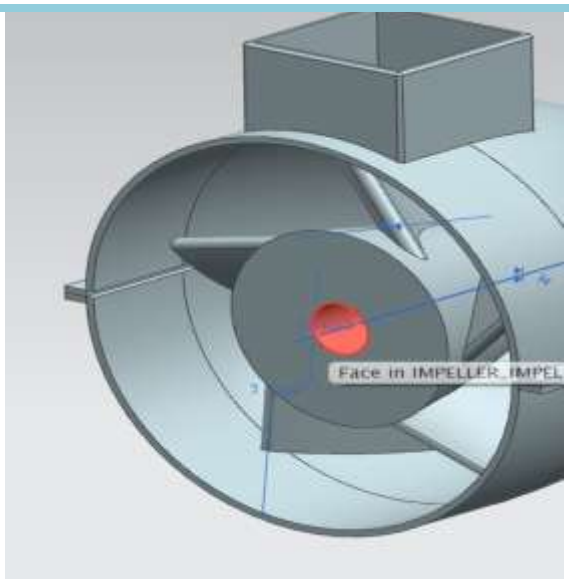




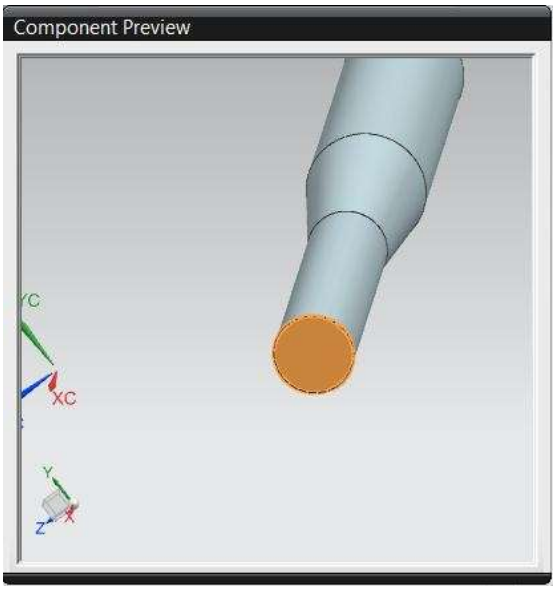
Now we will apply the Center constraint to the model. Save the assembly file. We will now add the shaft.

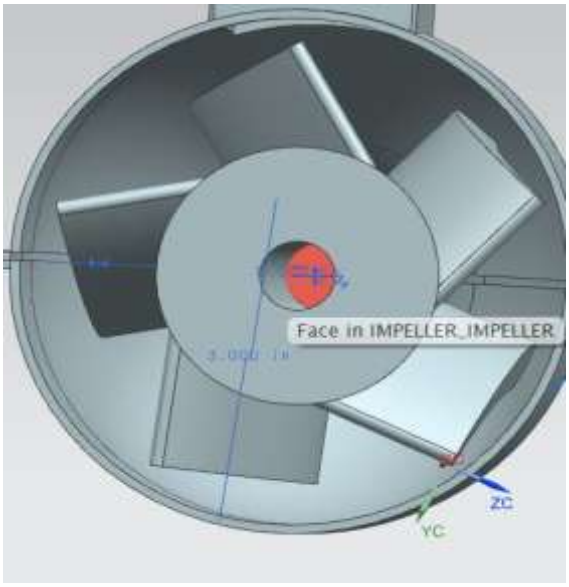
- Click on **ASSEMBLIES → COMPONENTS → ADD COMPONENT**
- Open the file **Impeller_shaft.prt**
- Click **OK** on the dialog box
- Choose the **Touch Align** icon.
- Choose the *Infer Center/Axis* option in the ***Geometry to Constrain*** dialog box in the **Assembly Constraints** window as shown in the figure on right
- Select the two surfaces, first on the shaft in the preview window and then on the impeller on the main screen as shown in the figures below



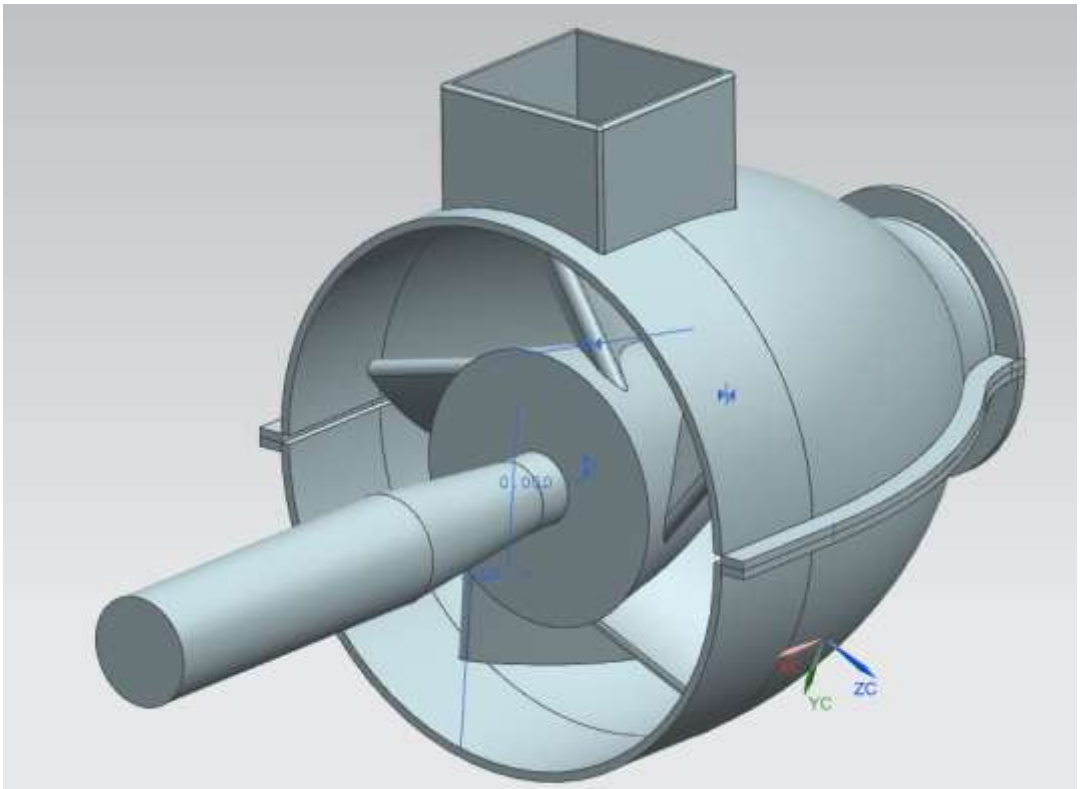


- Choose the **Touch Align** constraint
- First, select the face on the shaft and then select the bottom face of the hole in the impeller as shown.
- Choose **APPLY** and then click **OK**



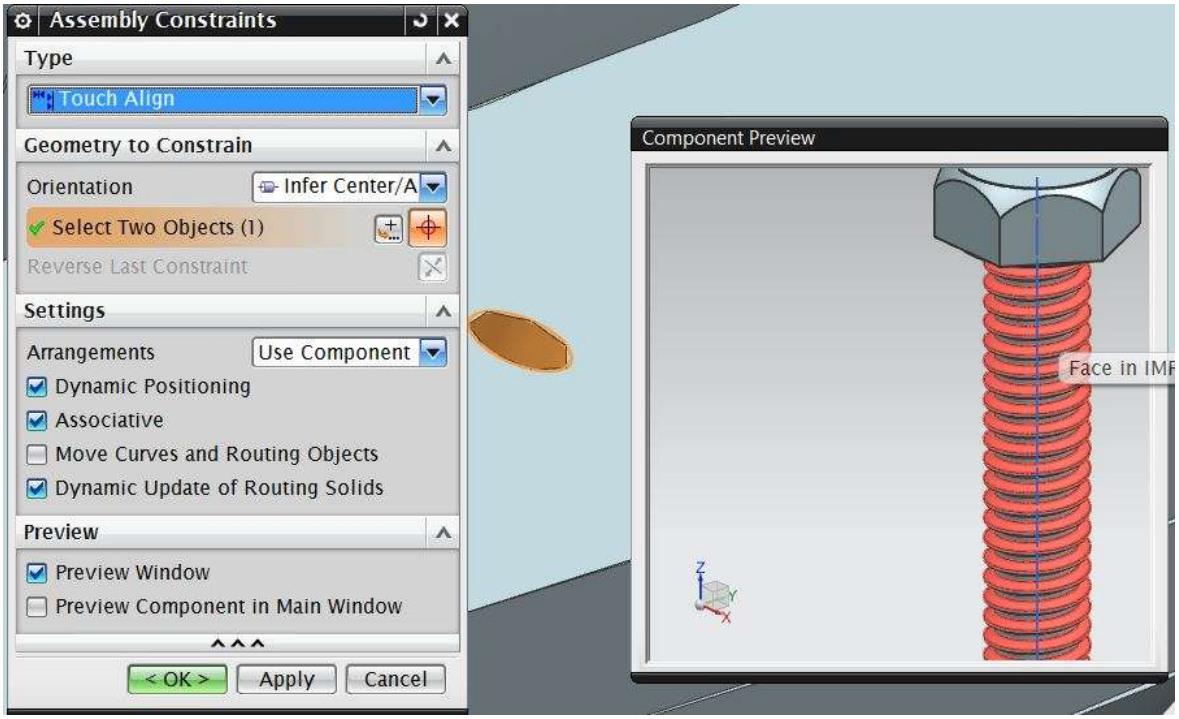


The assembly will now look like the figure below.



Note: Now make 2 holes with dimension of the bolt on the impeller casings. Refer to previous chapters for the same. Diameter of the hole should be 0.25.

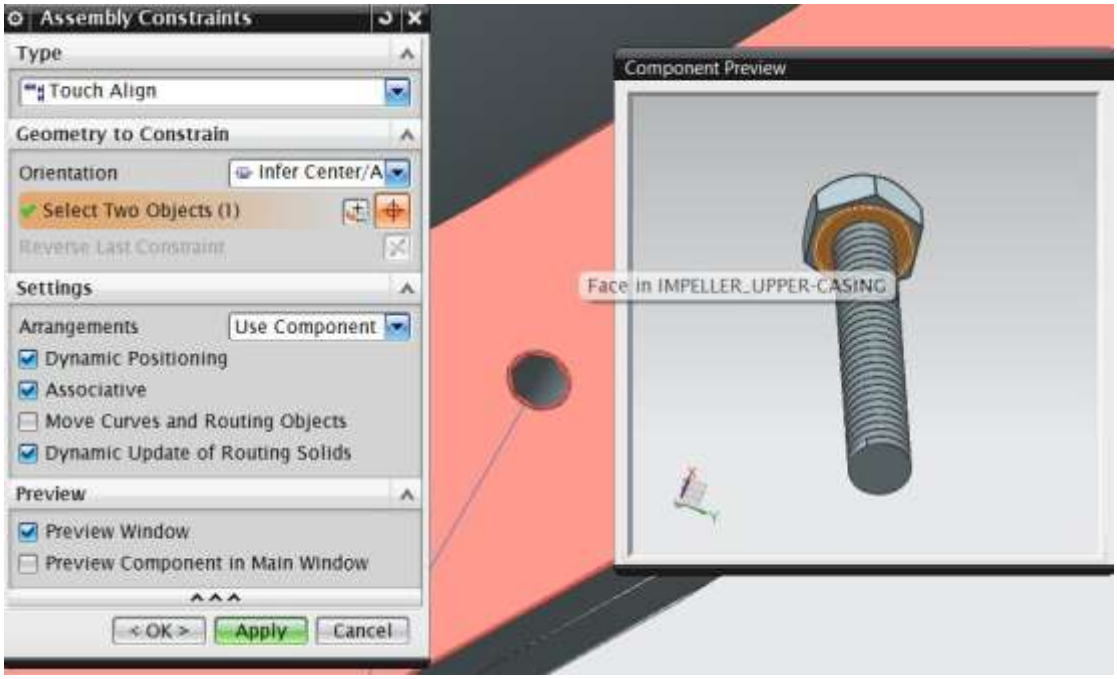
- Click on **ASSEMBLIES → COMPONENTS → ADD COMPONENT**
- Open the file **Impeller_hexa-bolt.prt**
- Choose the **Touch Align** constraint. Use the *Infer Center/Axis* option in the ***Geometry to Constrain*** dialog box
- First, select the outer cylindrical threading on the bolt and then select the inner surface of the hole on the upper casing as show in the figures below.



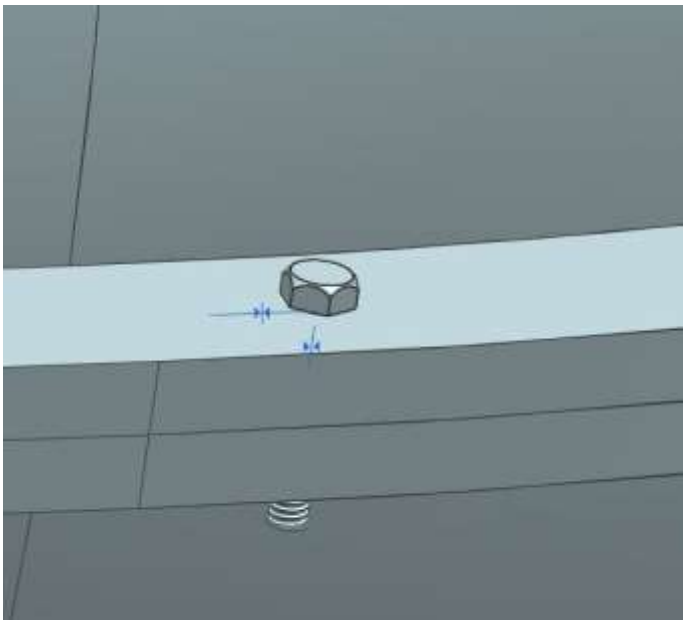
- Again in the **Touch Align** constraint change the ***Geometry to Constrain*** option to *Prefer Touch*

- Select the flat face on the bolt and the face on the rib of the upper casing as shown

- Click **APPLY** and then **OK**



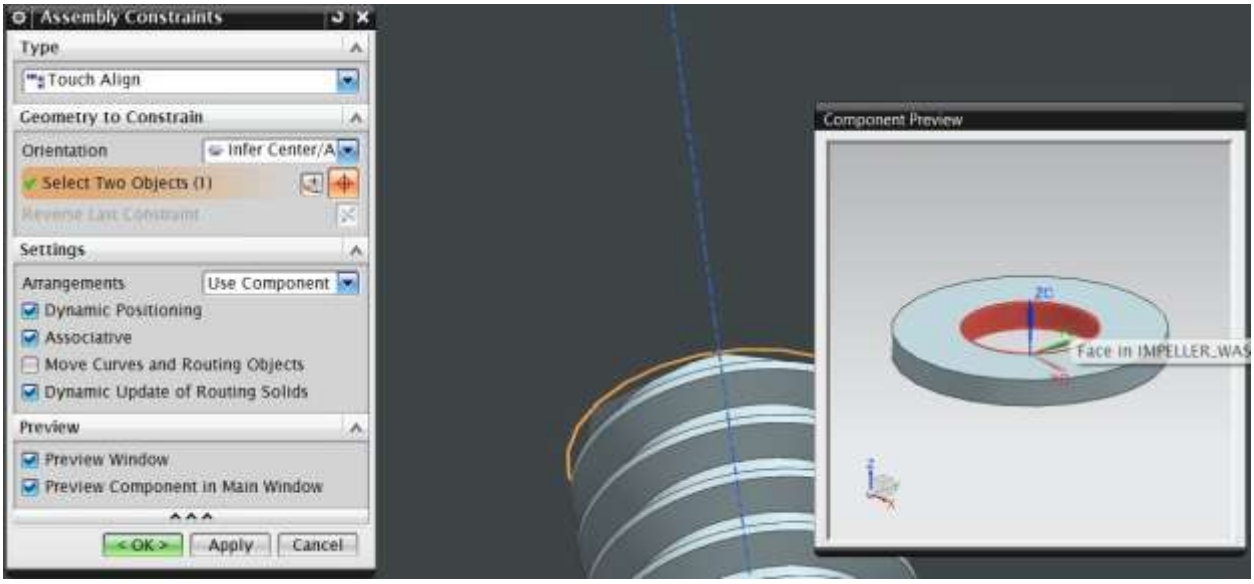
The assembly is shown below.



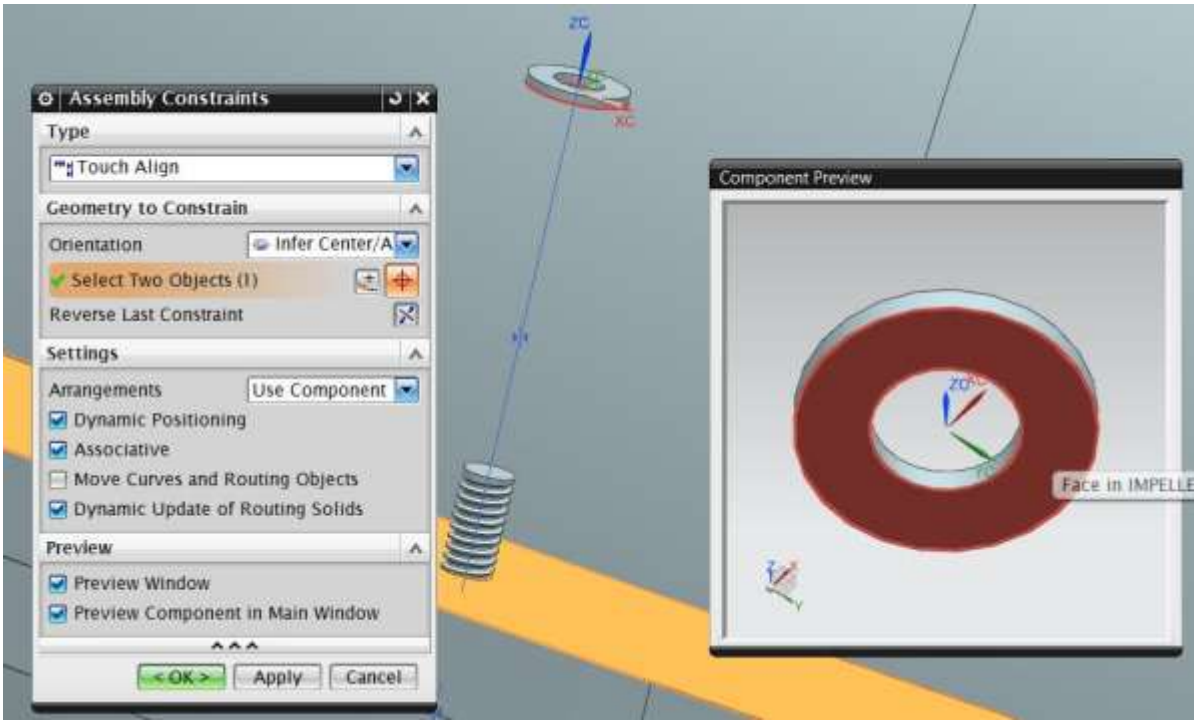
- Repeat the same procedure as before to add the part file **Impeller_washer.prt** [create a washer of inner diameter 0.25 and outer diameter 0.75]

- Choose the **Touch Align** constraint. Use the *Infer Center/Axis* option in the ***Geometry to Constrain*** dialog box

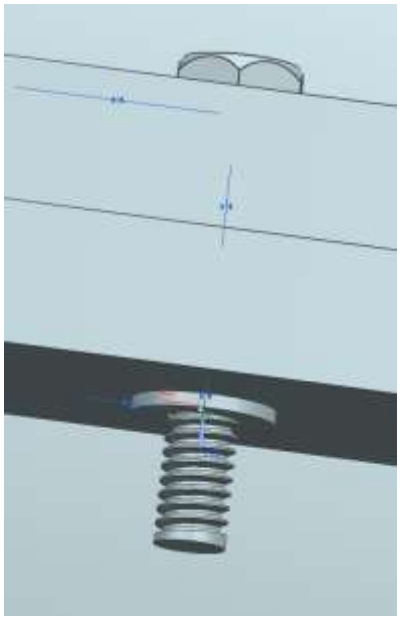
- Select the inner face of the washer and the cylindrical threading on the bolt as shown



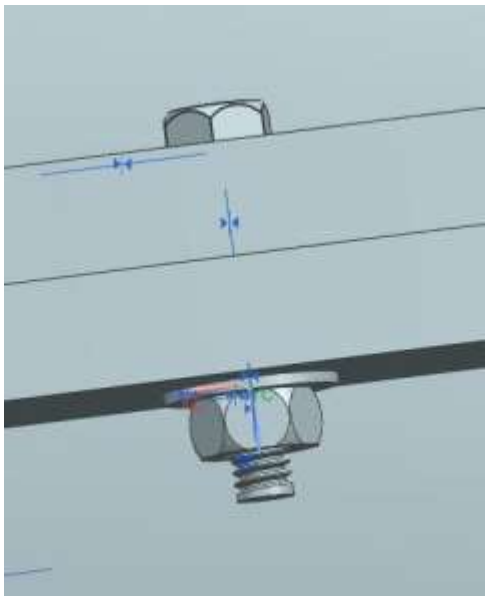
- Again in the **Touch Align** constraint change the ***Geometry to Constrain*** option to *Prefer Touch*
- Select the flat face of the washer and then the face on the rib of the lower casing as shown
- Click **APPLY** and then **OK**



The Assembly is shown below.



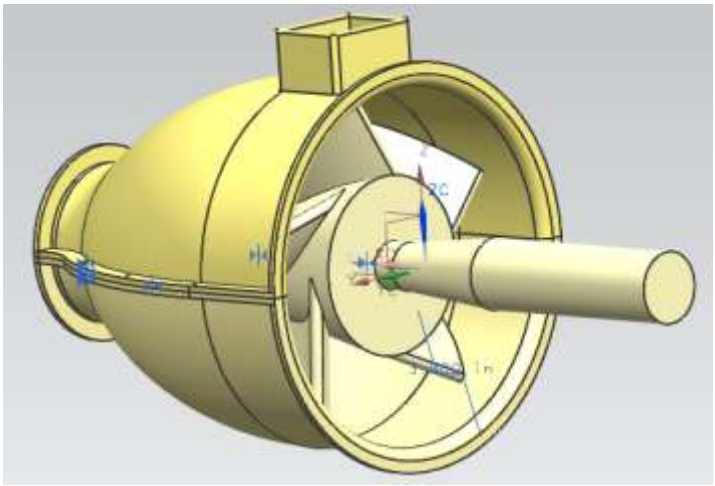
- Add the part file **Impeller_hexa-nut.prt** in the same way as we attached the bolt and the washer.



- Repeat the same procedure to add bolts, washers, and nuts to all the holes in the casing. This completes the assembly of the impeller

There is a simpler way to assemble the bolt, washer, 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.

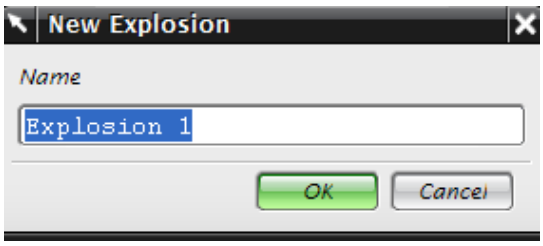
The Final Assembly will look as the shown below. Save the Model.



8.7 EXPLODED VIEW OF IMPELLER ASSEMBLY

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 kind 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→ EXPLODED VIEWS → NEW EXPLOSION**

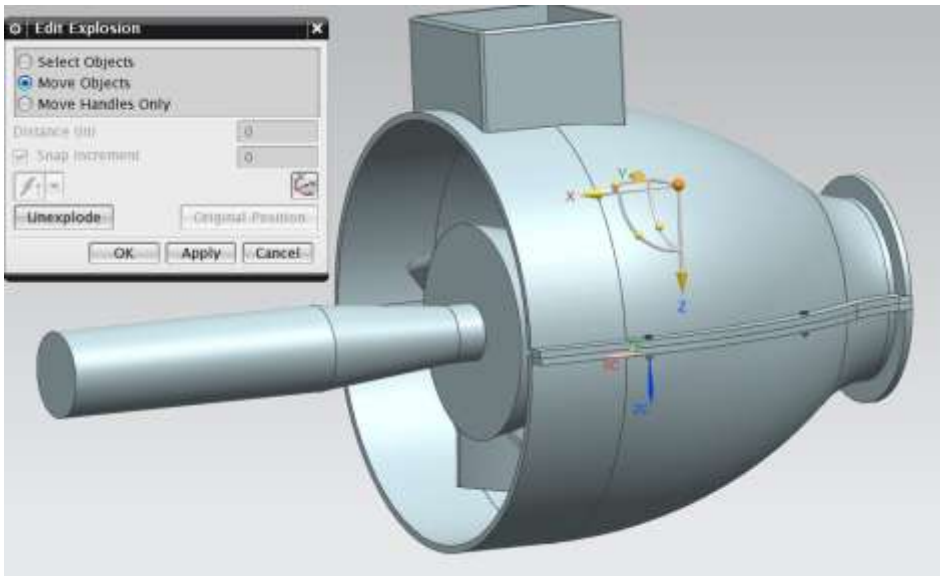


This will pop a Dialog box asking for the name of the **Explosion** view to be created. You can leave name as the default name and choose **OK**

Now the UG environment is in ***Exploded view*** environment though you do not find any difference. When we start exploding some 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's start exploding the components.

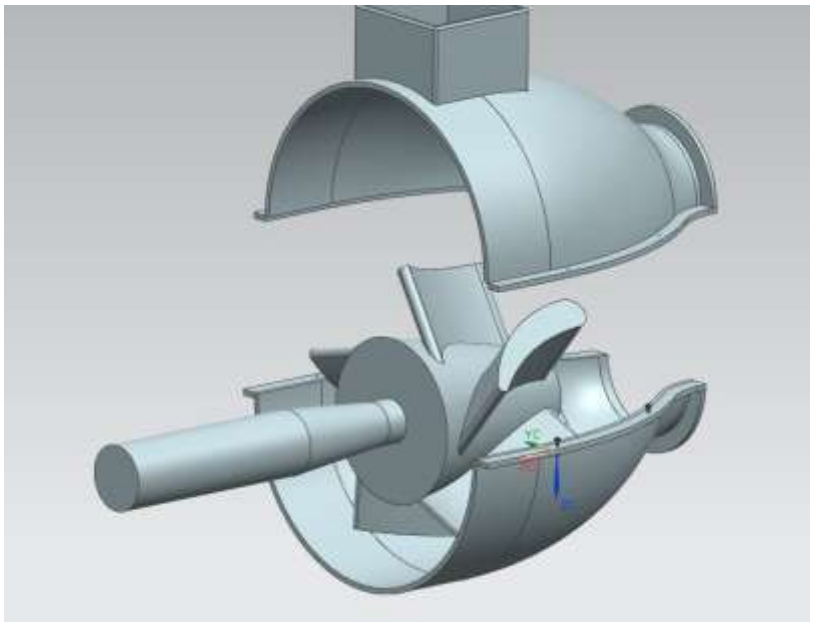
- Right Click on the Upper casing and choose **EDIT EXPLOSION**

The Edit Explosion window will pop up along with a Coordinate system 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]** as shown in the following figure.

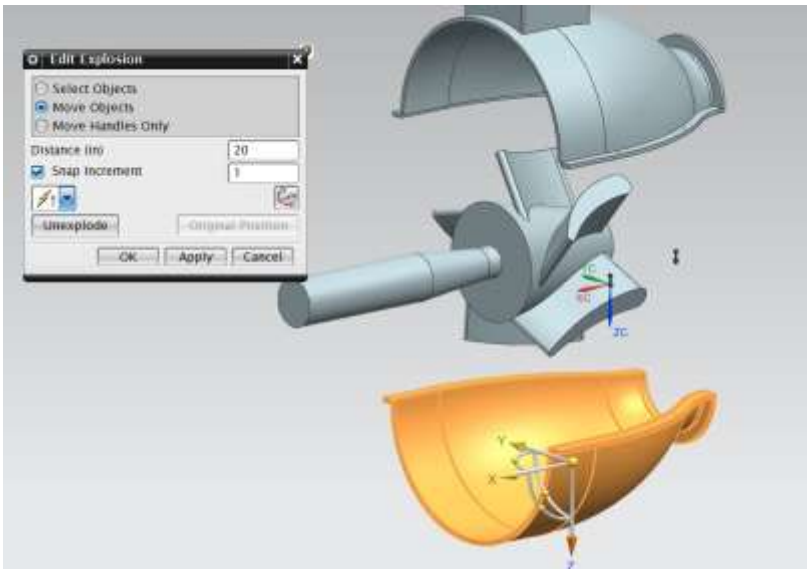
- Choose **OK**



- 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.

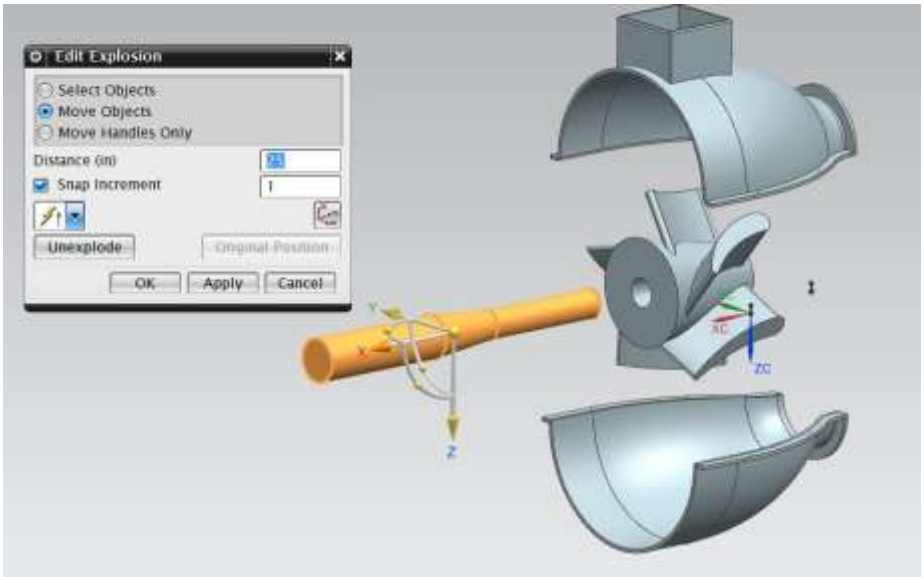
- Click on the **Z-axis**; hold the mouse and drag downwards until the reading in the Distance shows **20** as shown in the following figure. Choose **OK**



- Right click on the shaft and choose **EDIT EXPLOSION**.

- This time click on the **X-axis**; hold the button and drag to the right side until the reading in the distance shows **-25** as shown in the following figure

- Choose **OK**

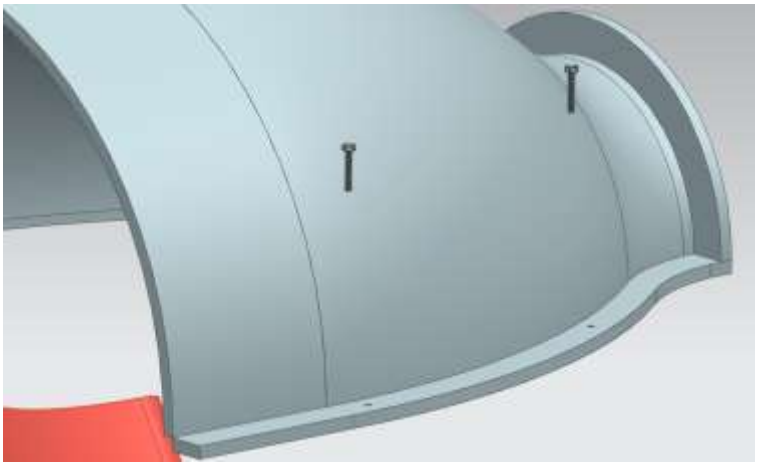


- Select all the four hexagonal bolts in the assembly by clicking on them

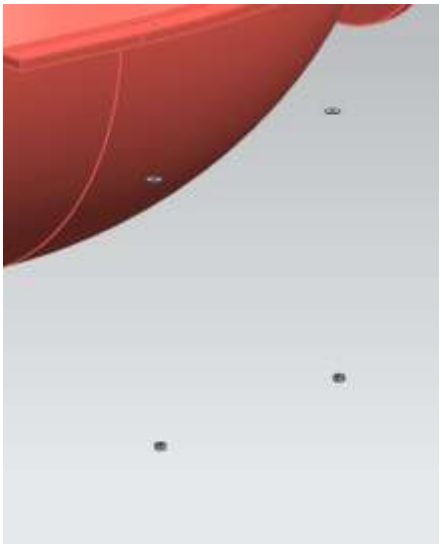
- Right click on one of them and choose **EDIT EXPLOSION**

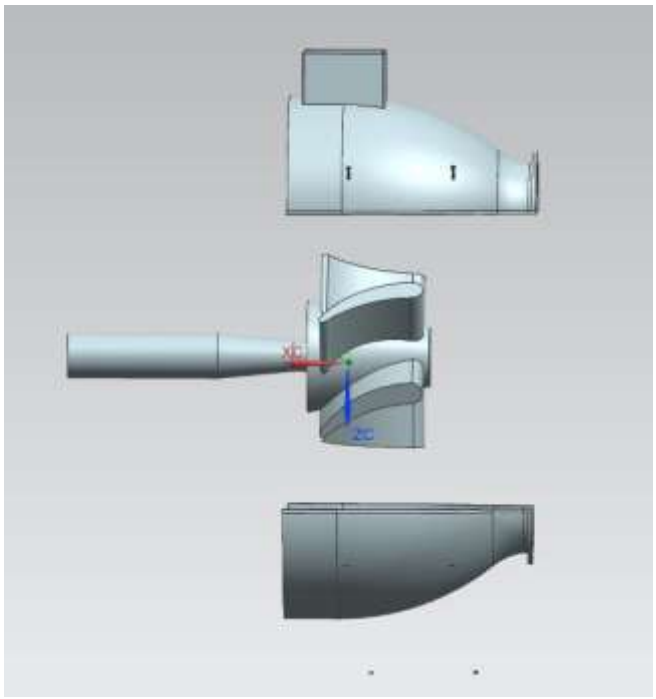
- This time click on the **Z-axis**; hold the button and drag upwards until the reading in the Distance shows **25** as shown in the following figure. This will move all the six bolts together to the same distance.

- Choose **OK**



Likewise, select all the four hexagonal nuts together and move them downwards to a value of **-30** and the six washers to the distance of **-27**. This is the Exploded view of the assembly. The following are the pictures of the Final Exploded view. You can rotate and see how it looks like.





It you want to retain the original assembly view you can **unexplode** any particular component,

- Right click on the component and choose **UNEXPLODE**.

If you want to unexplode all the components,

- Choose **ASSEMBLIES → EXPLODED VIEWS → UNEXPLODE COMPONENT**

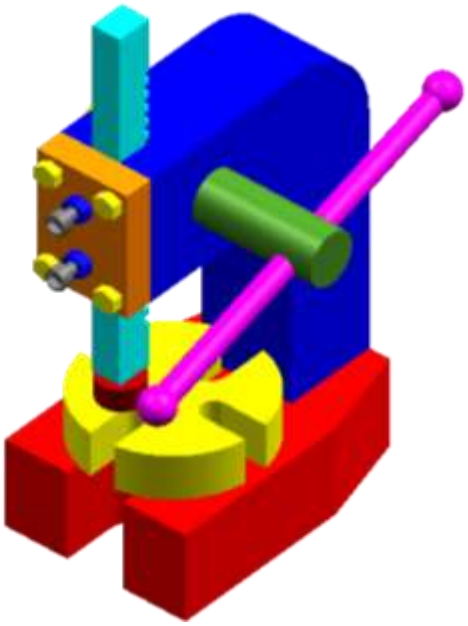
Select all the components and choose **OK.**

8.7 EXERCISE - ARBOR PRESS ASSEMBLY

In this tutorial, we have modeled various parts, some of which are components of the arbor press, which is shown below. Assemble the arbor press using the components that you have modeled in addition to ones that are provided to you that you have not modeled before. The complete list of parts that the arbor press assembly consists of includes:

- Allen Bolt
- Allen Nut
- Base
- Circle base
- End clip
- Handle
- Hexagonal Bolt
- L-bar
- Pin
- Pinion
- Pinion handle
- Plate
- Rack
- Sleeve

All these parts are provided in a folder that can be accessed along with this tutorial in the same internet address.



CHAPTER 9- FINITE ELEMENT ANALYSIS

FEA, or Finite Element Analysis, is a technique for predicting the response of structures and materials to environmental factors such as forces, heat and vibration. The process starts with the creation of a geometric model. The model is then subdivided (meshed) into small pieces (elements) of simple geometric shapes connected at specific node points. In this manner, the stress-strain relationships are more easily approximated. Finally, the material behavior and the boundary conditions are applied to each element. Software such as NX9.0 computerizes the process and makes it possible to solve complex calculations a matter of minutes. It can provide the engineer with deep insights regarding the behavior of objects.

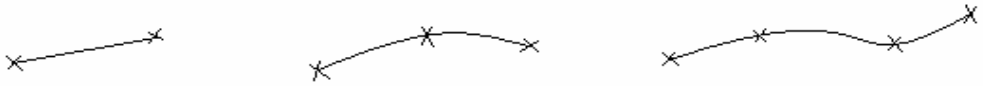
Some of the applications of FEA are Structural Analysis, Thermal Analysis, Fluid Flow Dynamics, and Electromagnetic Compatibility. Of these, FEA is most commonly used in structural and solid mechanics applications for calculating stresses and displacements. These are often critical to the performance 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 solid geometries.

9.1 INTRODUCTION

9.1.1 Element shapes and nodes

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

One-dimensional elements:



2-node (Linear) | 3-node (Quadratic) | 4-node (Cubic)

Two-dimensional elements:

Triangular:

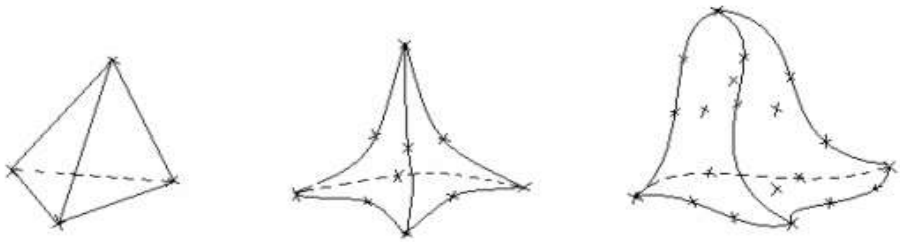
3-node (Linear) | 6-node (Quadratic) | 10-node (Cubic)

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

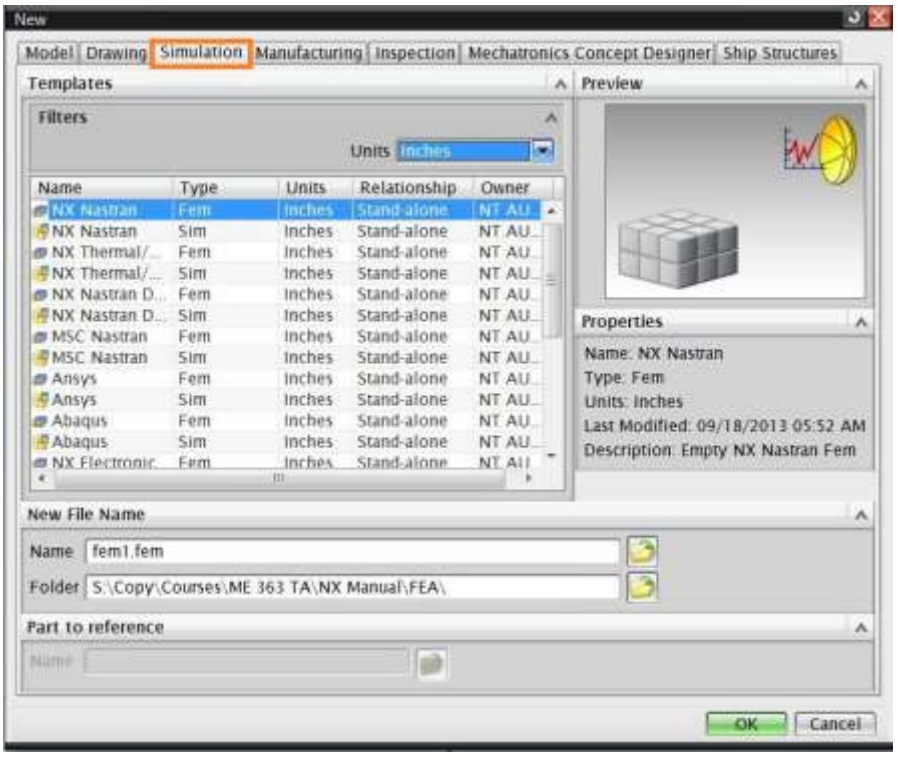
The results of FEA should converge to the exact solution as the size of finite element becomes smaller and smaller.

9.1.2 Structure Module

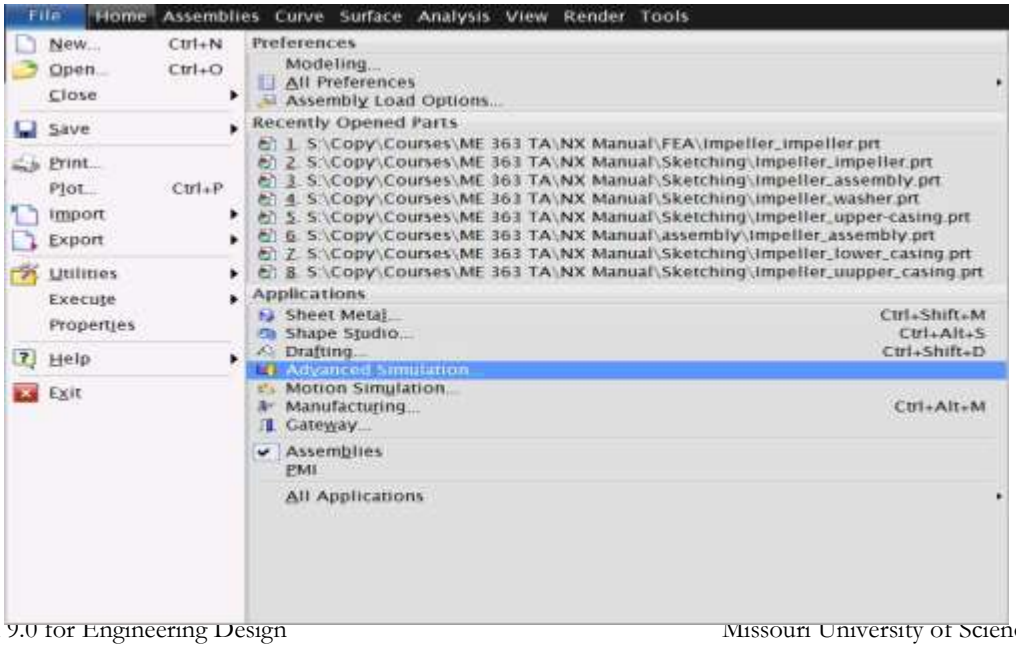
- Copy and paste the file **Impeller_impeller.prt** into a new folder to avoid changes being made to the assembly

- Open this newly copied file

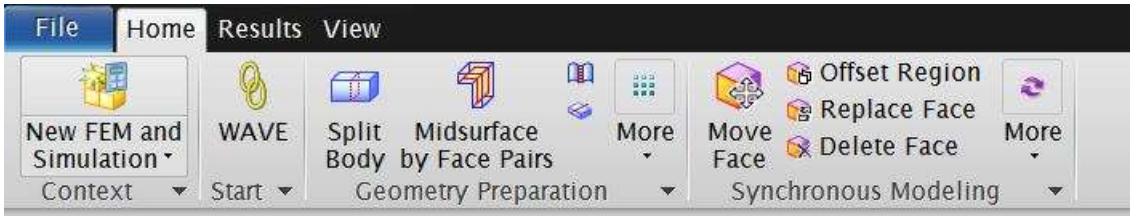
- Click on **NEW** ⟶ **ADVANCED SIMULATIONS** if the part is NOT already opened in the NX window as shown below



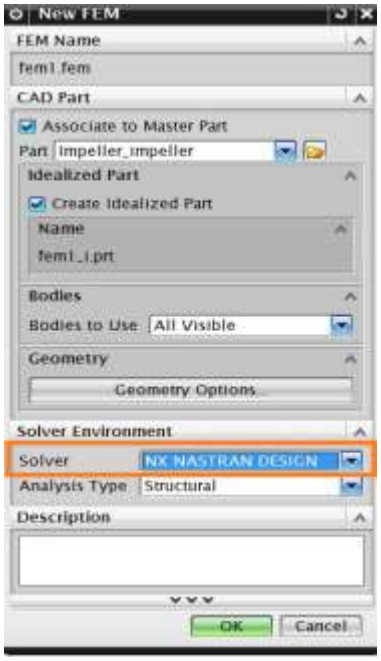
- If part is already opened in NX, then click on **NEW** ⟶ **ADVANCED SIMULATIONS**



The following figure is the toolbar for Finite Element Modeling and Analysis of Structures.



Solution: Clicking on this icon will open up the **CREATE SOLUTION** window where you can select the solver algorithm from one of these: NX NASTRAN, MSC NASTRAN, ANSYS or ABAQUS. In addition, you can choose the type of analysis to be performed. In this tutorial, only Structural Analysis will be covered with NX NASTRAN.



Material Properties: This allows you to change the physical properties of the material that will be used for the model. For example, if we use steel to manufacture the impeller, we can enter the constants 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 available in NX9.0.

Loads: This option allows you to exert different types of forces and pressures to act on the solid along with the directions and magnitudes.

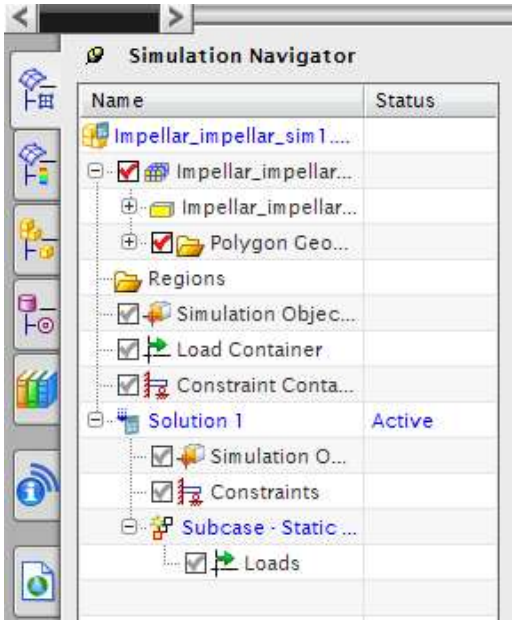
Boundary Conditions: Boundary conditions are surfaces that are fixed to arrest the degrees of freedom. Some surfaces can be rotationally fixed and some can be constrained from translational movement.

3D Tetrahedral Meshes: This icon is one of the mesh options that can be used to discretize the model as discussed in beginning of the chapter. 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.

Solve: This is the command to solve all the governing equations by the algorithm that you choose and all the above options. This solves and gives the result of the analysis of the scenario.

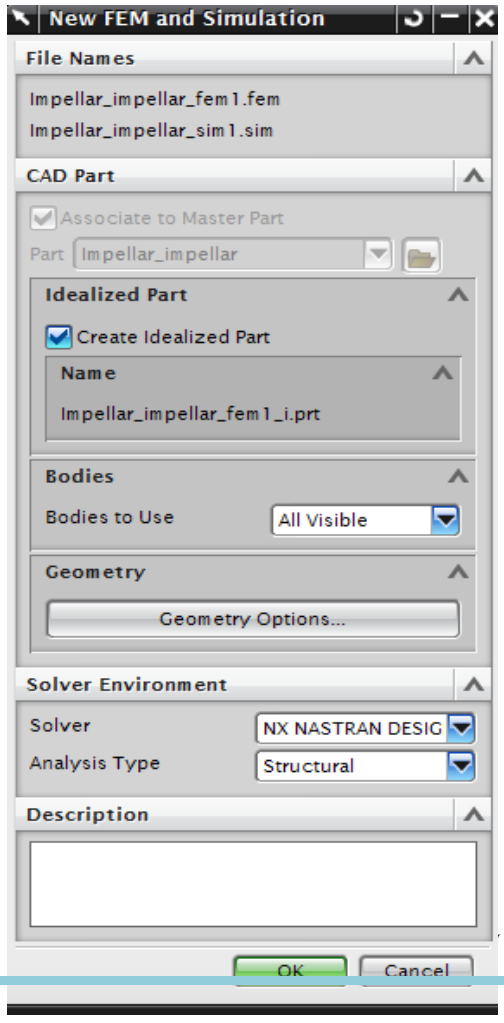
9.1.3 Simulation Navigator



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 the list of the scenarios created for the master model file. In each scenario, it displays the list of loads, boundary conditions, types of meshes, results, reports generated and so on.

9.2 SOLUTION CREATION



The **DESIGN SIMULATION** module is in a way different from when the first scenario is created. NX9.0 creates a folder of the same name as that of the file and at the same location where the file is located. For every scenario or Soultion, it creates five different files with the name of the scenario. They are xxx.SIM, xxx.DAT, xxx.txt, xxx.out and xxx.VDM. All the results generated for the scenarios are saved as **.vdm** files. You can think of a scenario model as a variation of a master design model. Scenarios contain all the geometric features of the master model. They also support body promotions and interpret expressions.

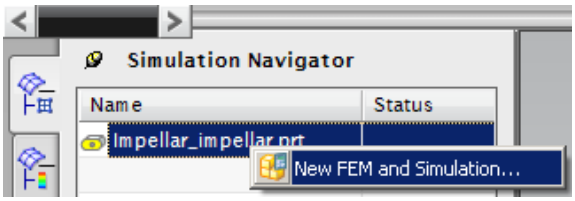
Body promotions are used to provide an independently modifiable copy of the master model geometry and serve as a place to hold scenario-specific features such as mid-surfaces.

The scenario model's geometry is linked to the master model geometry, but a scenario may have additional unique information. For example, the master model may contain all the information about the model's geometry, but the scenario model will contain additional motion data, such as information about links and joints.

Now we will create a scenario. *Note:* When you first open any file in Design Simulation module, it will automatically pop up with Solution creation window to create a solution.

- Click on the Simulation Navigator icon on the navigator toolbar

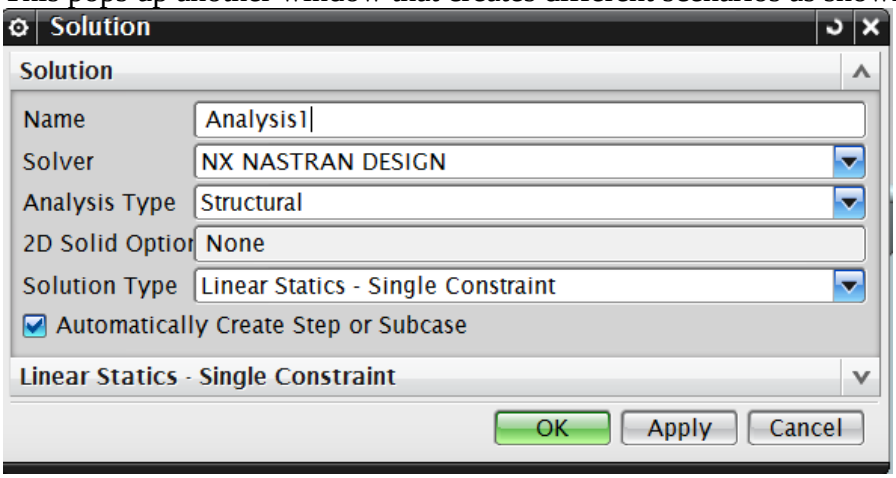
- Right-click on **Impeller_impeller** and choose **New FEM And Simulation**



This will pop up the **New FEM and Simulation** dialog box to create a new scenario.

- Click **OK**

This pops up another window that creates different scenarios as shown below



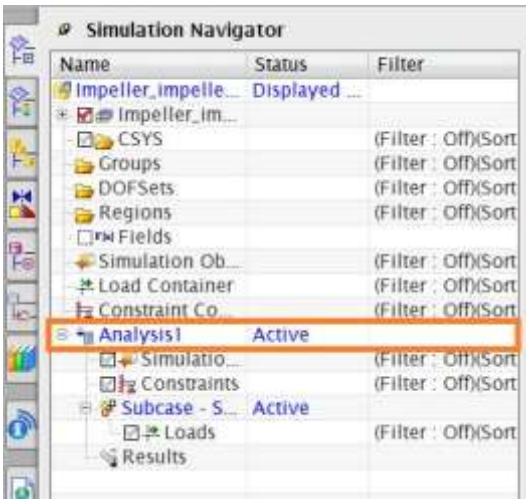
In the **Create Solution** window, you can select the **Solver** and the **Solution Type**.

- Enter the ***Name*** of the first scenario as **Analysis_1**

The default Solver type is **NX NSATRAN DESIGN** and Analysis type as **STRUCTURAL**.

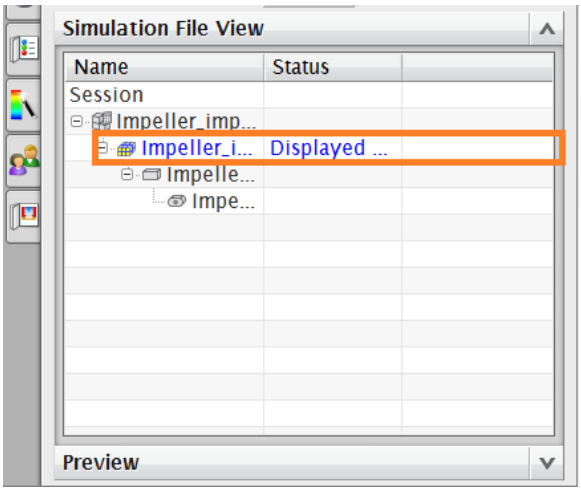
- Choose **OK** to create a new **Solution** called Analysis_1, which is displayed in the **Simulation Navigator**

The Simulation Navigator will now look like the following figure.

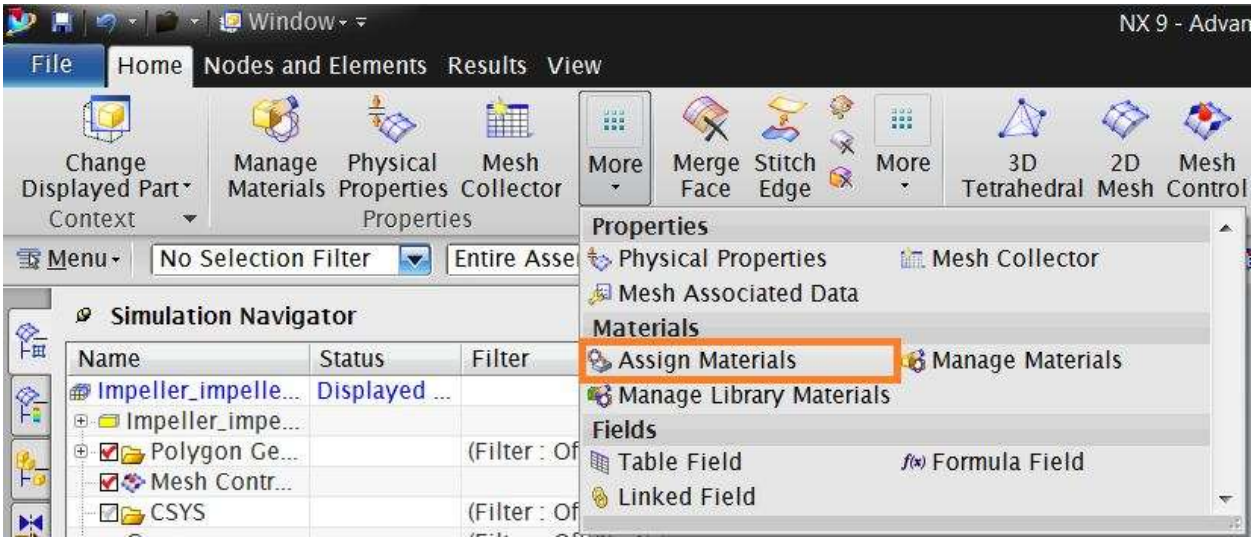


9.2.1 Material Properties

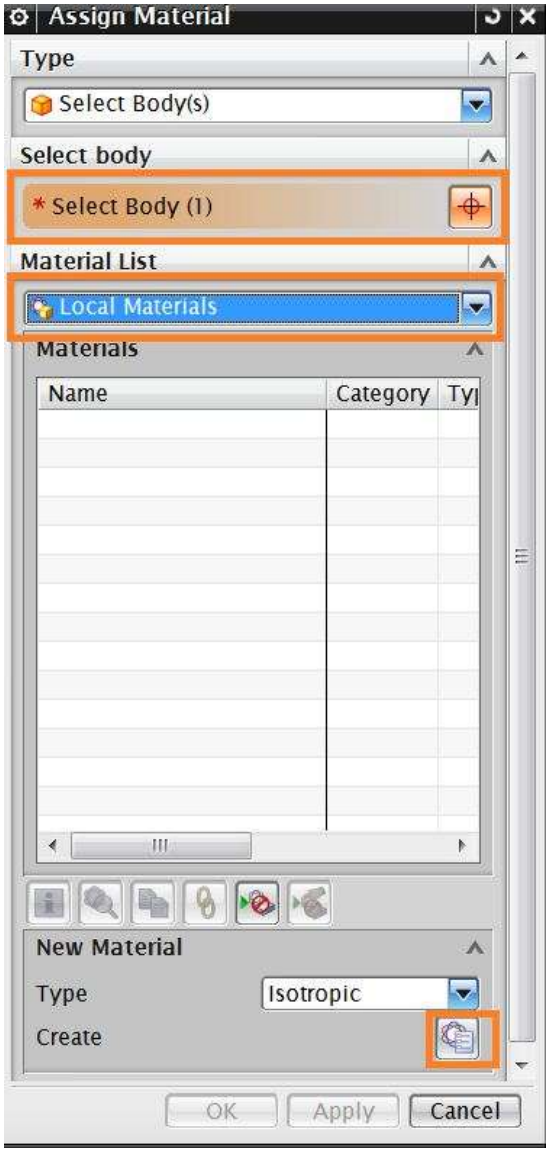
The next step is to give the material properties to the solid model for this scenario. Because we don't have any data in the library to retrieve for standard material, we will create one. Let us assume that we will use steel to manufacture the impeller.
For assigning the materials click on the second impeller file in the simulation view window as shown below



- Click on the **Material Properties** icon on the Toolbar



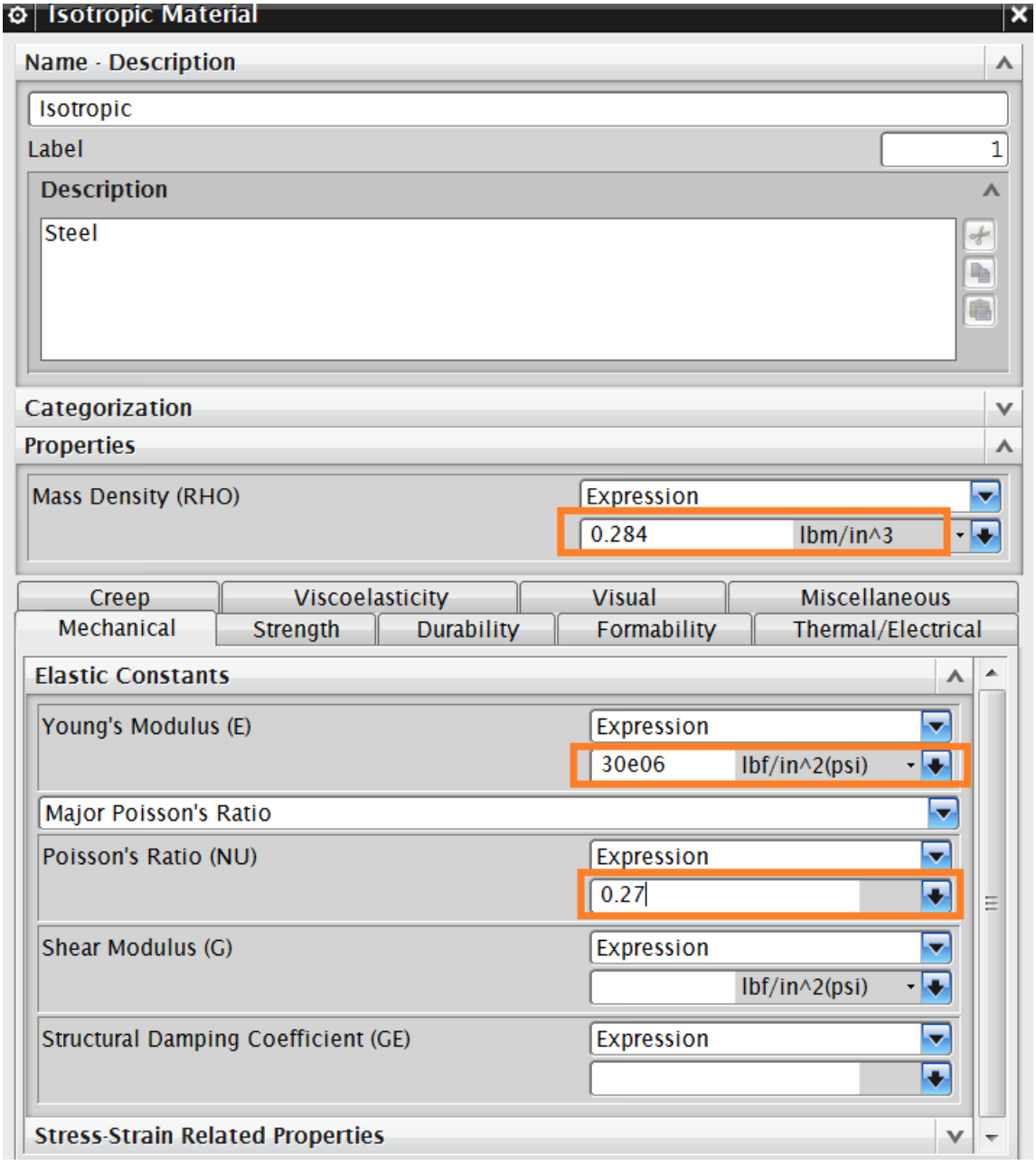
The **Materials** window will pop up. You have the option of choosing the pre-defined materials from the *library* or create another material.



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

(Note that 30e6 represents $30X10^6$)

- Choose **OK** to exit the ***Isotropic Material*** window
- Click on the **Impeller** model
- Click **Apply** and then **OK**

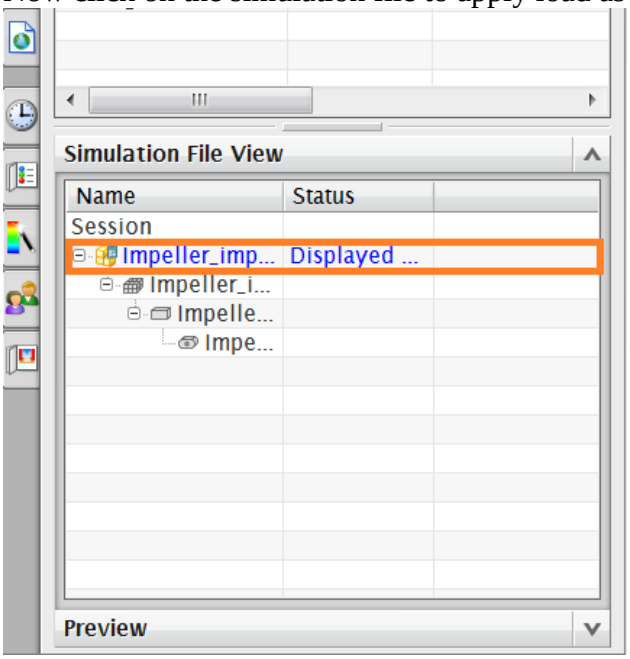


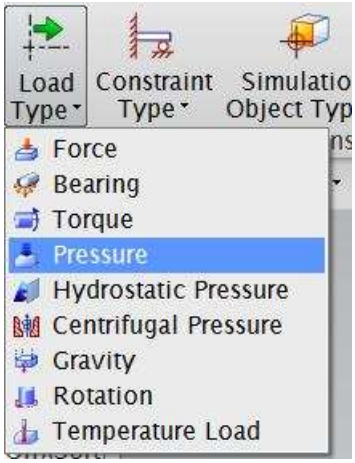
This will assign the material properties to the impeller. Now let us attach the load.

9.2.2 Loads

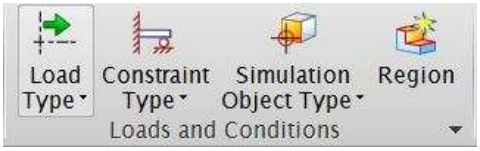
The loads applied on the solid model should be input to the system. For the impeller, the major force acts on the concave surfaces of the turbine blades. This loading can be approximated by normal pressure on all the five surfaces. Since we are not too concerned about the magnitude of the load, let us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

Now click on the simulation file to apply load as shown below

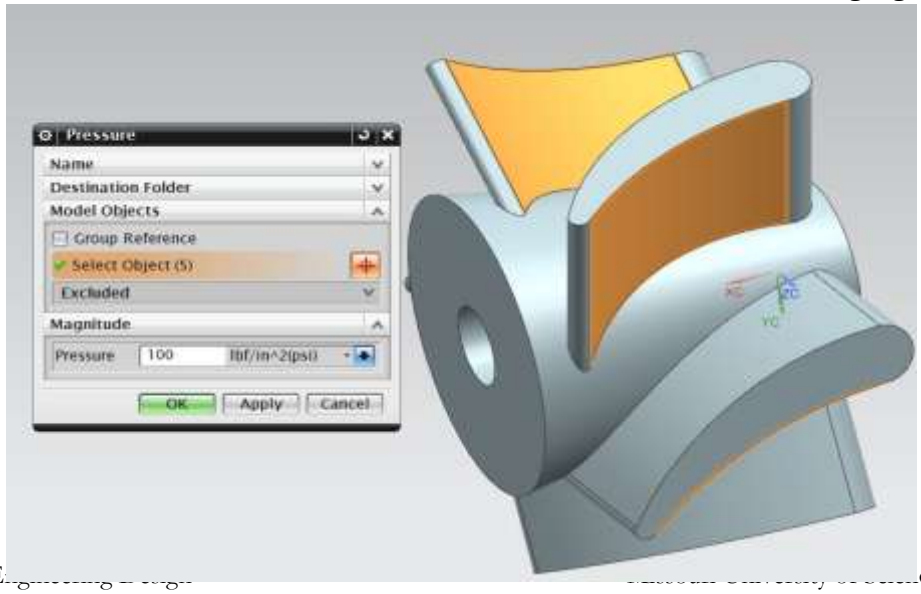




- Click on the ***Loads*** icon in the HOME bar



  - In the ***Load Type*** select pressure
- Click on the five concave surfaces of the blades as shown in the following figures

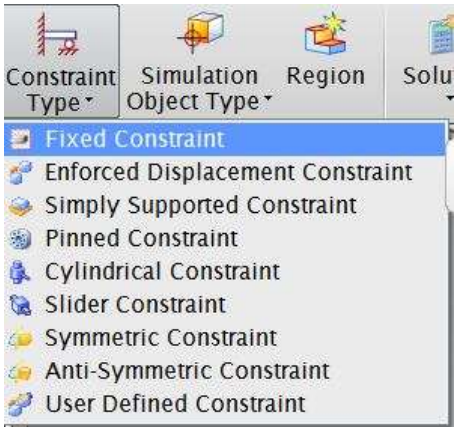


- Enter the value for Pressure as 100 and keep the units as lb-f/in^2 (psi)

- Choose **OK**

9.2.3 Boundary Conditions

Let us give the boundary conditions for this scenario. 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.



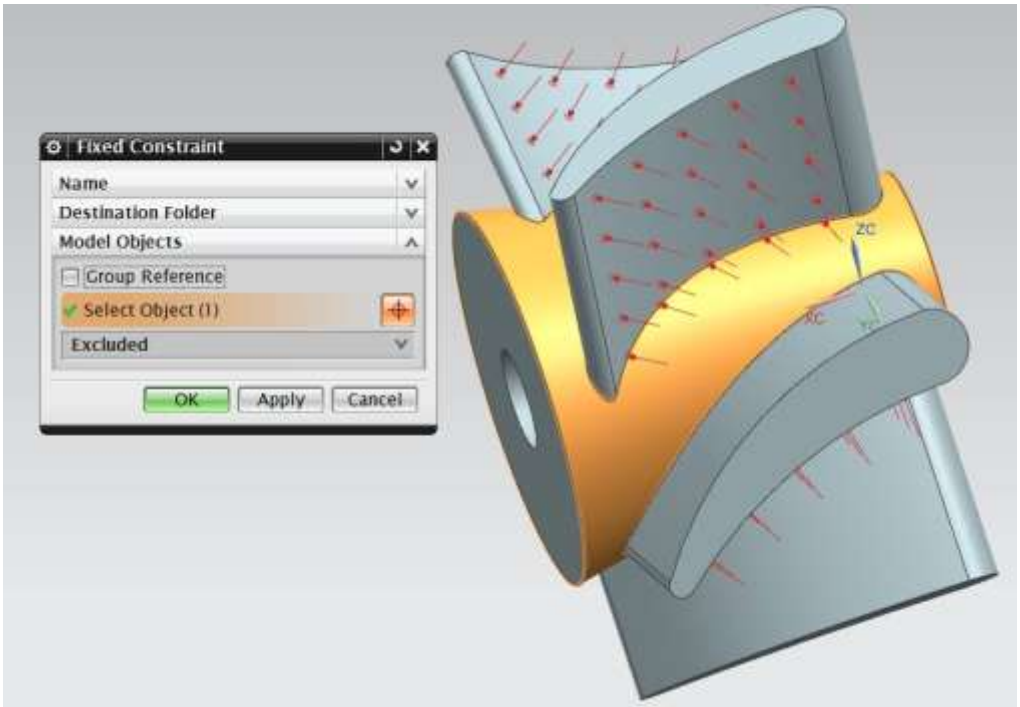
- Click on the **Constraint Type** icon

- 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 drop-down menu on the toolbar.

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

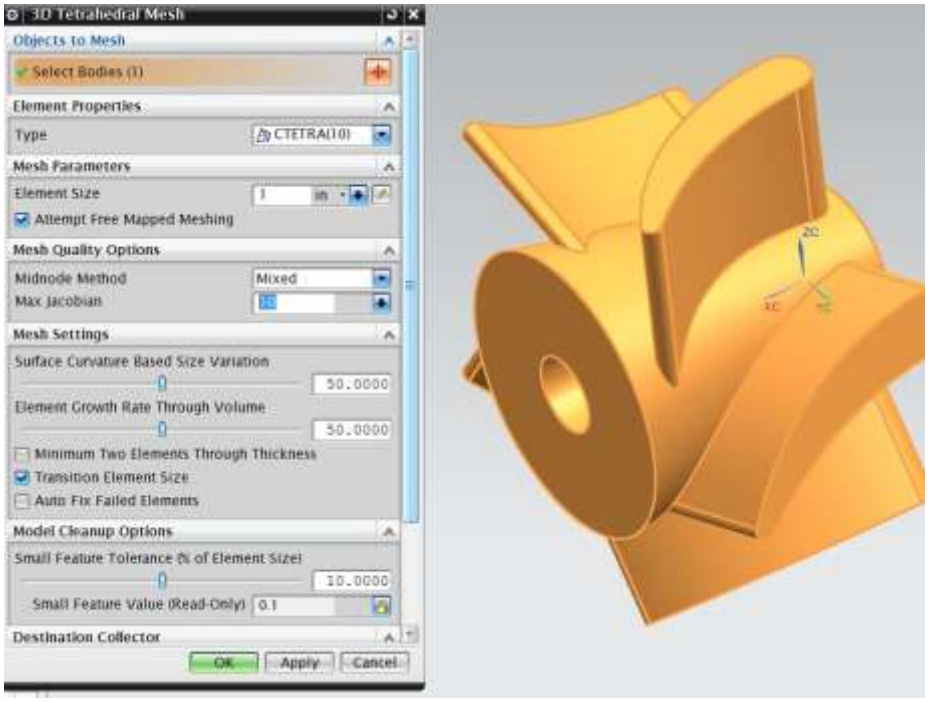
- Choose **OK**



9.2.4 Mesh

The ***'Mesh'*** option discritizes the model into small elements. It can be defined as the first step or the last in the FEA process depending upon the material properties. In our case we define it last.

- Select .fem file from the **simulation file view**
- Click on the **3D Tetrahedral Mesh** icon



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

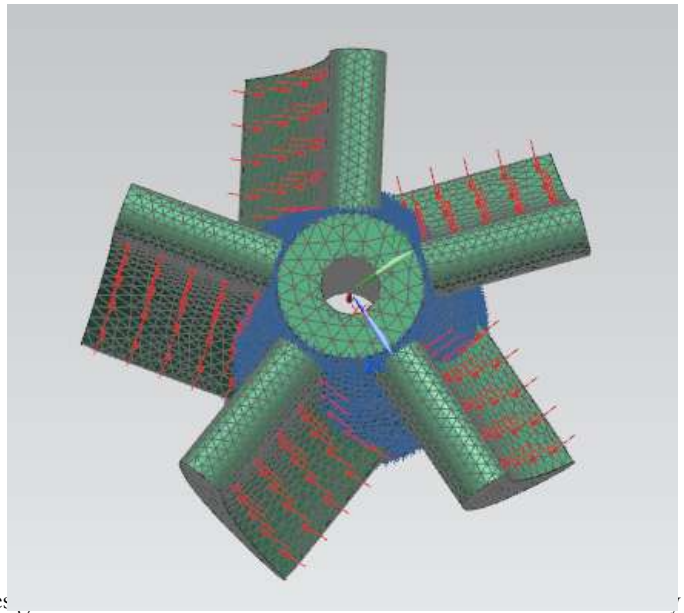
- Click on the solid object model on the graphic screen.

There are two types of tetrahedral elements available in NX9.0. One is 4-nodes and another is 10-node.

- Choose the **Type** to be **TETRA10**

- Enter the **Overall Element Size** as **1.0**
- Choose **OK**

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

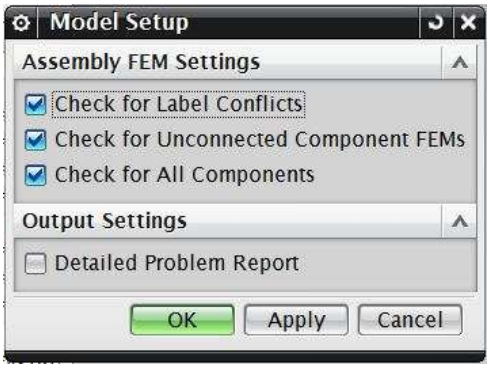


Note: While meshing the solid there is a trade-off you need to consider. If you choose a smaller element with higher nodes you will get better accuracy in your analysis than larger element. However, the time required to solve the model with smaller elements will much greater than with larger element.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

9.3 RESULT AND SIMULATION

9.3.1 Solving the Scenario

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

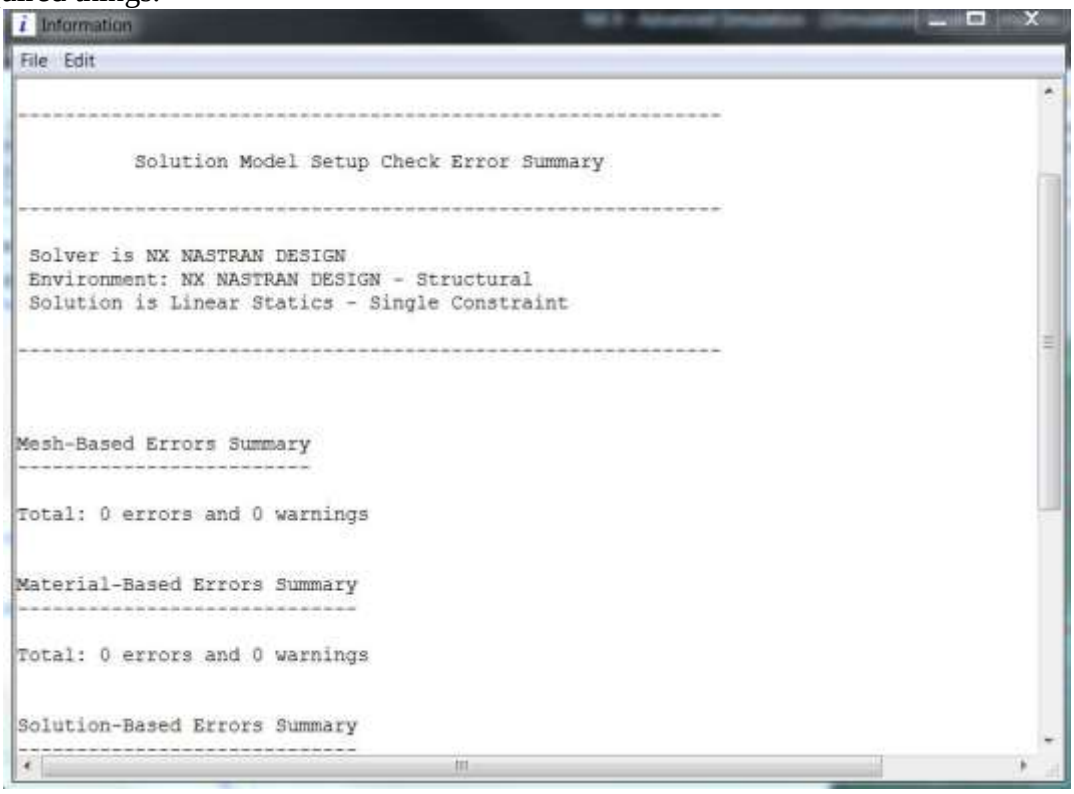


- Click on the MENU ⟶ ANALYSIS ⟶ FINITE ELEMENT MODE CHECK⟶ MODEL SETUP

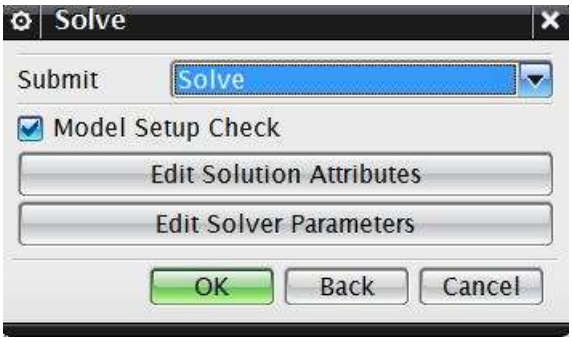
This will pop-up a menu as shown on the right.

- Choose **OK**

This will display the result of the **Check.** You will be able to see any errors and warnings in a separate window. The errors or warnings in the FEA model creation are; no material, no loads and so on. In case you get these 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.

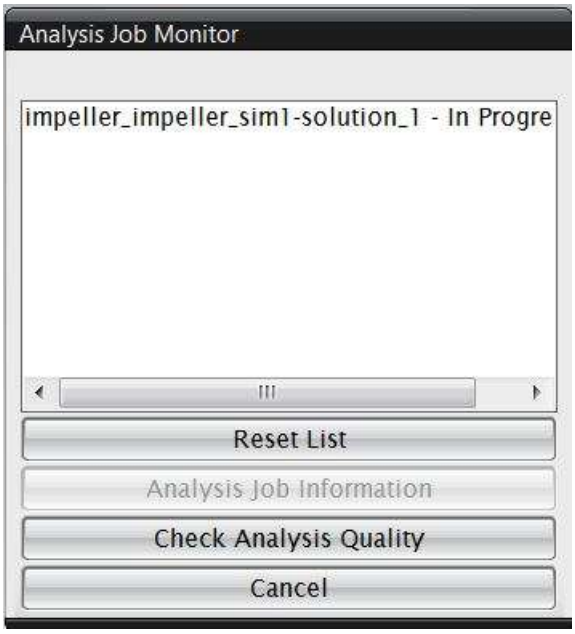


- Click on the ***Solve*** icon

This will open the **Solve** window.

- Click **OK** without making any changes

It may take a while to generate the results. 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 **Running**



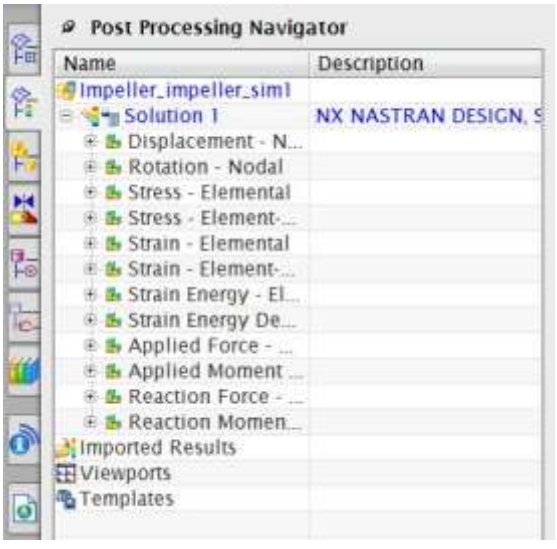
- Click on **CANCEL** when the **Analysis Job Monitor** window says **Completed**

9.3.2 FEA Result

- Open the **Simulation Navigator**

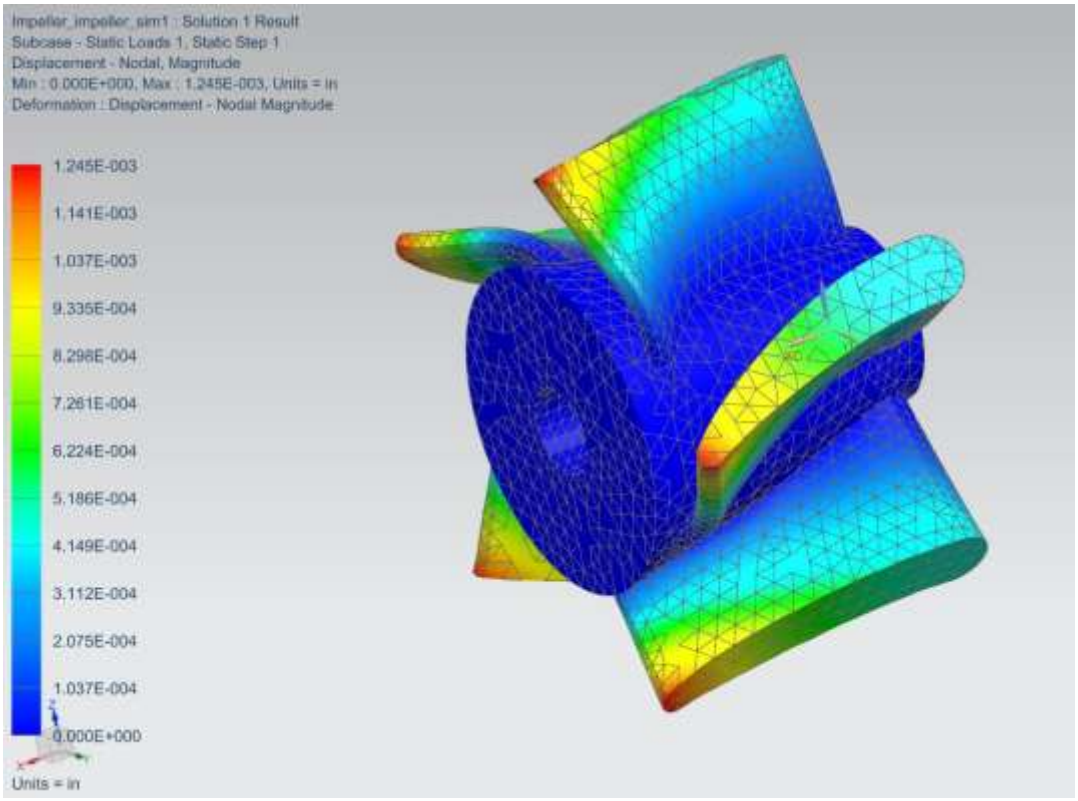
- Click on the **Post Processing Navigator**

This will take you into the **Post-Processing Navigator**. The Post-Processing Navigator shows all the Solution you created. 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 menu

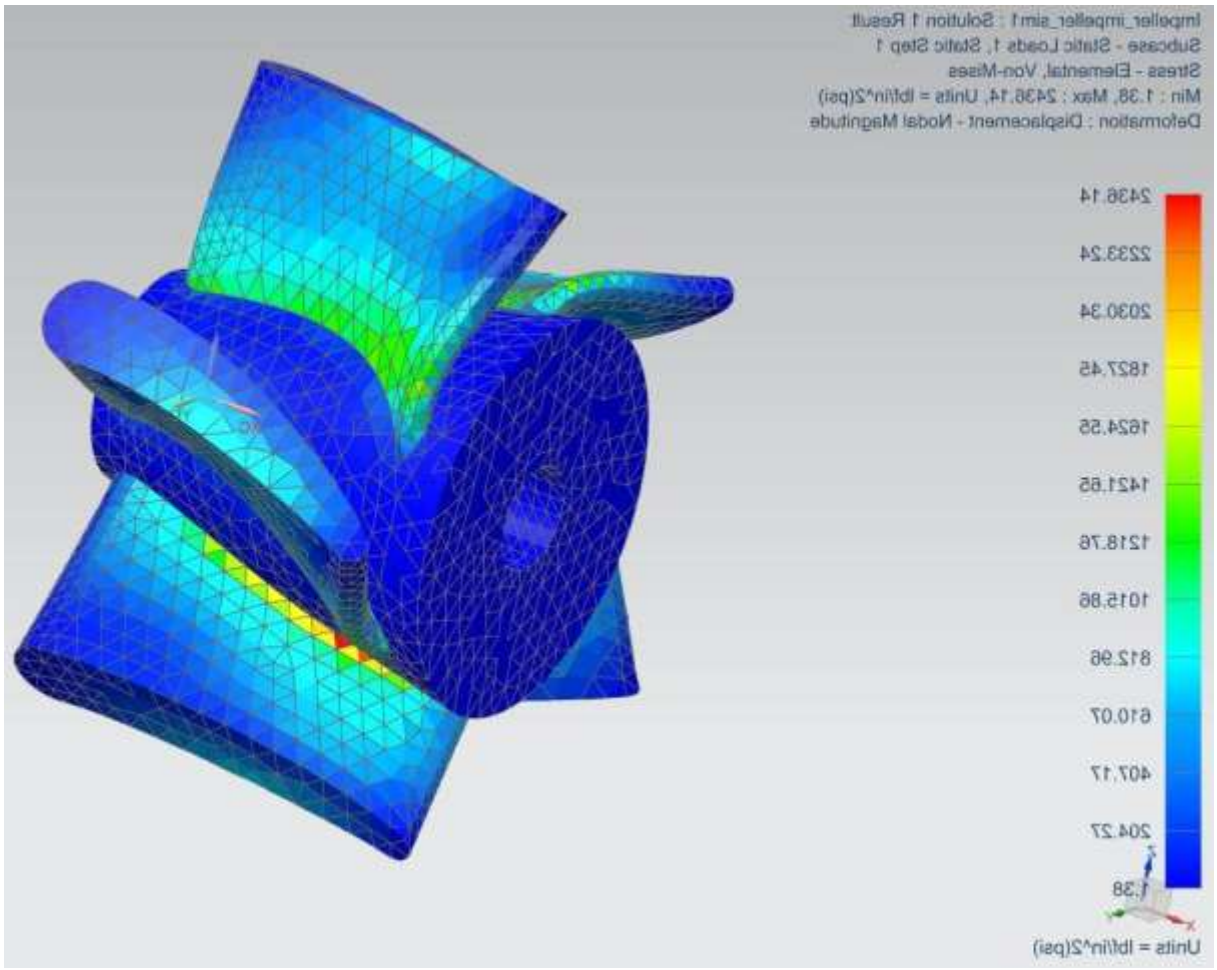
The screen will now appear as shown below.

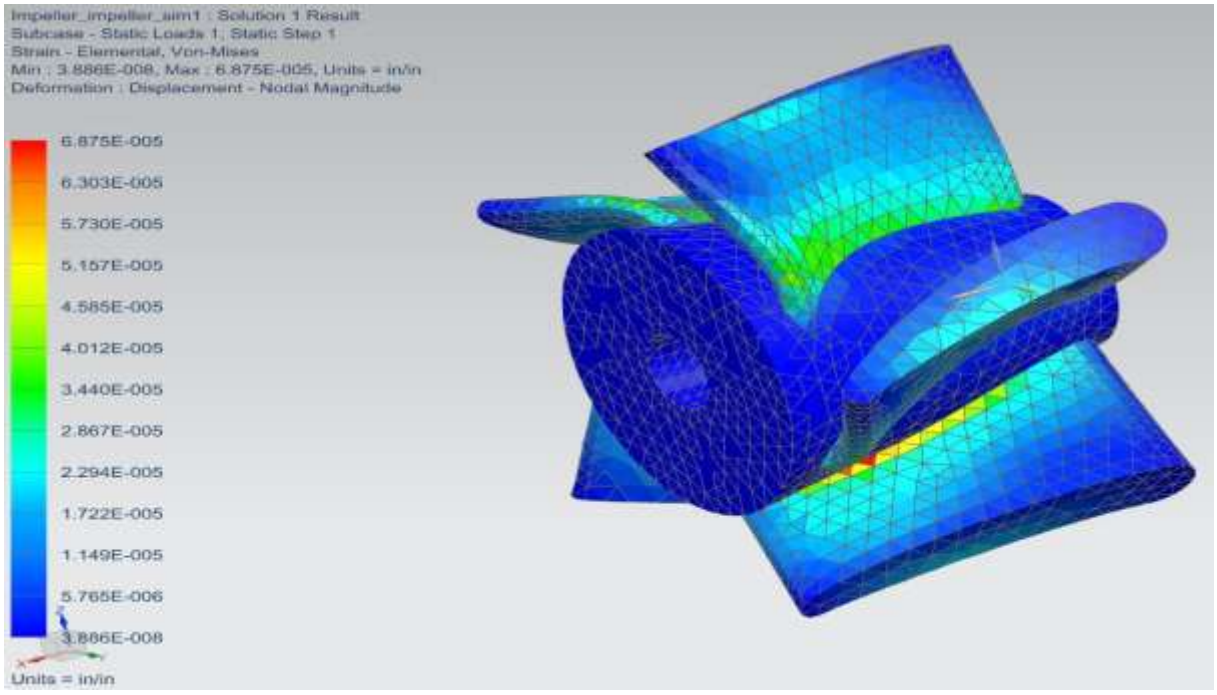


You can easily interpret the results from the color-coding. The orange-red color shows the maximum deformation zones and the blue area shows the minimum deformation zones. You can observe that because the conical core is fixed, it experiences zero deformation. The analysis also shows that the maximum deformation experienced at the tip of the blades is 1.245×10^{-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.

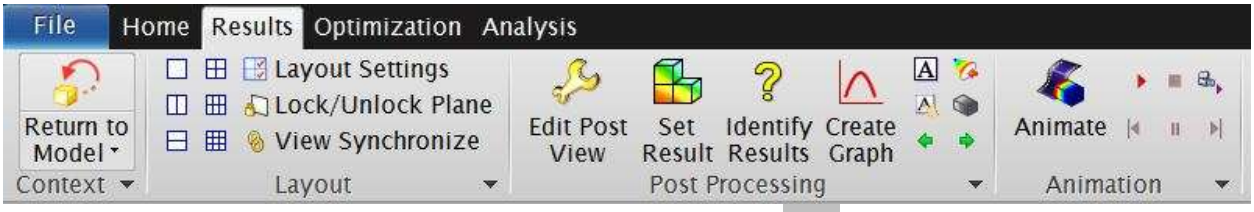




9.3.3 Simulation and Animation

The ***Post Processing Toolbar*** should appear when you select the **Design Simulation Module.** However, in case it does not become visible follow these steps.

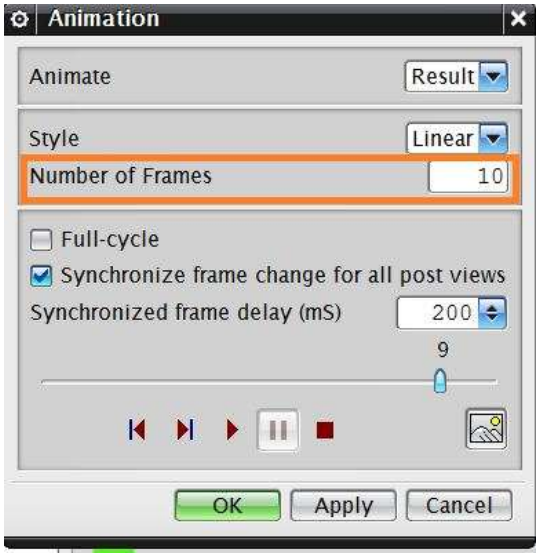
- Click on the **RESULTS** tab. A section for Animation can be seen on it as follows



- Click on the **Animation** icon on the **RESULTS Toolbar**

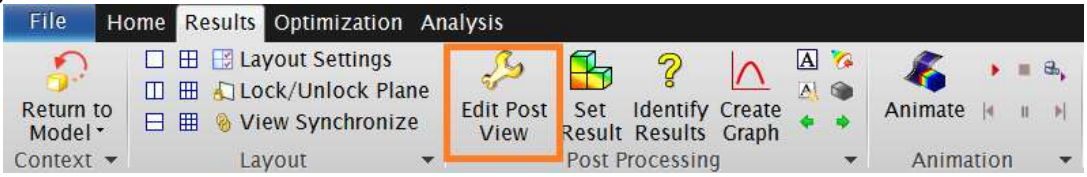
- In the **Animation Setup** window, change the number of frames to **10** and click on the **Play** button to see the animation of the deformation

You can also find the play button on the **Post Processing** Toolbar itself.



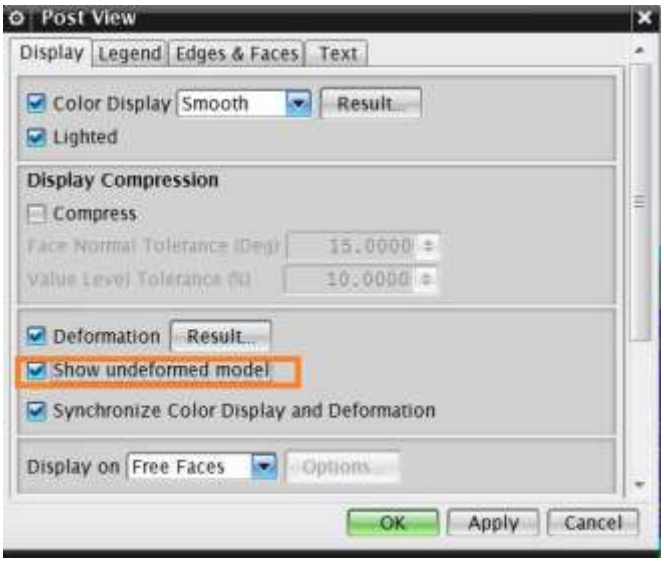
You can now see an animation of how the impeller is deformed as the loads are applied to the blades.

- To make any setting changes in the results display, click on the **Post View** icon as shown in figure below

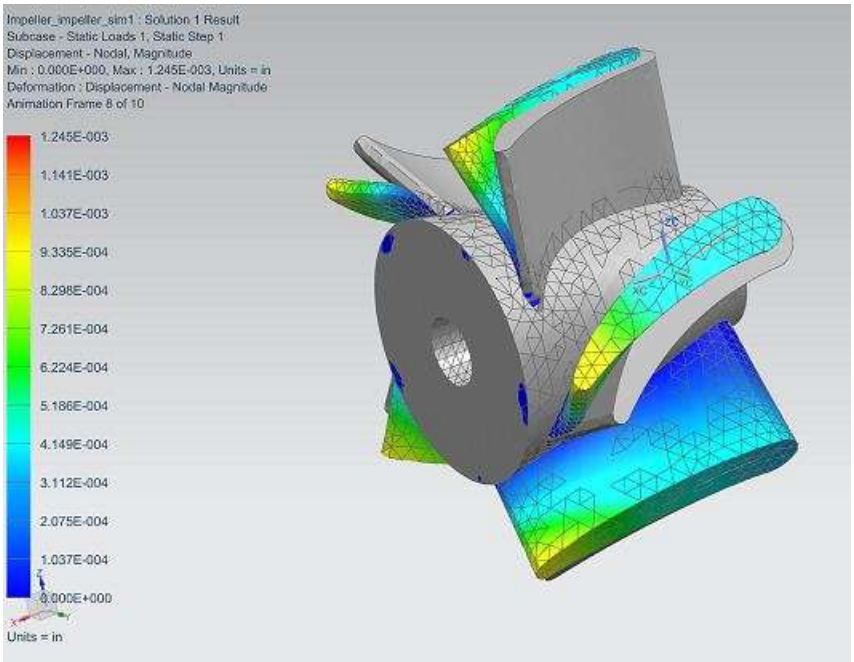


- In the popped up **Post View** Dialog box click on **Deformed Results** under **DISPLAY** tab button

- In the **Deformed Results** dialog box check the **Show Un-deformed Model** as shown in the second figure below and choose **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.



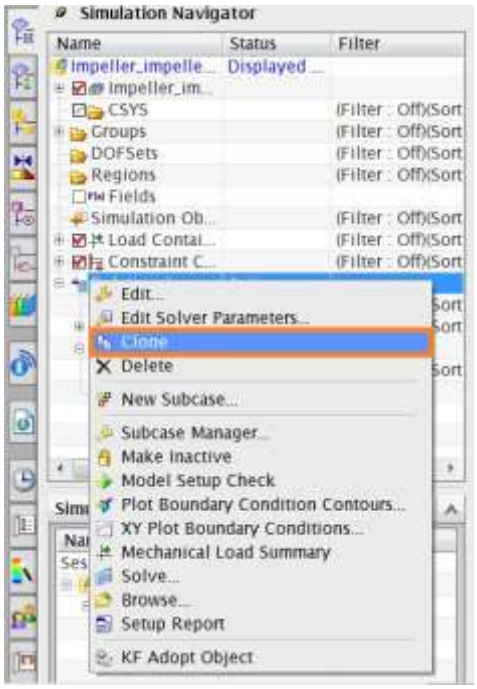
- Click on the **Stop** button

- Right-click on the **Analysis_1** in the **Post-Processing** bar and click on **Unload**. This should take your screen back to the meshed model

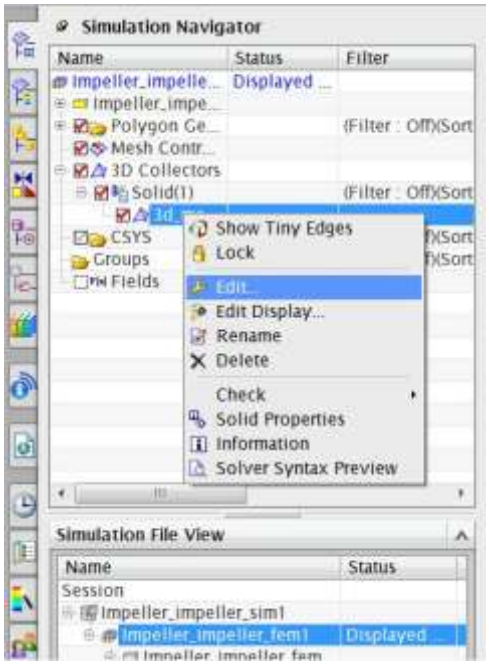
There are two ways to improve the accuracy of FEA results.

- Reduce the size of element
- Increase the order of interpolation polynomial (i.e. use quadratic or even cubic instead of linear polynomials)

The second approach is preferred because it is more efficient in terms of computation time and takes less memory space. However, let us try to create a scenario using the first option.



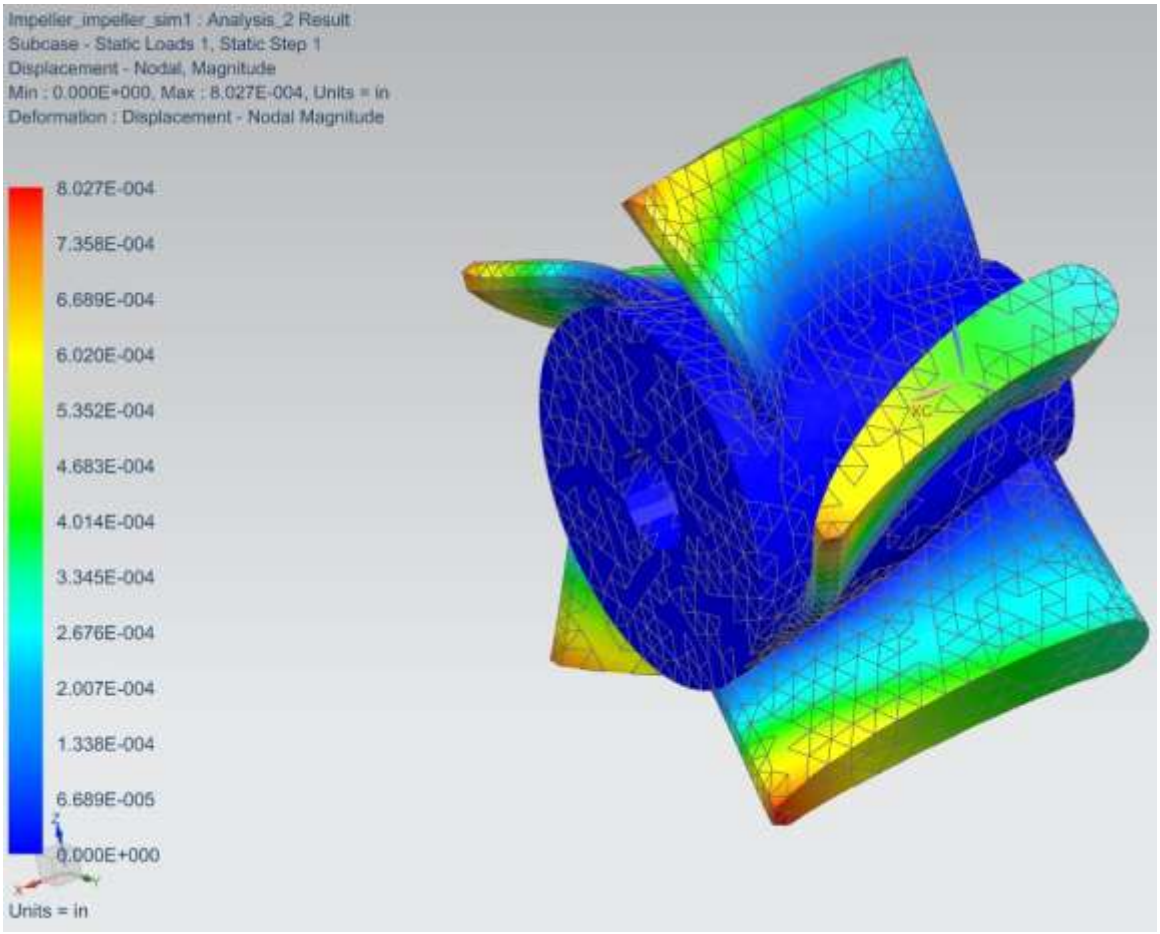
- Right-click on **Analysis_1** in the **Simulation Navigator**
- Choose **CLONE** to copy the first scenario
- Choose **OK** on the **Message** box
- Once **Copy of Analysis_1** is created, rename it to **Analysis_2**
- Go to .fem1 file in the **Simulation File View**.
- Right click on the **3D Mesh (1)** and click ***Edit***
- In the dialog box shown, change the Type to **TETRA4**
- Choose **OK**
- Go to .sim1 file in the **Simulation File View**
- Click on the **Solve** icon to solve the scenario
- Click **OK**



The **Analysis Job Monitor** should show the status of **Analysis_2** to be ***Completed.***

- Click **CANCEL**
- In the **Simulation Navigator**, double-click on **Results** for **Analysis_2**

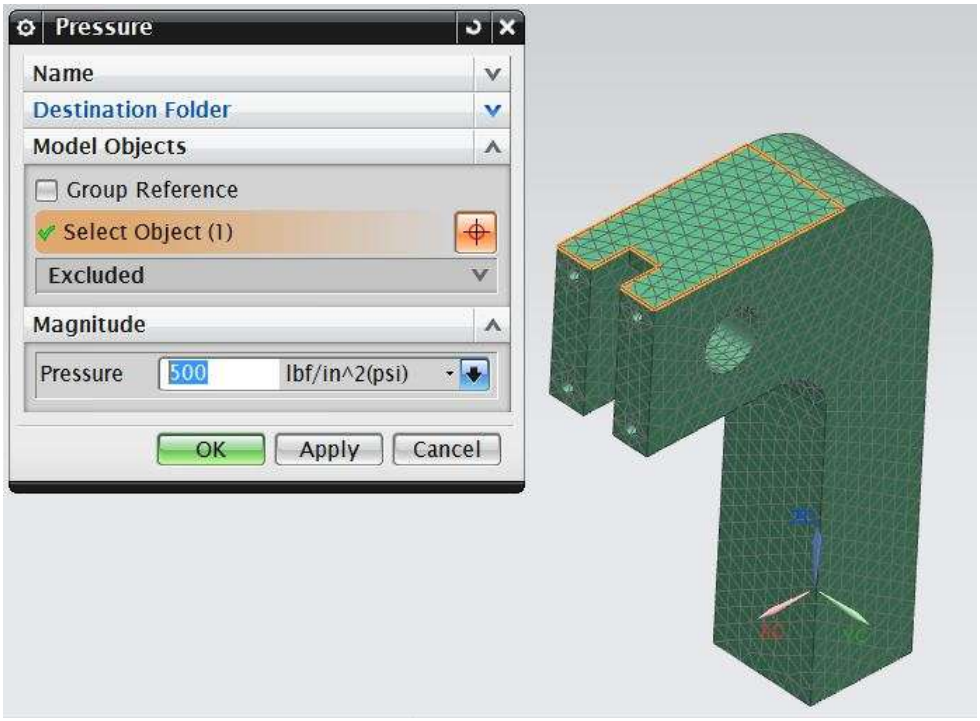
The figure below shows the analysis. You can observe the change in the maximum deviation. Save all the scenarios and close the files.


Impeller_impeller_sim1 : Analysis_2 Result
Subcase - Static Loads 1, Static Step 1
Displacement - Nodal, Magnitude
Min : 0.000E+000, Max : 8.027E-004, Units = in
Deformation : Displacement - Nodal Magnitude
8.027E-004
7.358E-004
6.689E-004
6.020E-004
5.352E-004
4.683E-004
4.014E-004
3.345E-004
2.676E-004
2.007E-004
1.338E-004
6.689E-005
0.000E+000
Units = in

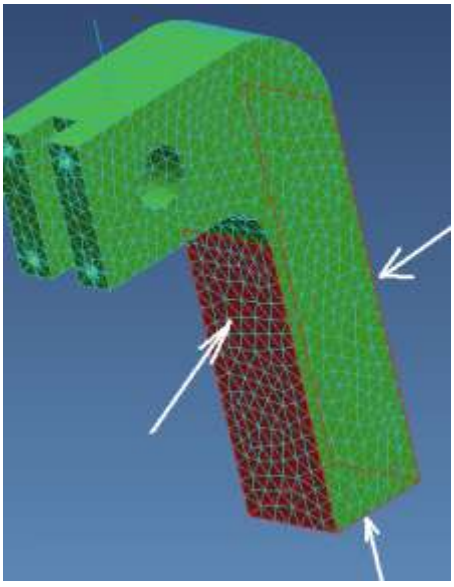
9.4 EXERCISE - ARBORPRESS L-BAR

Open the file '**Arborpress_L-bar.prt**' and do a similar structure analysis, considering the material as steel. For the mesh, the element size should be 10.00 and the type Tetra10.

For the loads, apply a normal pressure with a magnitude of 500 on the top surface as shown in the 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 following figure.



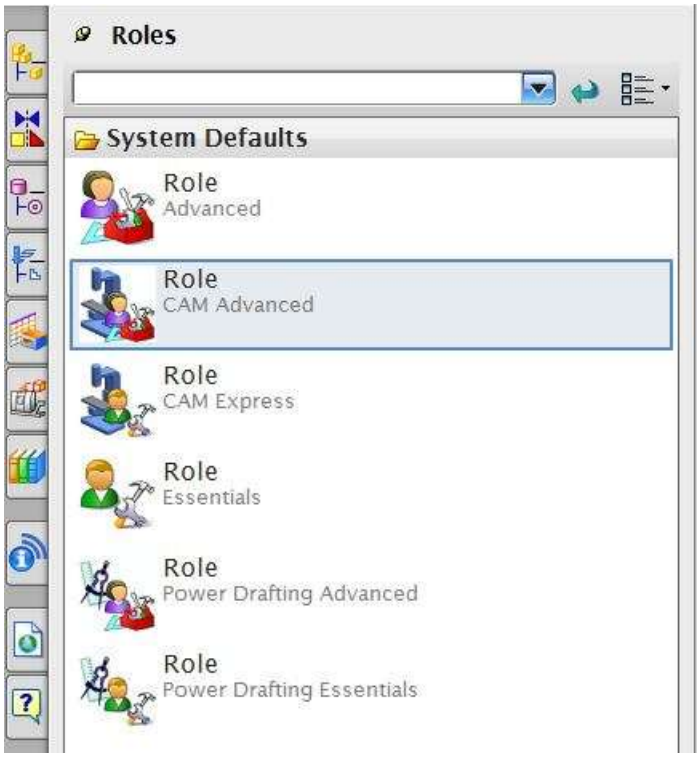
CHAPTER 10- MANUFACTURING

As we discussed in Chapter 1 about the product realization process, the models and drawings created by the designer have to undergo other processes to get to the finished product. This being the essence of CAD/CAM integration, the most widely and commonly used technique is to generate program codes for CNC machines to mill the part. This technological development reduces the amount of human intervention in creating CNC codes. This also facilitates the designers to create complex systems. In this chapter, we will cover the Manufacturing Module of NX 9.0 to generate CNC codes for 3-Axis Vertical Machining Centers. The manufacturing module allows you to program and do some post-processing on drilling, milling, turning and wire-cut EDM tool paths.

10.1 GETTING STARTED WITH MANUFACTURING MODULE

A few preparatory steps need to be performed on every CAD model before moving it into the CAM environment. Throughout this chapter, we are going to work with one of the models that were given in the exercise problems. For a change, 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** and click on the **INDUSTRY SPECIFIC** tab. A drop-down menu will pop up in which the **CAM Advanced** role can be seen as shown in the figure.



10.1.1 Creation of a Blank

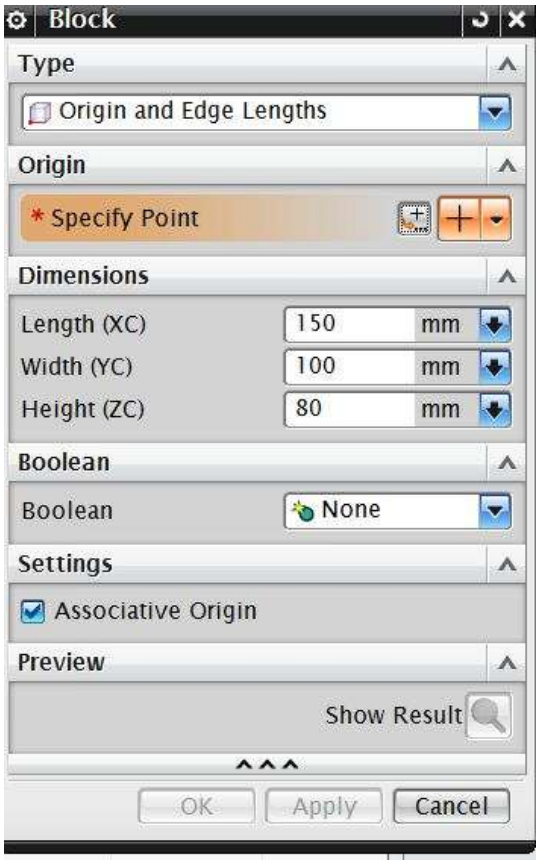
After completing the modeling, you should decide upon the raw material shape and size that needs to be loaded on the machine for the actual machining. This data has to be input in NX9.0. This can be achieved in two ways. The first method is by creating or importing the model of the raw material as a separate solid in the same file and assigning that solid as the **Blank**. The second method is by letting the software decide the extreme dimensions of the designed part and some offset values if wanted. The later method allows a quick way of assigning the raw size details but it can only be used for prismatic shapes.

- Open the file '**Die_cavity.prt'** of the exercise problem in Chapter 6

- Click on **FILE** → **MODELING**(under preferences)

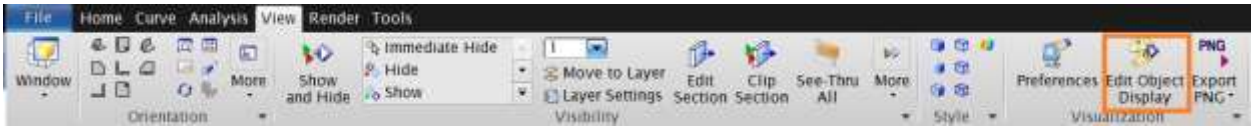
- ***INSERT*** a block with the following dimensions and positioning.
  Length = **150** mm
  Width = **100** mm
  Height = **80** mm

- In the **Point Constructor** icon located on the toolbar choose the lower most edge of the base block, so that the new block created wraps up the whole previous model.

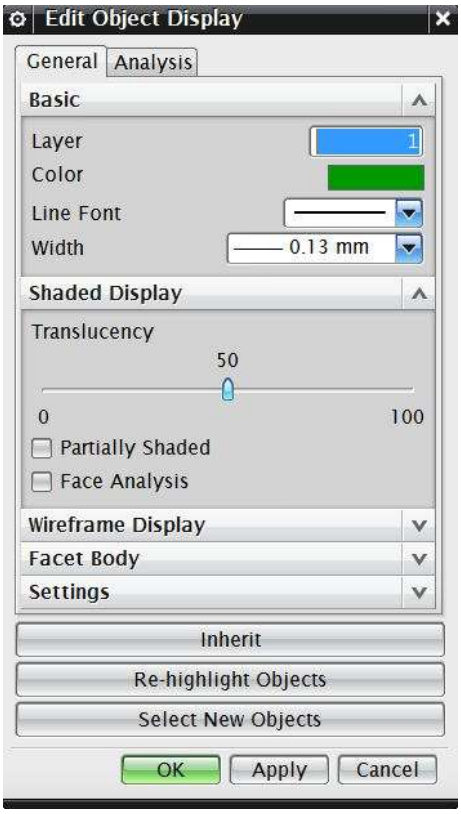


This block encloses the entire design part so we will change the display properties of the block.

- Click on the **EDIT OBJECT DIPLAY** icon on the **VIEW** toolbar as shown.



Two features, labeled as **BLOCK**, show up in the **QUICKPICK** menu, one for the design part and one for the block.



- Move your mouse on the labels to see which one represents the block you created just now.
- Select the block you created
- Click **OK**
- When the window pops up, change the display color and change the **Translucency** to **50**
- Then click **OK**

Hide the block you just created by right clicking on the block in the Part Navigator. This will make the raw block disappear from the environment. Whenever you want to view or work on this solid, reverse the blanks. This is done by pressing **<Ctrl> + <Shift> + B**.

10.1.2 Setting Machining Environment

Now we are set to get into the Manufacturing module.

- Select **FILE → NEW → MANUFACTURING → MILL TURN**

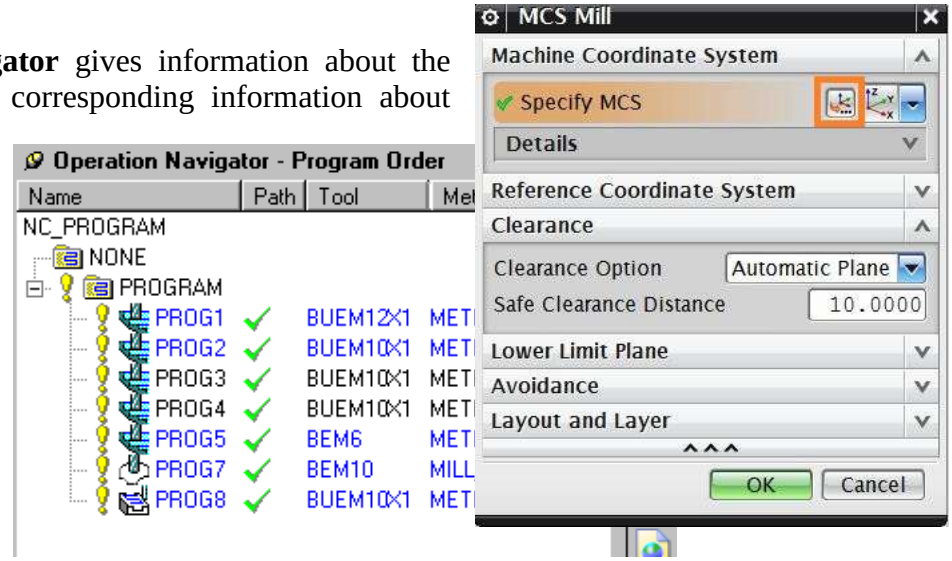
There are many different customized CAM sessions available for different machining operations. Here, we are only interested in the Milling operation.

10.1.3 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 right on the **RESOURCE BAR**

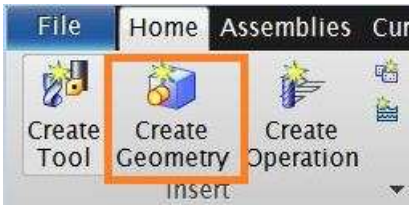
The **Operation Navigator** gives information about the programs created and corresponding information about the cutters, methods, and strategies.



The list of programs can be viewed in different categorical lists. There are four ways of viewing the list of programs in the **Operation Navigator**. The four views are ***Program Order view***, ***Machine Tool view***, ***Geometry view*** and ***Machining Method view***. Click on geometry view

10.1.4 Machine Coordinate System (MCS)

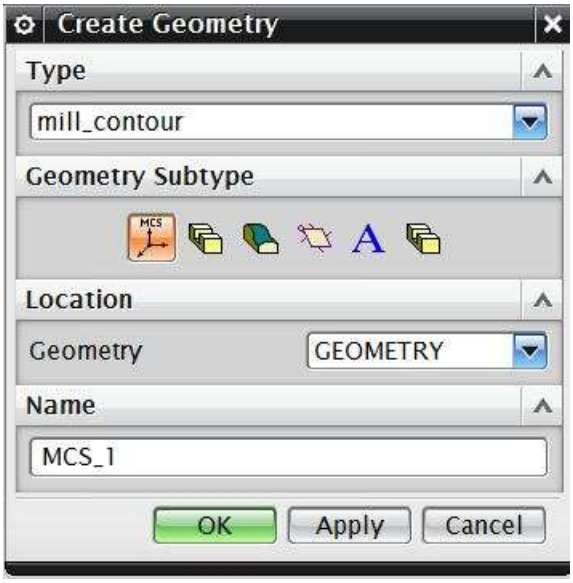
- Click on the **Create Geometry** icon in the toolbar to initiate setup for programming



You will see a **Create Geometry** pop-up.
You should be able to see the **mill_contour** as the program name in the **Operation Navigator**. If you do not see it, click on the **Geometry View** button in the Toolbar again.

- Click **OK**

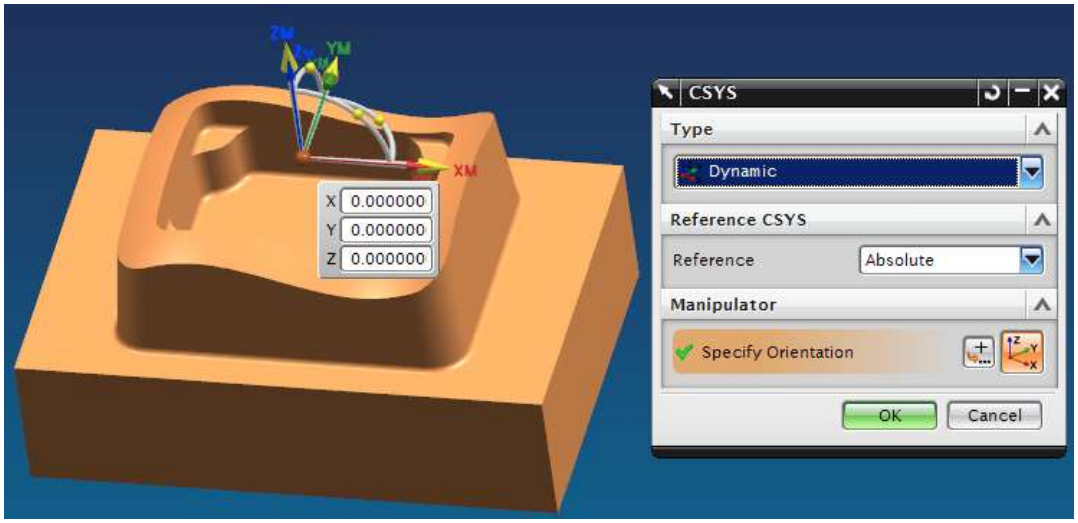
Another pop-up window will allow you to set the **MCS** wherever, you want. By default NX9.0 takes the original AbsoluteCS as the MCS.



- Click on the button shown. This will highlight the default WCS of the part and assign it as the **MCS**

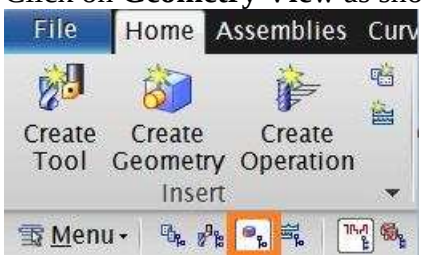
- Click **OK** to select it as the **MCS**

- Click **OK** when you are done orienting and positioning the MCS



10.1.5 Geometry Definition

- Click on **Geometry View** as shown below**.**

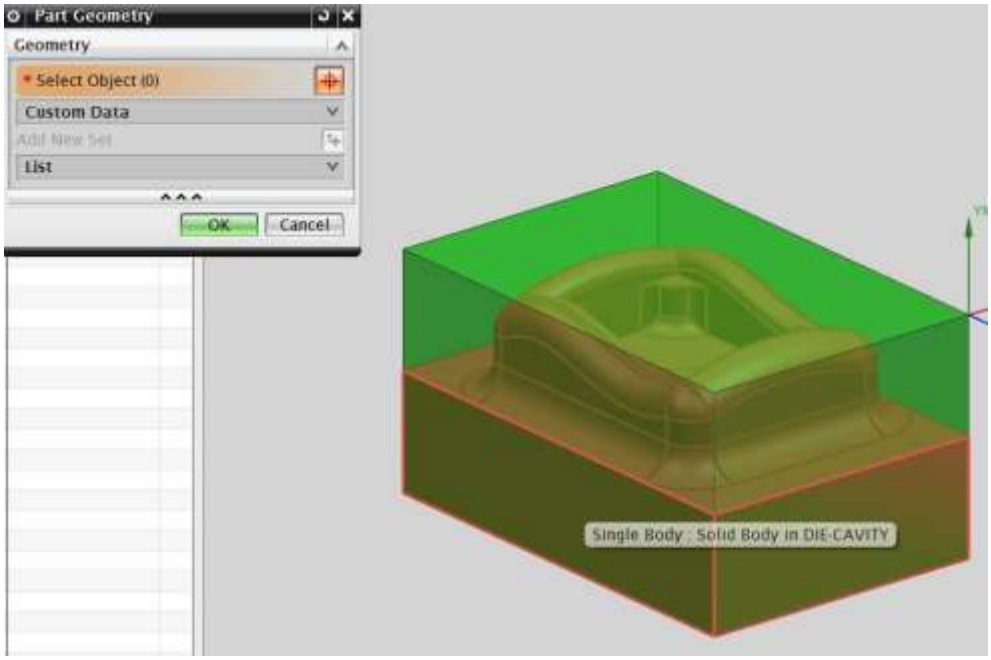


- **Expand all + signs in the operation navigator**
- **Double-click** on **WORKPIECE** in the **Operation Navigator**. If you don't see it, click on the plus sign next to **MCS_MILL**

The pop up window ***MILL GEOM*** appears. This is where you can assign the part geometry, blank geometry, and check geometry if any.

- Click on the **Part** icon

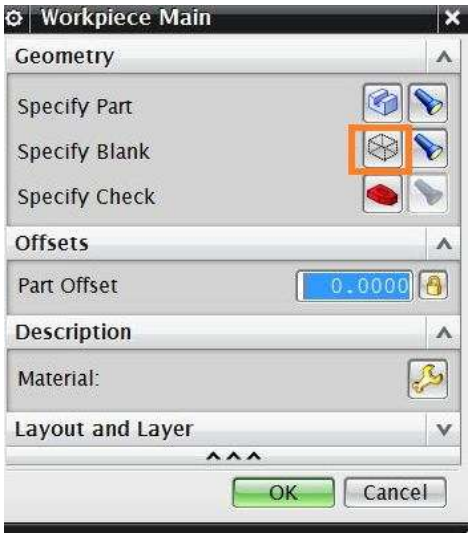
- Select the design part and click **OK**

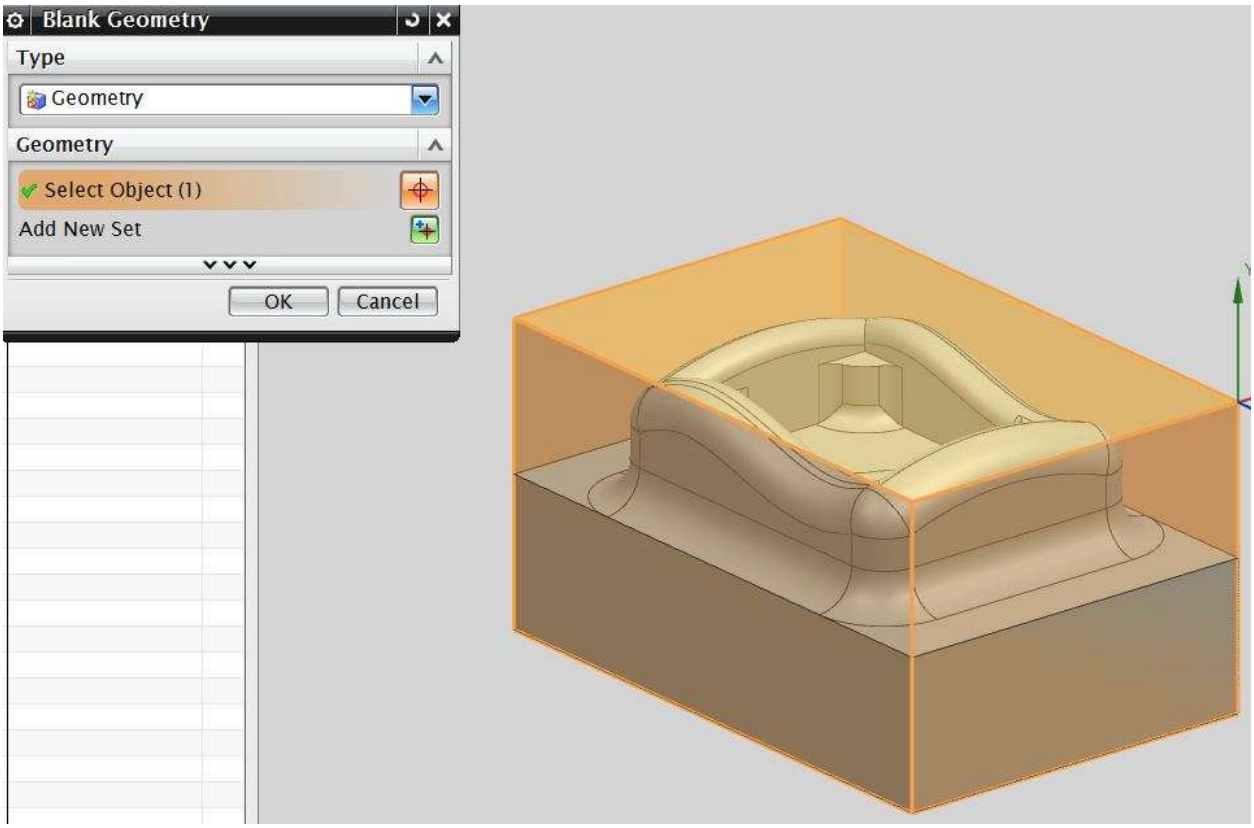


Now we have to select the **Blank** Geometry.

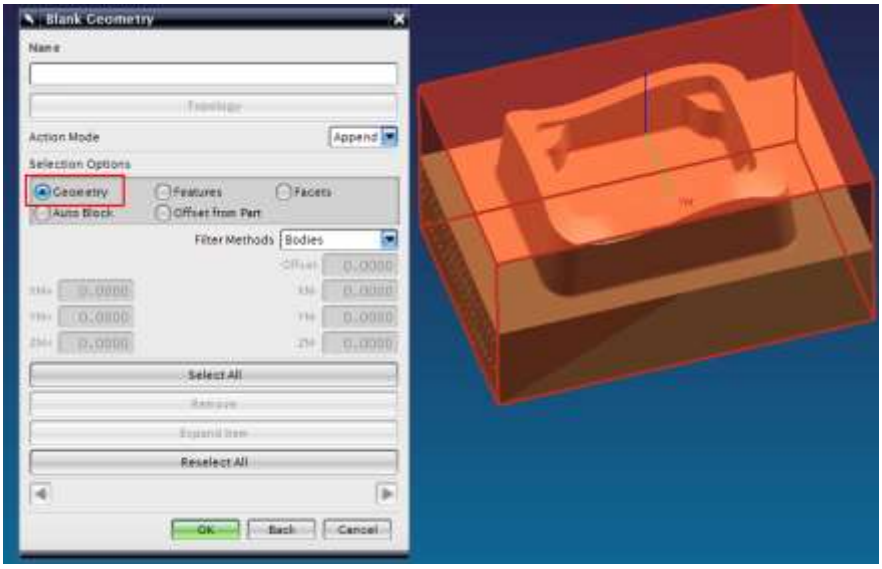
- Click the **Blank** icon

This will open the ***Blank Geometry Window***. As mentioned earlier there are many ways to assign 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 Rectangular Solid we can use that as the Blank geometry.





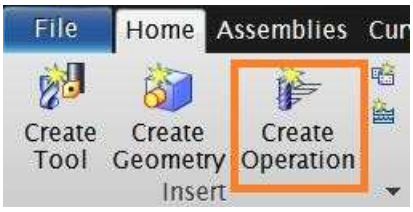
Now we are finished assigning the Part and Blank geometries. Sometimes it may be required to assign Check geometry. This option is more useful for shapes that are more complex or 5-Axes Milling operations where the Tool cutters have a higher chance of dashing with the fixtures. In our case, it is not very important to assign a Check Geometry.



10.2 CREATING OPERATION AND PARAMETER SETTING

10.2.1 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 the most efficient. Here, the basic guidelines are given for the most widely and frequently used strategies. The chapter will also cover important parameters that are to be set for the programs to function properly.

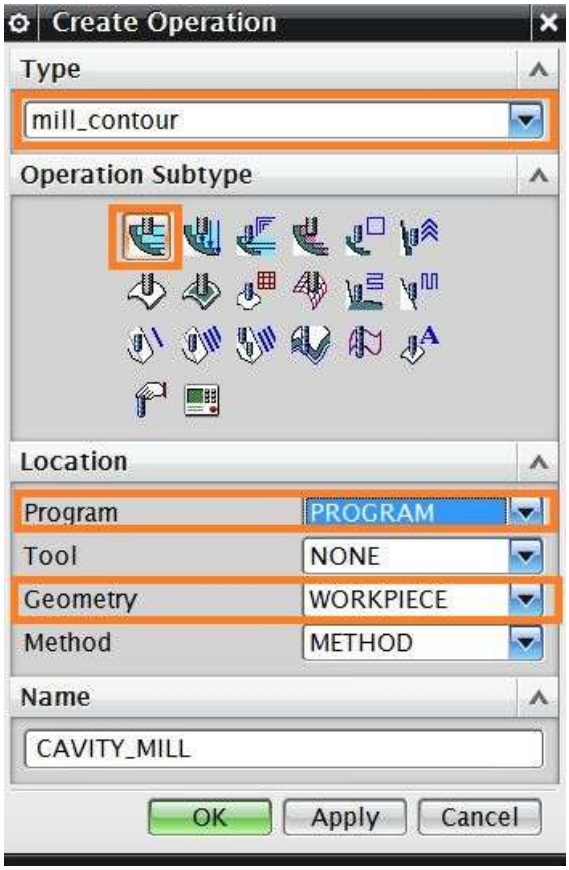


- Click on the **Create Operation** icon in the toolbar as shown

The **Create Operation** window will pop up.

- Make sure the **Type of Operation** is **Mill_Contour**

There are many different subtypes under Mill-Contour, namely Cavity Mill, Z-Level Follow Cavity, Follow Core, Fixed Contour, 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 any situation depends on your experience.



- Click on the **CAVITY_MILL** icon at the top left as shown in the figure
- Change the **Program** from **NC_PROGRAM** to **PROGRAM**
- Change the **Use Geometry** to **WORKPIECE**
- The program takes the default name **CAVITY_MILL**
- Click **OK**

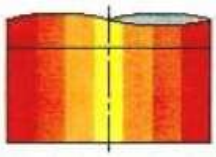
The program parameters window with CAVITY_MILL in the title bar will pop up. On this window, you can set all the parameters for the program. A brief introduction on every important parameter and terminology will be given as we go through the sequence.

10.2.3 Tool Creation and Selection

One of the most important decisions to make is to select the right shape and size of the tool 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 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 this need.

Flat End Mill Cutters:

These cutters have a sharp tip at the end of the cutter as shown in the figure. These cutters are used for finishing parts that have flat vertical walls with sharp edges at the intersection of the floors and walls.

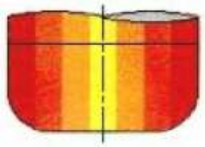


Ball End Mill:

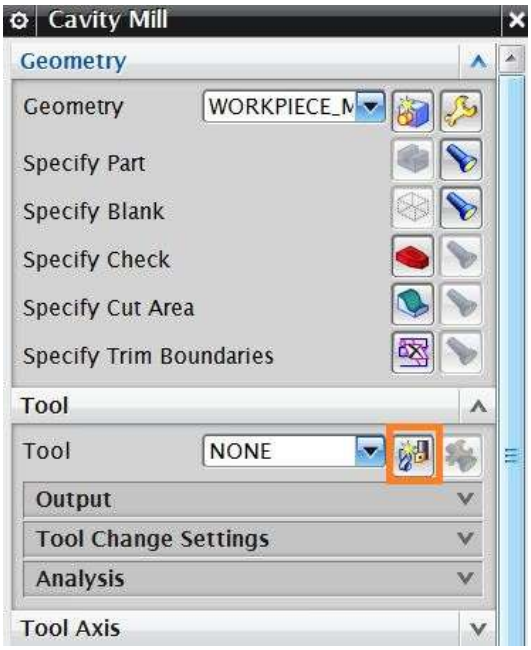
These cutters have the corner radii exactly equal to half the diameter of the shank. This forms the ball shaped profile at the end. These cutters are used for roughing and finishing operations of parts or surfaces with freeform features.

Bull Nose Cutters:

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.



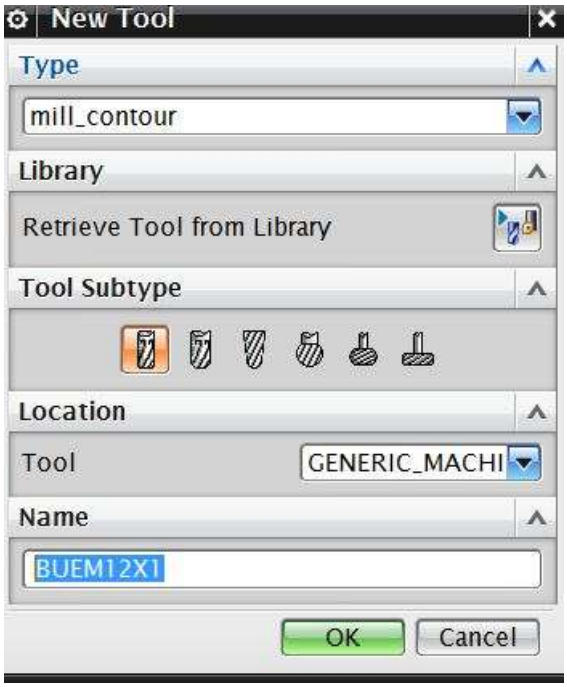
The cutter that we are going to use to rough out this huge volume is BUEM12X1 (Bullnose End Mill with 12 diameter and 1 corner radius).

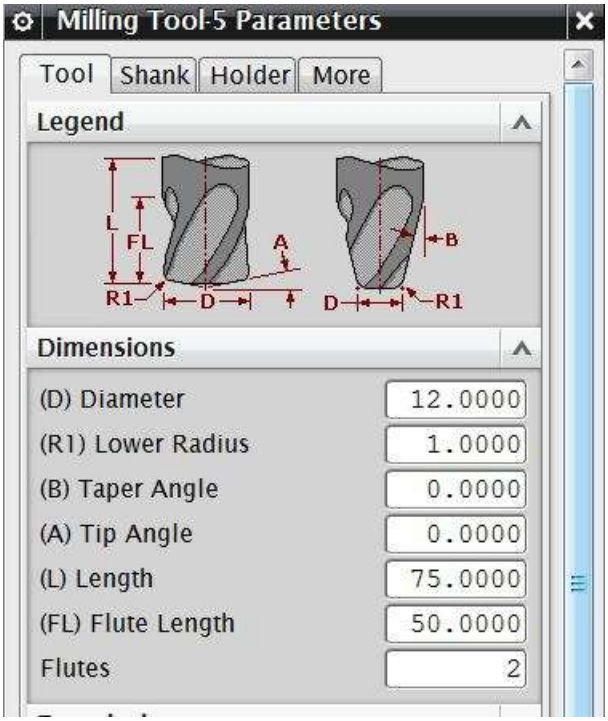


- In the **CAVITY_MILL** pop-up menu click on the **Create New** button in the **TOOL,** dialog box
- Click **NEW**
- On the **New Tool** window, select the **Mill** icon
- Type in **BUEM12X1** as the **Name** and click **OK**

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 cutters from the library.

- Enter the values as shown in the figure below.

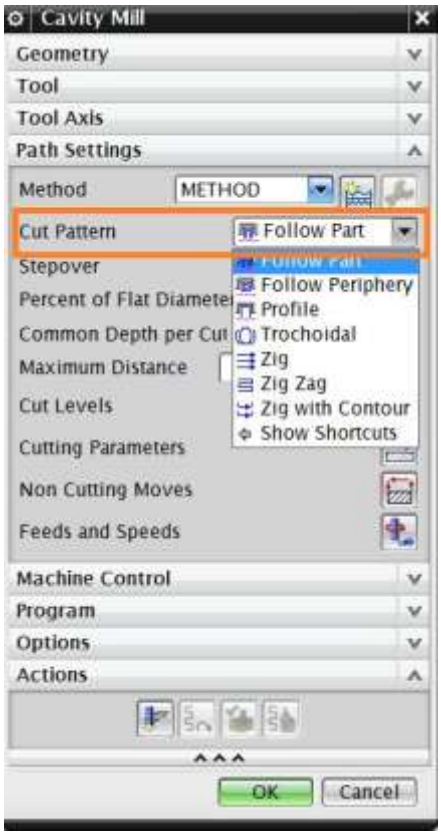




- Click **OK**
- In the **CAVITY_MILL** menu click on the **Path Settings** option

10.2.4 Tool Path Settings

There are different options in which the tool can move. The following is a description of each.



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: This takes a linear path in only one direction of flow.

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.

Follow Periphery: This takes the path depending upon the periphery profile. For example, the outer periphery of our part is rectangular. So the tool path will be generated such that it gradually cuts the material from outside to inside with the Stepover value. This option is mostly used for projections and cores rather than cavities.

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.

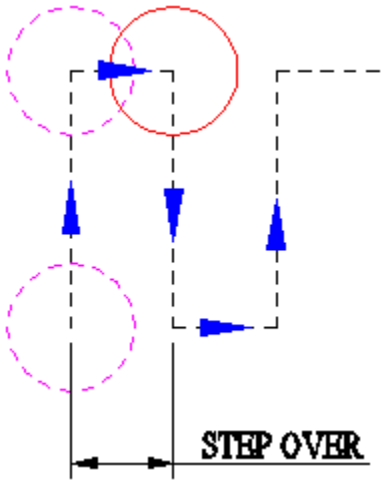
Trochoidal: This cutter is huge and is used for removing a large amount of material. The bulk of material is removed by gradual trochoidal movements. The depth of cut used will be very high for this strategy.

Profile: This takes the cut only along the profile of the part geometry. It is used for semi-finishing or finishing operations.

- For this exercise, select the **Follow Part** icon from the **Cut Pattern** drop-down menu since we have both projections and cavities in our part.

10.2.4 Step Over and Scallop Height:

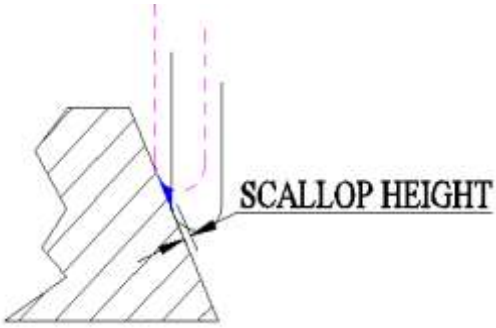
Step Over:

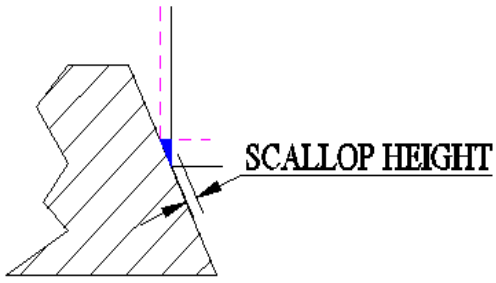


This is the distance between the consecutive passes of milling. It can be given as a fixed value or the value in terms of cutter diameter. The step-over should not be greater than the effective diameter of the cutter otherwise; it will leave extra material at every level of cut and result in an incomplete milling operation. The numeric value or values required to define the step-over will vary depending on the step-over option selected. These options include Constant, Scallop, Tool Diameter, etc. For example, **Constant** requires you to enter a distance value in the subsequent line.

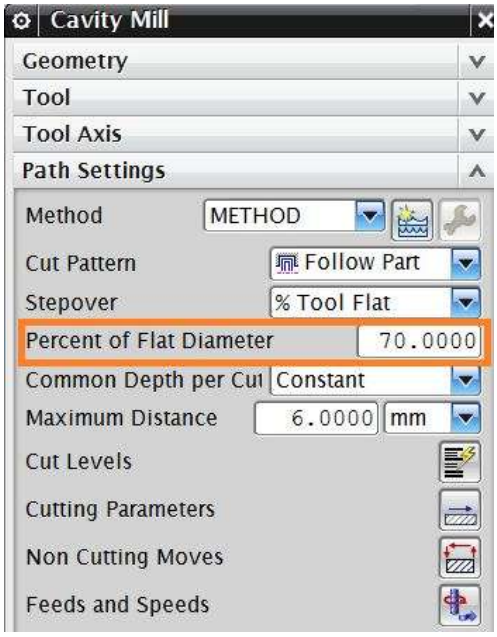
Scallop Height:

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.



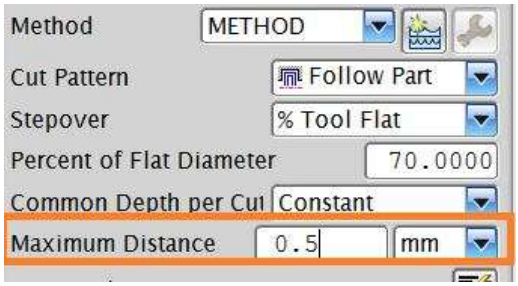


➢ For the **Step-over**, select **TOOL FLAT** and change the ***Percent*** to **70**.



10.2.5 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.

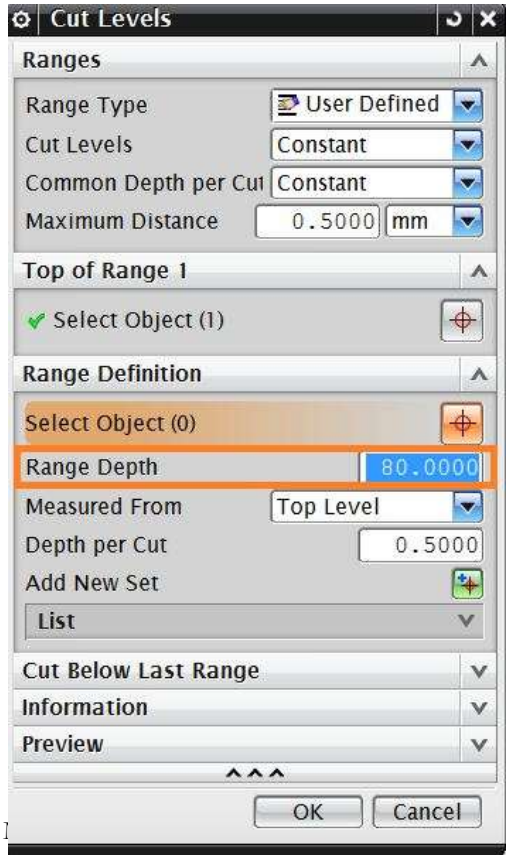


➢ Change the **Common Depth per Cut** value to be **0.5**

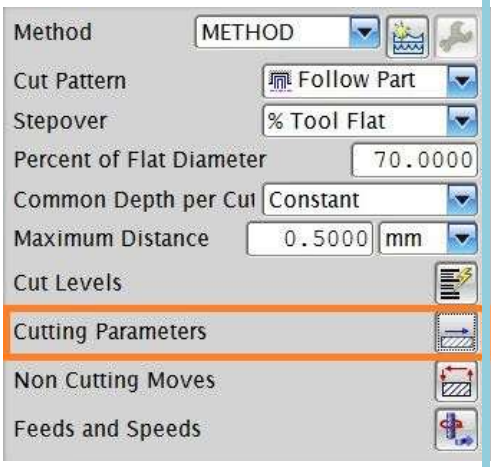
Now we will add the level ranges. This will split the part into different levels along the Z-direction to be machined.

➢ Click on **CUT LEVELS** as shown below

This will pop up a Dialog box for **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 Range Depth value (0-100). We are not going to mill up to the bottommost face of the Part, but up to the floor at 40mm from top. Therefore, we must delete the last level.



- Use the up and down scroll buttons until you reach the level that has a **Range Depth** of **80**
- Select **OK** after making these adjustments

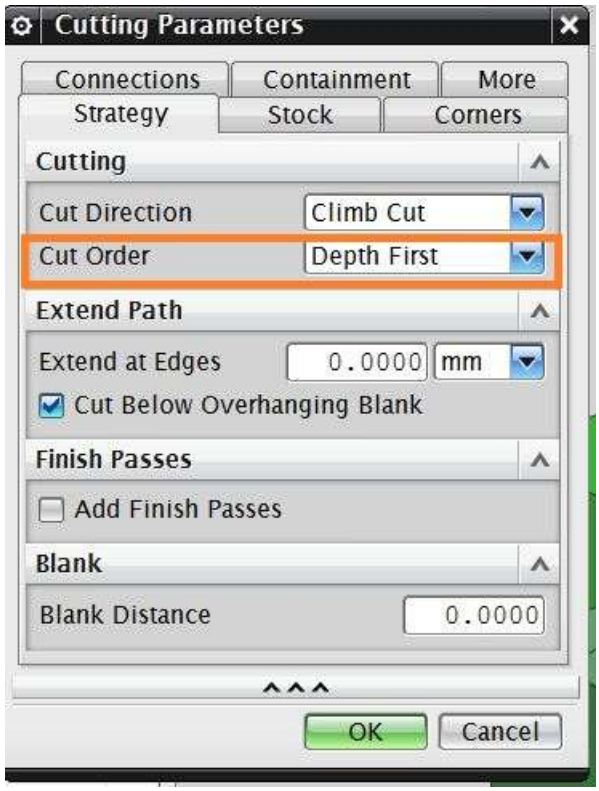


10.2.6 Cutting Parameters

- On the parameters window, click **CUTTING PARAMETERS**

This pops up another dialog box.

- Under the '**Strategy'** tab button, change the **Cut Order** from **Level First** to **DEPTH FIRST**

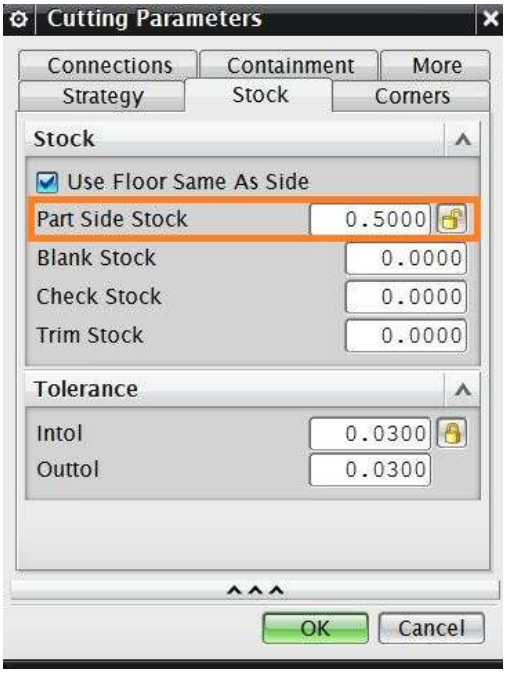


Changing the cut order to **Depth First** orders the software to generate the tool path such that it will mill one island completely up to the bottom-most depth before jumping to another level. The **Depth First** strategy reduces the non-cutting time of the program due to unnecessary retracts and engages at every 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**.

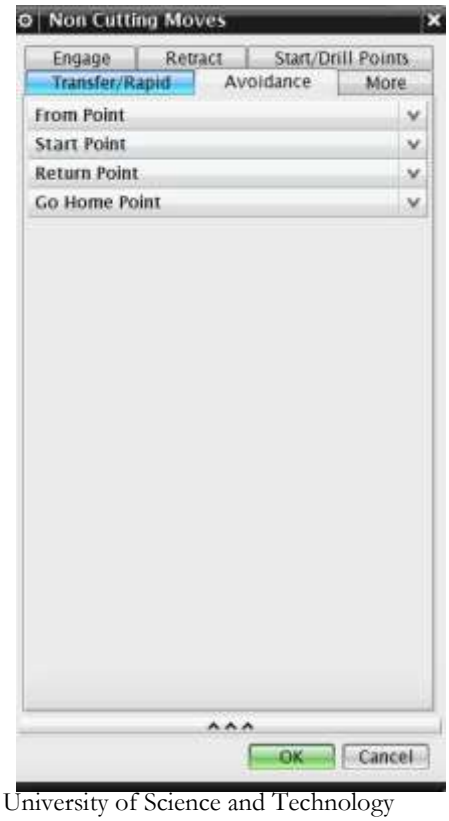
- Choose **OK**



10.2.7 Avoidance

- Click the **NON CUTTING MOVES**

- Click the **AVOIDANCE** tab



This window consists of many avoidance points like, ***Start Point***, ***Go Home Point***, etc. Of these, we are concerned with three points. They are as follows.

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 Constructor**, enter the coordinates of XC, YC and ZC as **(0, 0, 50)**

- Choose **OK**

- Choose **OK** again to go back to the **Avoidance** window

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 Constructor**

- Click **OK** to exit the **Point Constructor**

Clearance Plane:

This 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 top most 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**

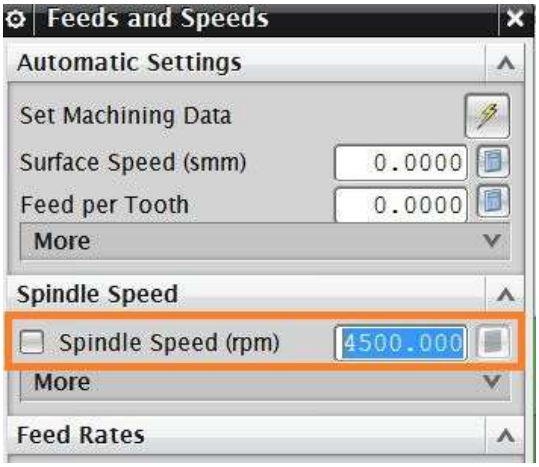
- Choose the **XY Plane** from the drop-down menu in **Type** tab

- Under the **Offset and Reference** tab enter the value as **3** in the **Plane Constructor** window

- **Click OK** twice to go back to the CAVITY_MILL parameters window

10.2.8 Speeds and Feeds

Choose **FEEDS AND SPEEDS** to enter the feed and speed parameters



Speed:

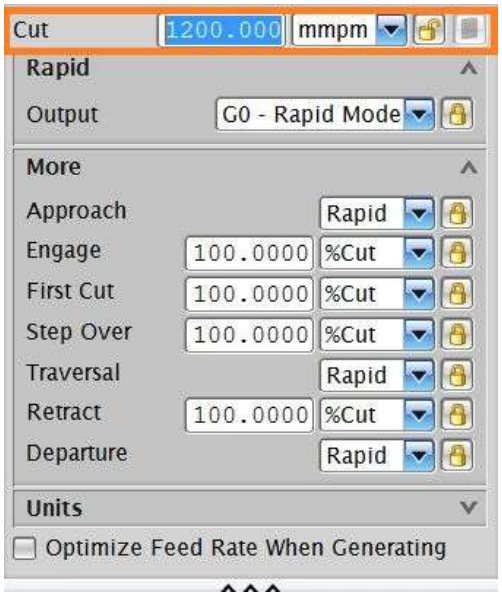
Speed normally specifies the rpm of the spindle (spindle speed). However, technically the speed refers to the cutting speed of the tool (surface speed). It is the linear velocity of the cutting tip of the cutter. The relative parameters affecting this linear speed are rpm of the spindle and the diameter 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.

- For this exercise, enter the values as shown in the figure. Make sure to enter the **Cut** value as **1200 mmpm.**

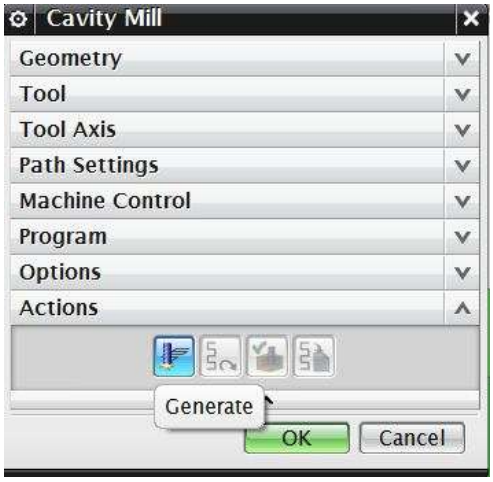
- Click **OK**

10.3 PROGRAM GENERATION AND VERIFICATION

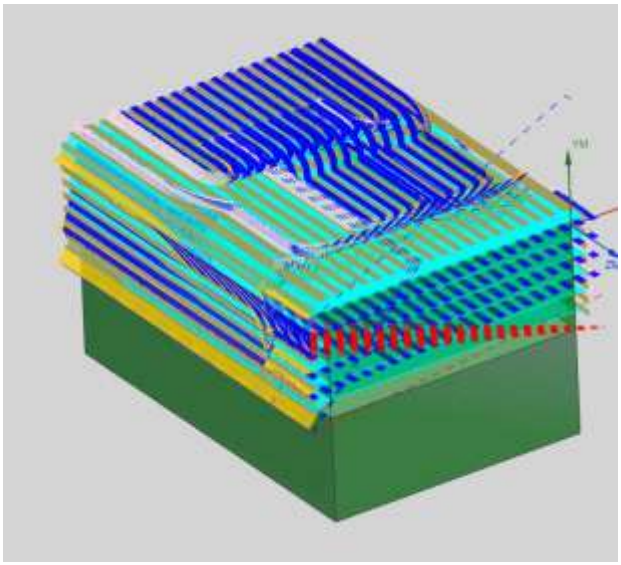
10.3.1 Generating Program

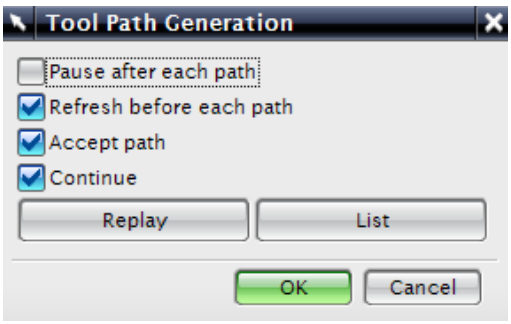
Now we are done entering all the parameters required for the roughing program. It is time to generate the program.

- 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.





During the generation, you may be prompted with a Display Parameters window.

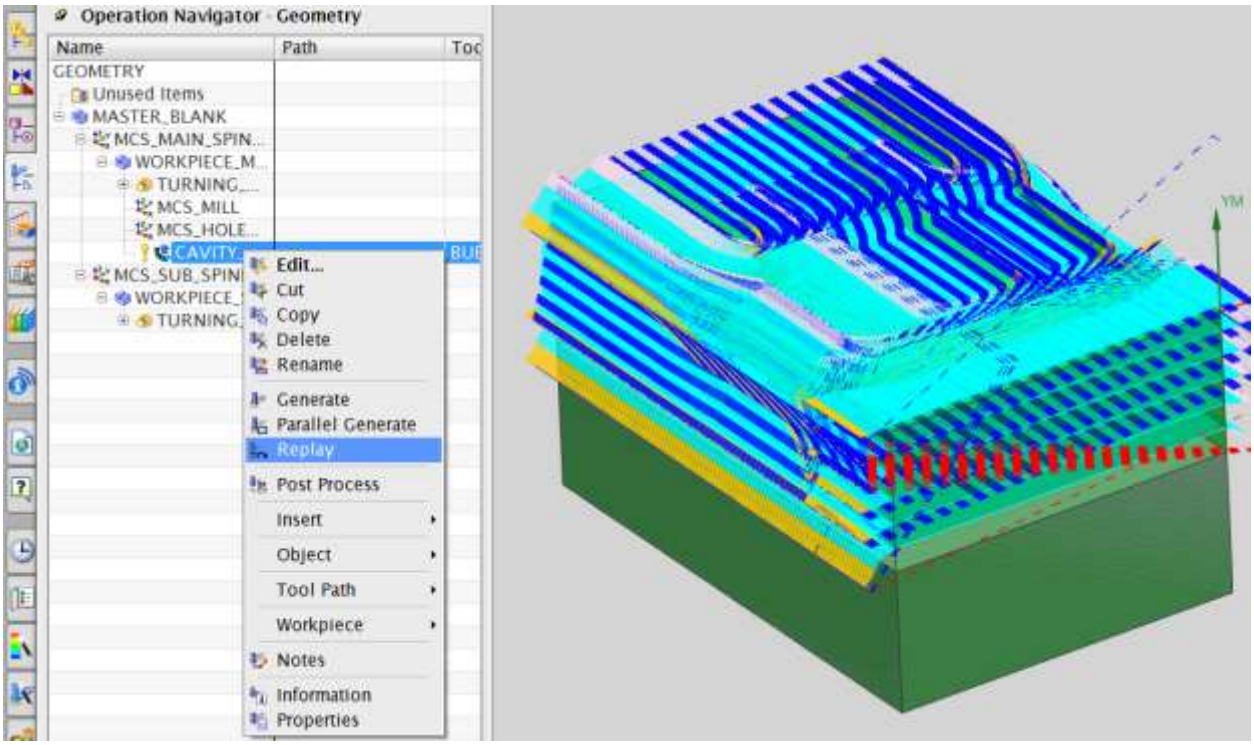
- Uncheck the box next to **Pause After Each Path**

- Then click **OK** to see the display of cut-levels and tool paths

- After the generation is done, click **OK** in the parameters window

10.3.2 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.



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.

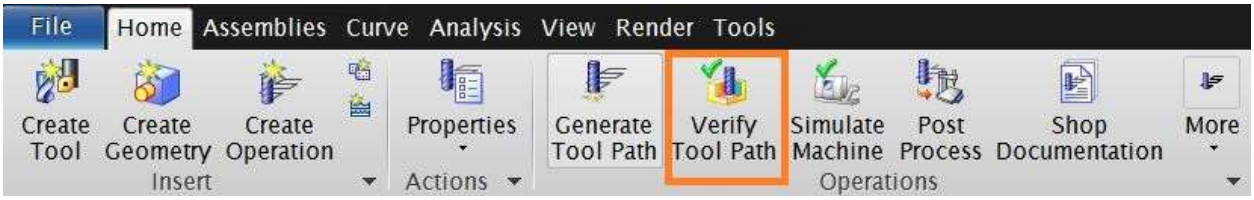
10.3.3 Tool Path Simulation

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, NX9.0 and other CAM software provide Tool-path verification and a Gouge check.

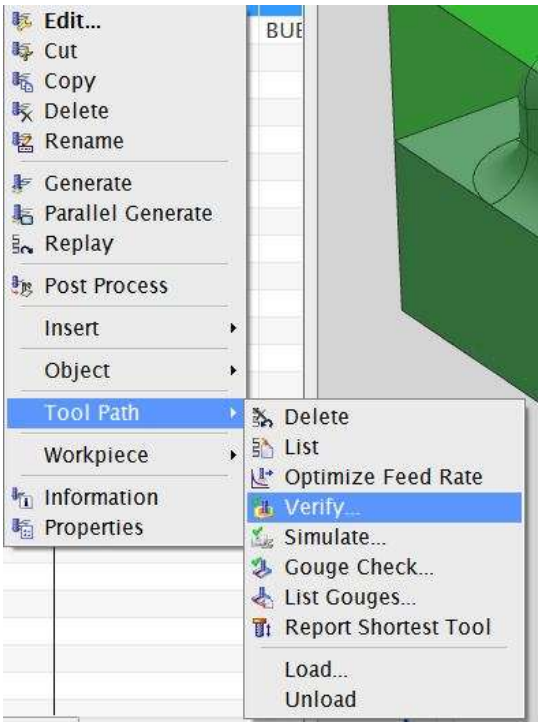
Verify:

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



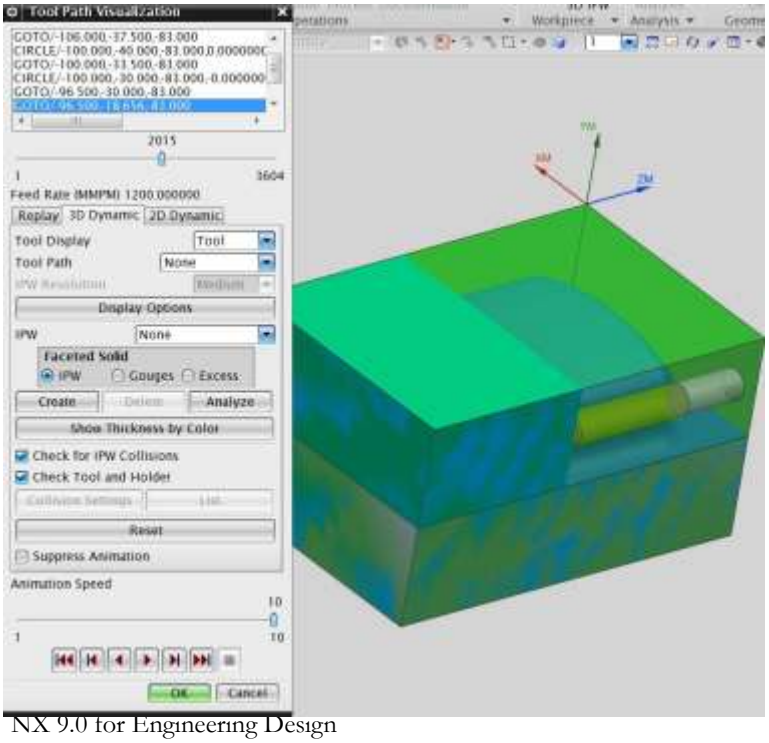
This will allow you to set the parameters for visualization of the Tool-Path.



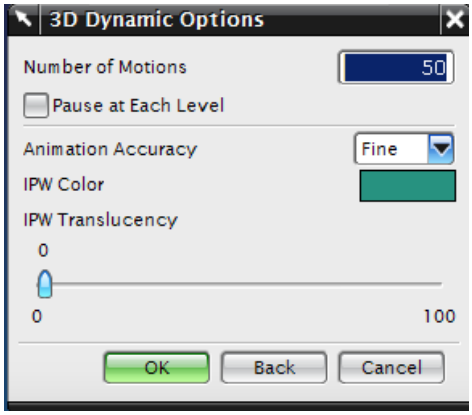
- On the **Tool Path Visualization** window, click on the **Play** icon to view the Tool Path motion

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



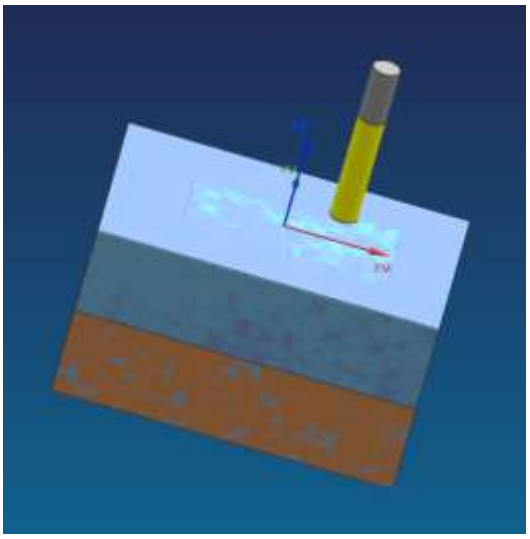
- Click on the **DISPLAY OPTIONS** button on the same window



- 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 below.

With this option, you will be able to view the actual cutting simulation and material removal through computer graphics. This is 3D Dynamic, where you can rotate, pan and zoom the simulation when it is playing. The cutting simulation is 3D.



Let us try the 2D Dynamic simulation. When this simulation is playing, you cannot do any other actions in NX9.0. Unlike 3D, you cannot rotate or zoom while playing. If you want to see the other side of the part, you have to stop the simulation, rotate and play again. This is faster than 3D Dynamic.

10.3.4 Gouge Check

Gouge Check is used to verify whether the tool is removing any excess material from the workpiece 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**

After the gouge check is completed, a message box will pop up saying "**No gouged motions were found**." If in case there are any gouges found, it is necessary to correct the program.

- Click **OK** on the message box

- Close the pop-up window, which says that there are no gauge motions found.

10.4 OPERATION METHODS

10.4.1 Roughing

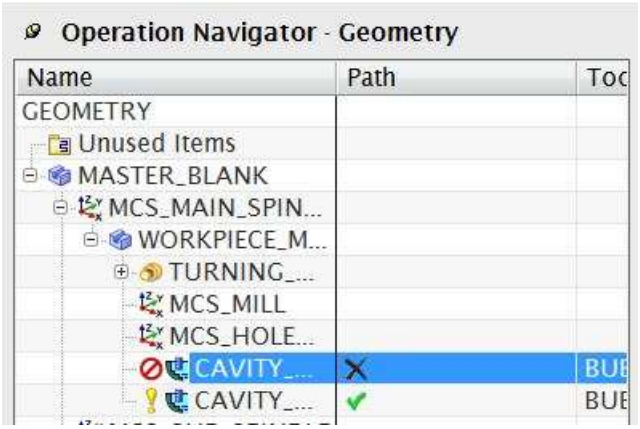
In case of milling operation, the first operation should be rough milling before finishing the job. The main purpose of roughing is to remove bulk material at a faster rate, without affecting the accuracy and finish of the job. Stock allowances are given to provide enough material for the finishing operation to get an accurate and good finish job. What we did in the earlier part of this chapter is generate a roughing program. Now we have to moderately remove all the uneven material left over from the previous program.

10.4.2 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 **CAVITY_MILL_1**



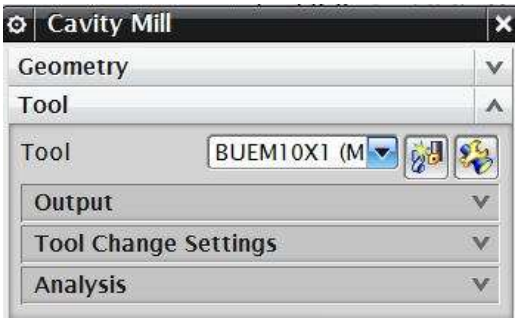
You can see that next to the newly created **CAVITY_MILL_1** is a red mark, which indicates that the program is not generated.

Cutter Selection:

Let us now set the parameters that need to be changed for the second program. Before we even 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

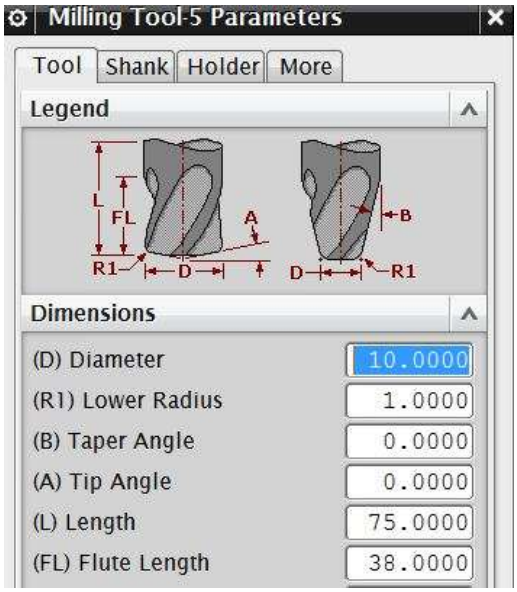
Just as we did in the previous program, we have to create a new cutter. In the **TOOL** tab, you will see the cutter you first chose. It will show BUEM12X1 as the current tool.



- Create a new cutter and name it **BUEM10X1**. It should have a diameter of **10** and a lower radius of **1**

- Click the **Common Depth per Cut** as **0.25** in the Path Settings tab.

Then click on **CUTTING PARAMETERS** tab

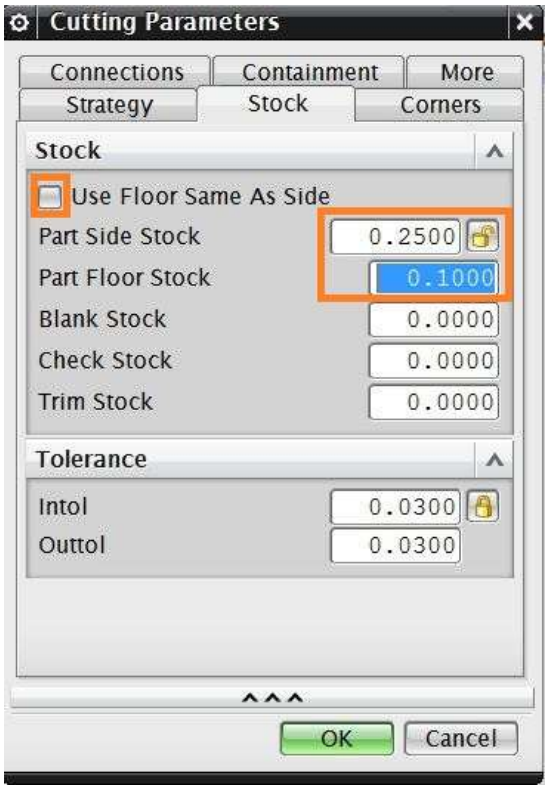


- Click on the **STOCK** tab button

- Uncheck the box next to **Use Floor Same As Side**

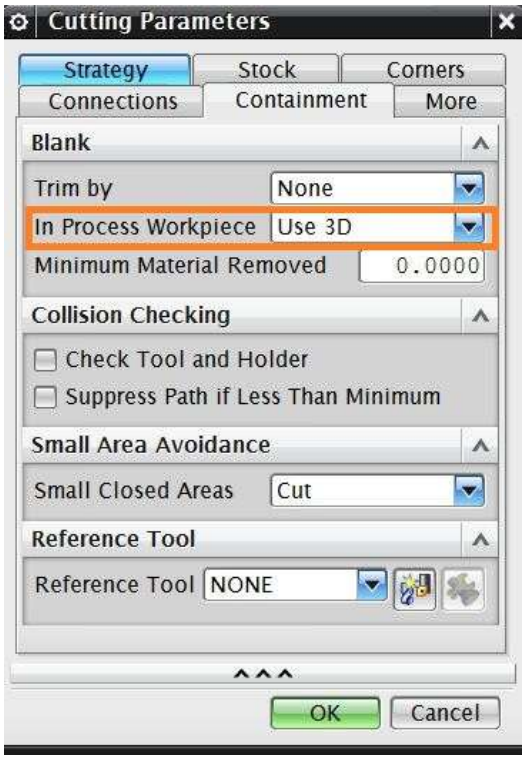
- Enter **0.25** for **Part Side Stock**

- 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**

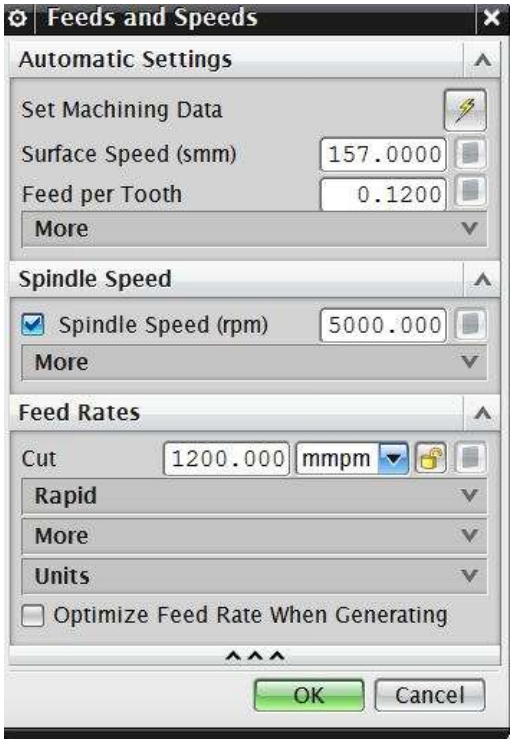


In-Process Workpiece is a very useful option in NX9.0. The software considers the previous program and generates the current program such that there is no unnecessary cutting motion in the ***No-material*** zone. This strategy reduces the cutting time and air cutting motion drastically. The algorithm will force the cutter to only remove that material, which was left from the previous program and maintain the current part stock allowance.

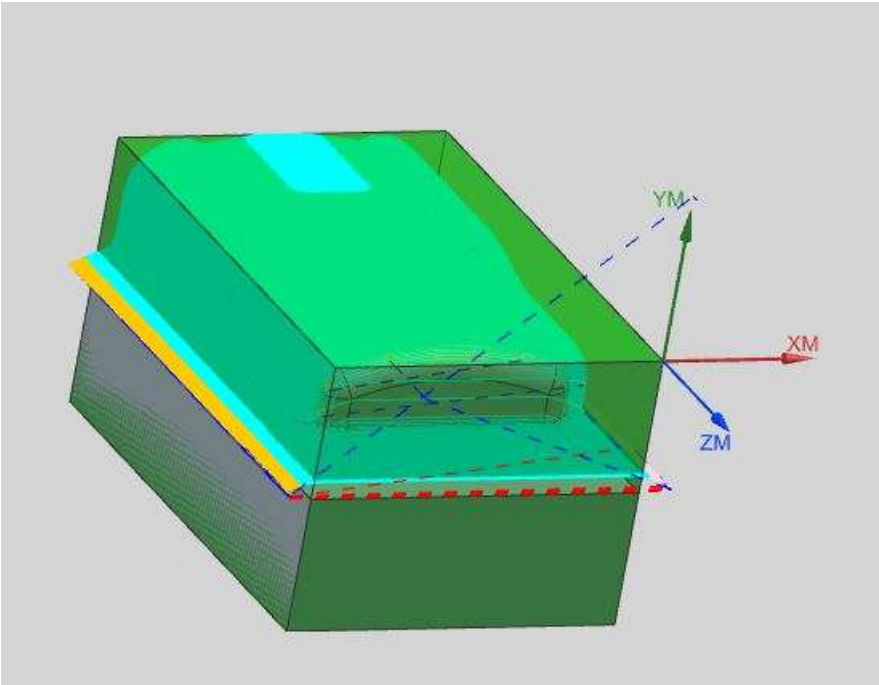
- Choose **OK** to return to the parameters window
- Click **FEEDS AND SPEEDS**
- Enter the Speed and Feed values as shown in the following figures
- Then click **OK**

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.

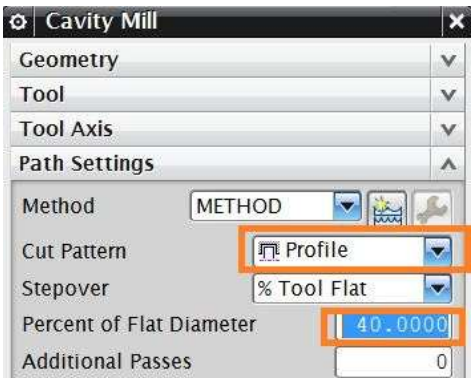


10.4.3 Finishing Profile

So far, we are done with the roughing and semi-finishing programs for the part. There is a sufficient amount of material left in the Workpiece to be removed in the finishing programs to obtain the accurate part geometry as intended in the design. 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.

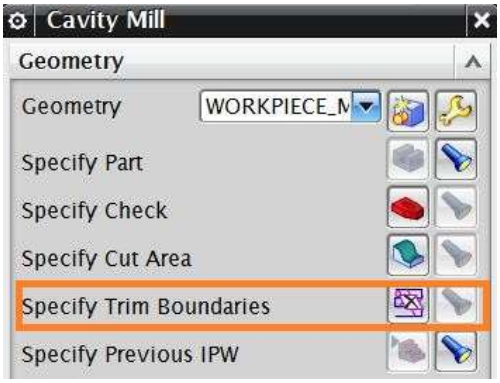
Outer Profile:

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 have to 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**

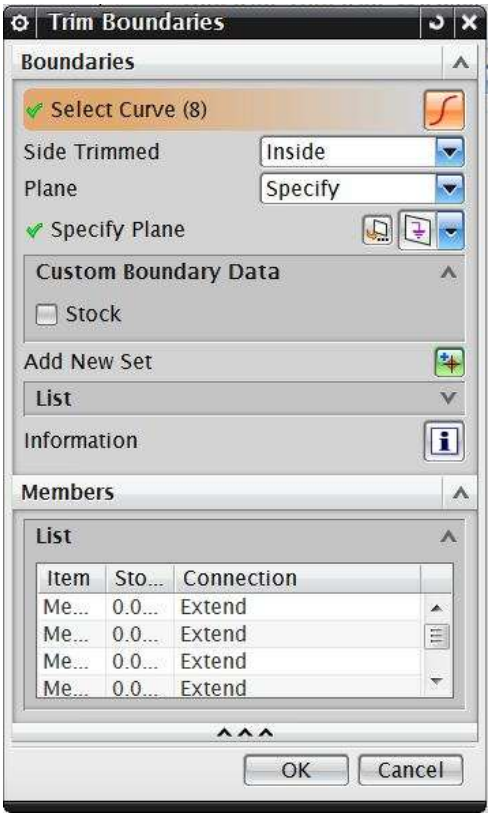
- Double click **CAVITY_MILL_2** to make parameter changes



- In the pop-up parameters window, change the **Cut Pattern** to **Profile** and the **Stepover percentage** to **40** as shown in the figure

- Click on the **Specify Trim Boundaries** tab

The **Trim Boundary** 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.



- Uncheck the **Ignore Holes** and check the box in front of **Ignore Islands**

- Change Filter Type to **CURVES**

- Change the **Plane** tab from **Automatic** to **Specify**

A new window will pop up as shown below. 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**

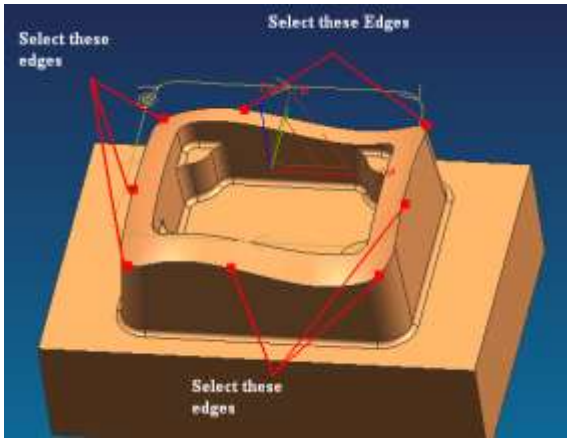
- Enter a value of **3** next to it

- Click **OK**

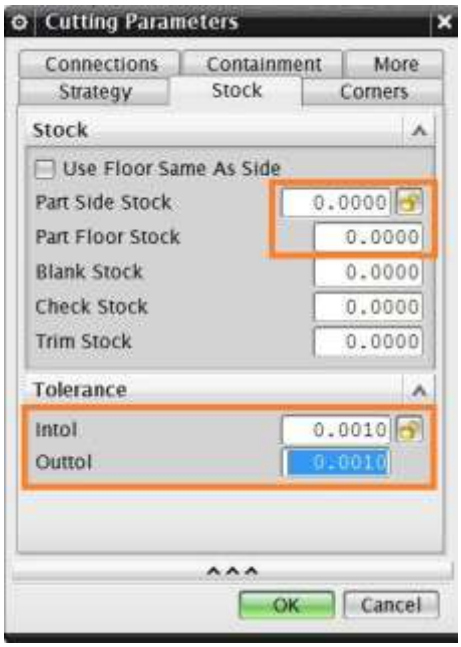
Now we will start selecting edges from the part. These selected edges will be projected on the Z = 3 plane as curves and used as the boundary.

- 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 pop up Dialog box, click on **STOCK** tab
- Enter the **Part Side Stock** and **Part Floor Stock** values to be **0.00**



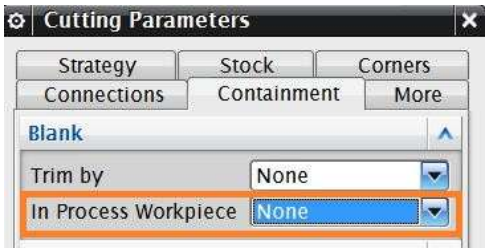
Intol:

Intol allows you to specify the maximum distance that a cutter can deviate from the intended path into the workpiece.

Outtol:

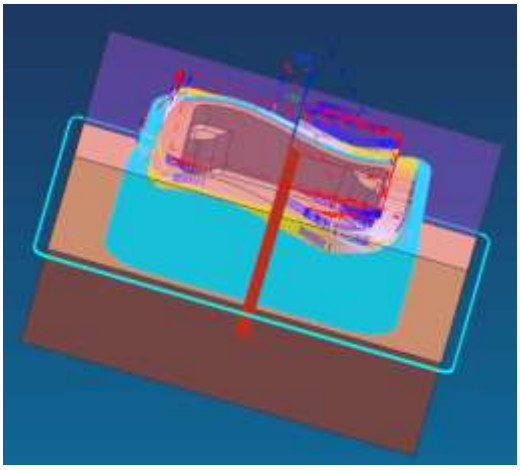
Outtol allows you to specify the maximum distance that a cutter can deviate from the intended path away from the workpiece.

- Enter the **Intol** and **Outtol** values to be **0.001** as shown in the figure
- Click on **CONTAINMENT** tab and change the **In-process Workpiece** to **NONE**



- 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.



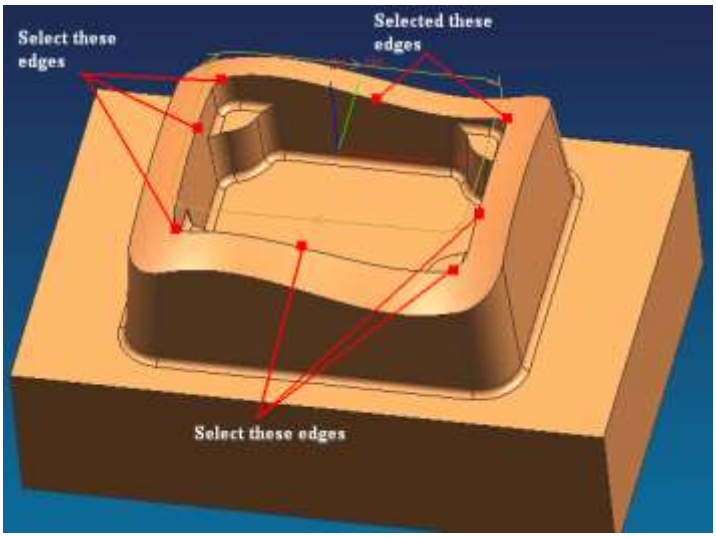
Inner profile:

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 pop up dialog box.

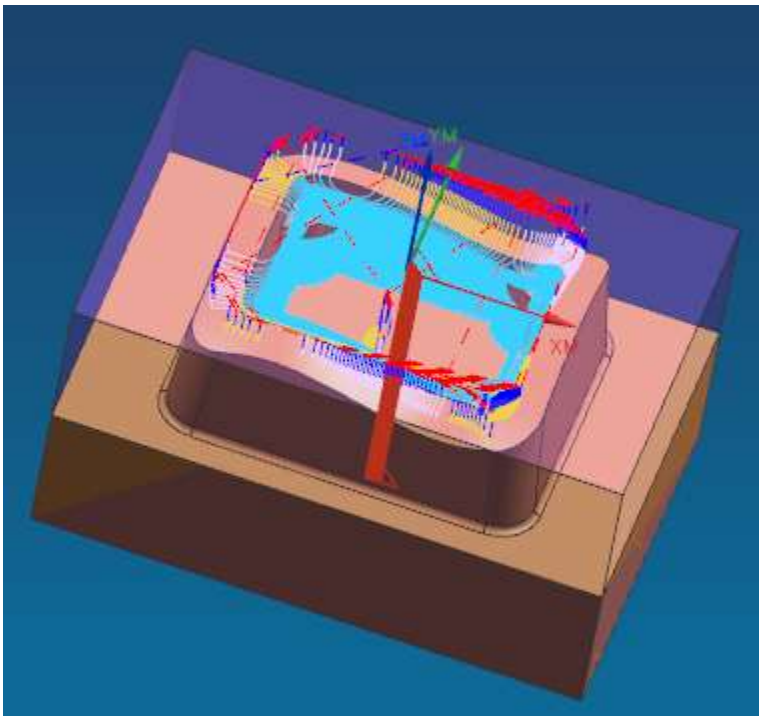
This will prevent the cutter from passing outside the boundary.

- Select the Filter Method to be **CURVES**
- Change the plane manually to be the **XC-YC** plane and enter the offset distance as **3**
- Click **OK**
- 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.
- Then click **OK**

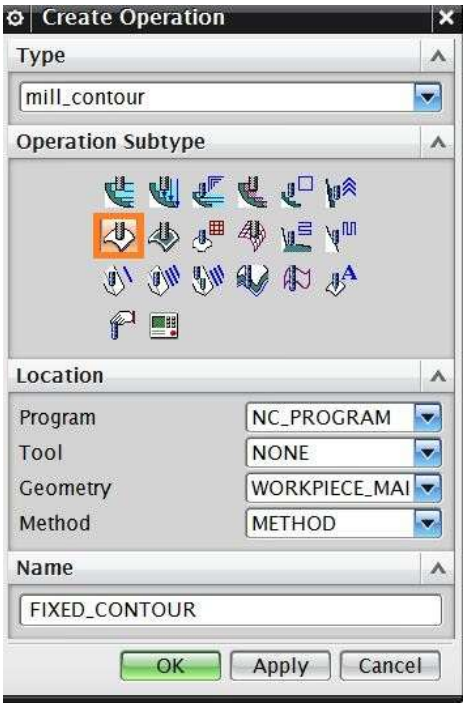


- Choose **OK** to return to the parameters window
- Generate the program.
- Click **OK** when the generation is finished.
- Click on **OK** if you get any warning message about the tool fitting

The finishing program for the outer 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.

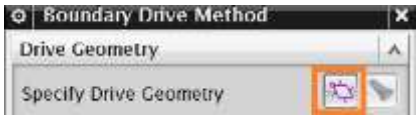


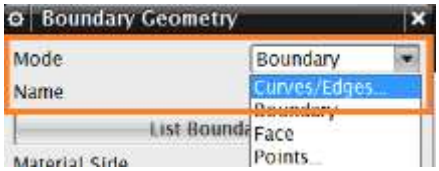
10.4.4 Finishing Contour Surface



Now we have to use a different type of strategy to finish the top freeform surface.

- Click on the **Create Operation** icon in the Toolbar
- Then click on the **FIXED_CONTOUR** icon as shown in the figure
- Choose **PROGRAM** for Program
- Choose **WORKPIECE** for Geometry
- Keep the default name of program
- Click **OK**
- On the parameters window, under **Drive Method**, select **BOUNDARY** even if it is already shown





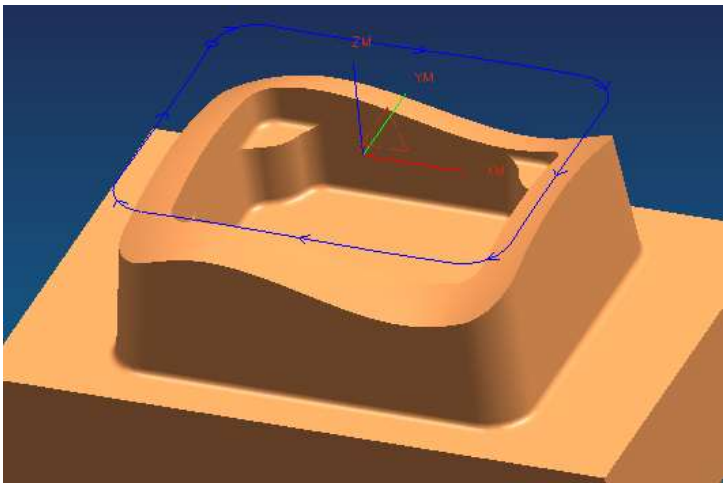
- Click on the **Spanner** icon as shown in the figure above to open the **Boundary Drive Method** menu
- On the **Create Boundary** window, change the ***Mode*** to **CURVES/EDGES**
- Select the **Material Side** to be **OUTSIDE**
- Select the **Tool Position** to be **ON**

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**

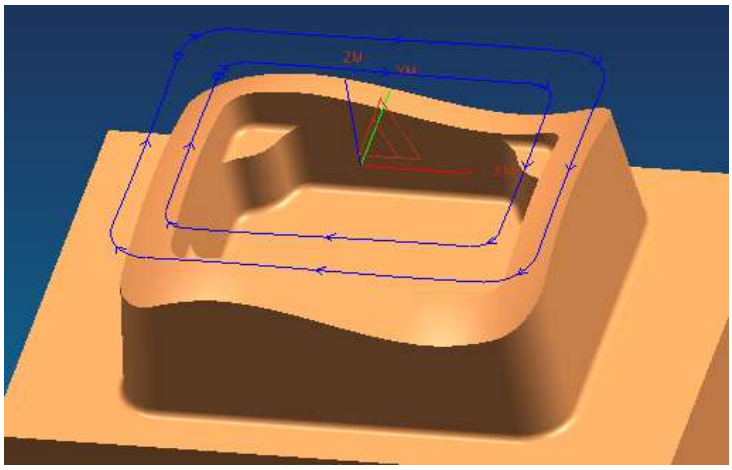
- Again, set the plane to be **XC-YC = 3**
- Click **OK**
- Select the outer loop of the top contour surface as shown in the figure. Remember to select the edges in a continuous order.



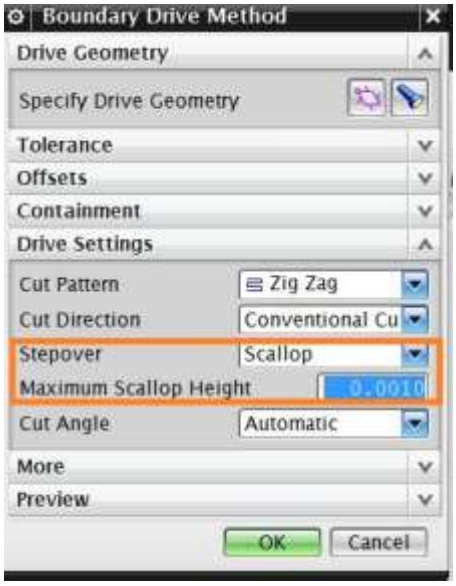
- Click **OK**

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 = 3**
- Select the inner edges of the contour surface as shown
- Click **OK** to return to the **Boundary Drive Method** window

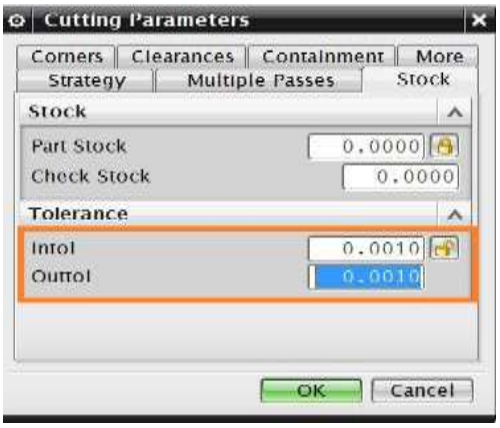


- Change the **Stepover** method to **SCALLOP** and enter the height to be **0.001** and click **OK**

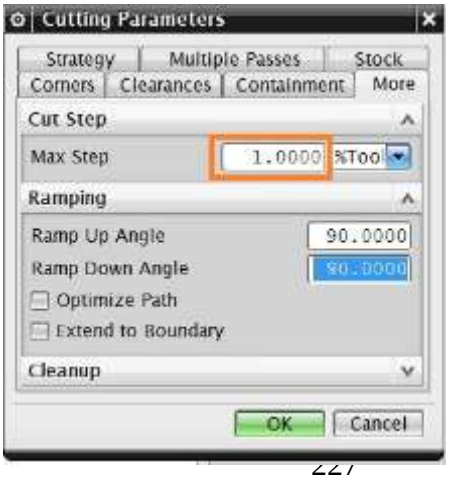


- Click on **Cutting Parameters**

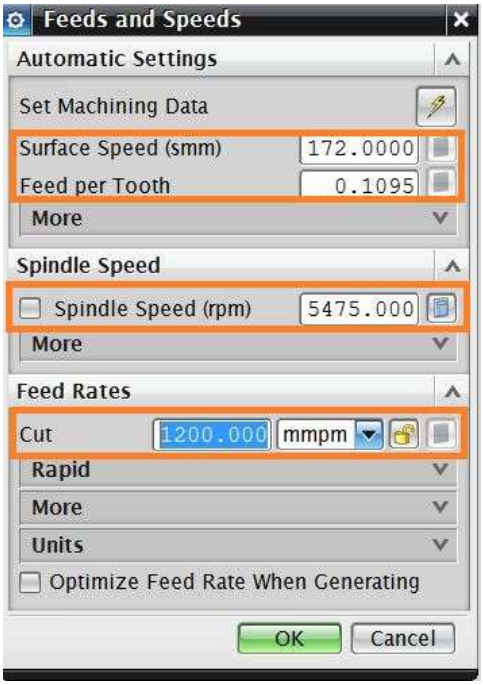
- Change the ***Tolerance*** values so that the **Part Intol** and **Part Outtol** is **0.001**



- Click on the **MORE** tab button and enter the value of **Max Step** as **1.0**



- Click **OK**
- Click on the **FEEDS AND SPEEDS** icon on the parameters window

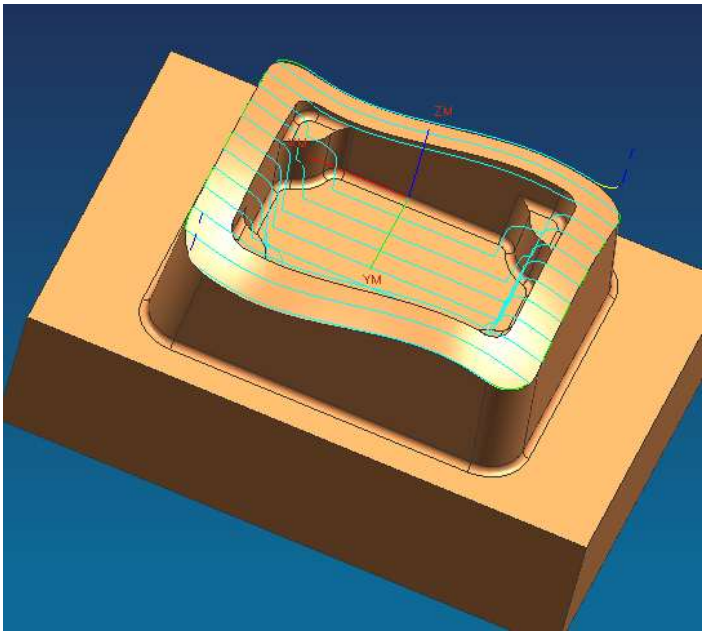


- Enter the parameters as shown in the figure on right
- Click **OK**

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**
- **Generate** the program

The contour surface is now finished and you can view the simulation by Tool Path verification.

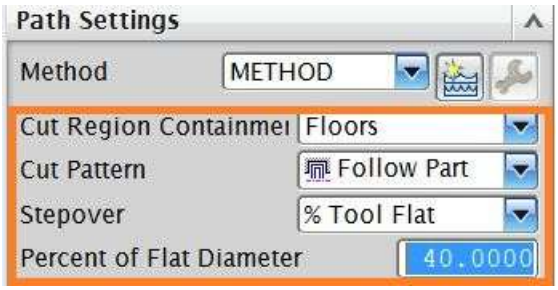


10.4.5 Flooring

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 the process. All the

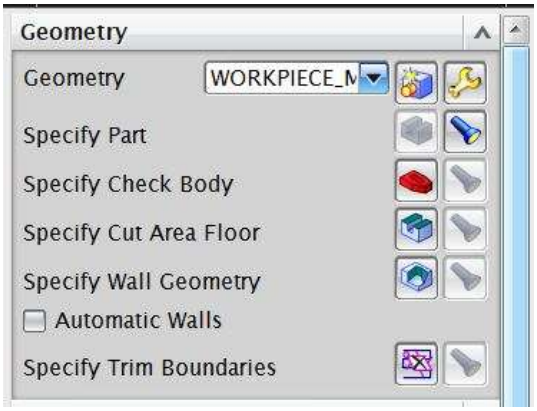
horizontal surfaces have to be finished. This planar operation runs the cutter in a single pass on every face.

- Click on the **Create Operation** icon on the Toolbar
- 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**



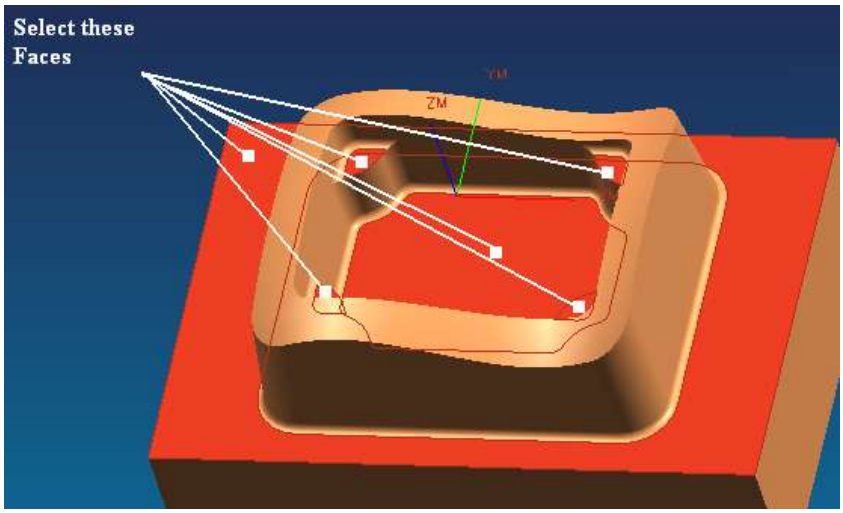
In flooring operations, it is always better to keep the Stepover value to be less than half of the diameter of the cutter in order to achieve more flatness on the planar surfaces.

Unlike previous programs, we have to select a cut area.

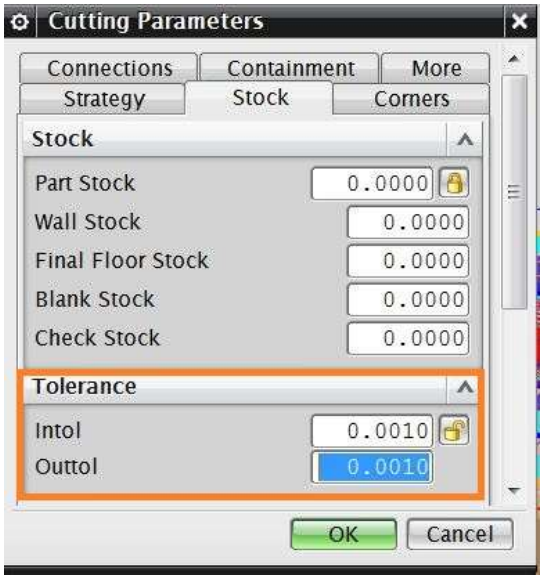


- Click on the **Specify Cut Area** tab
- Select the highlighted surfaces shown in the figure below.

In case you are not able to select the surfaces as shown go to **Part Navigator** and ***Hide*** the Blank, select the surfaces and ***Unhide*** the Blank again.



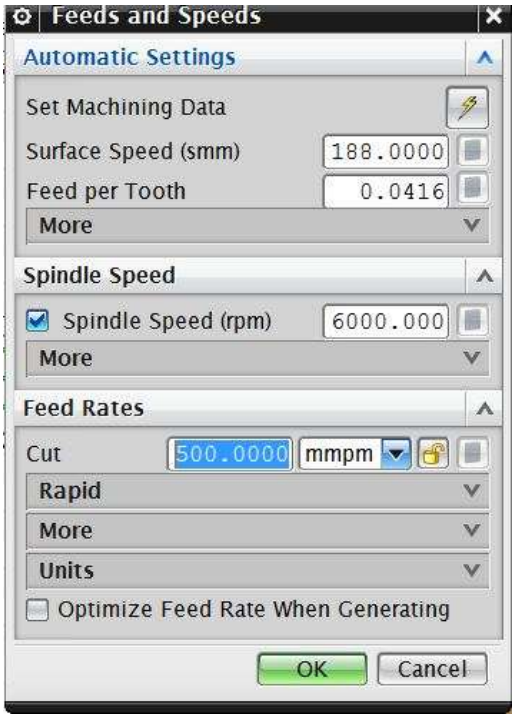
- Click **OK**
- Click on **CUTTING PARAMETERS** in the main parameter window
- Choose the **STOCK** tab button and enter the **Intol** and **Outtol** values as shown in the figure



- Click **OK**
- Click on **FEEDS AND SPEEDS**

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 exactly as shown in the figure

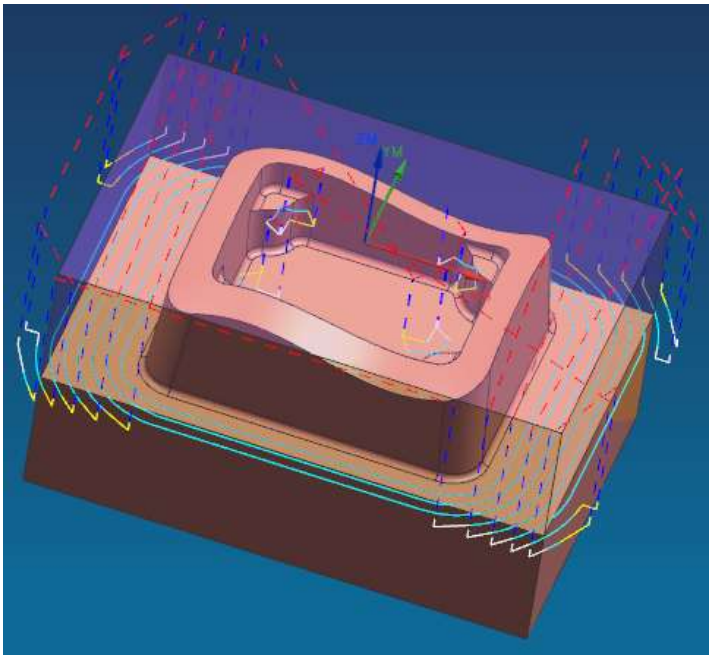


- Choose **OK**

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**
- Generate the program. Then replay and verify the cutter path

The following figure shows the ToolPath display for the flooring.



10.5 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 instance, 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 information is not in the initial NX9.0 tool path. Therefore, the tool path must be modified to suit the unique parameters of each different machine/controller combination. The modification is called post processing. The result is a post-processed tool path.

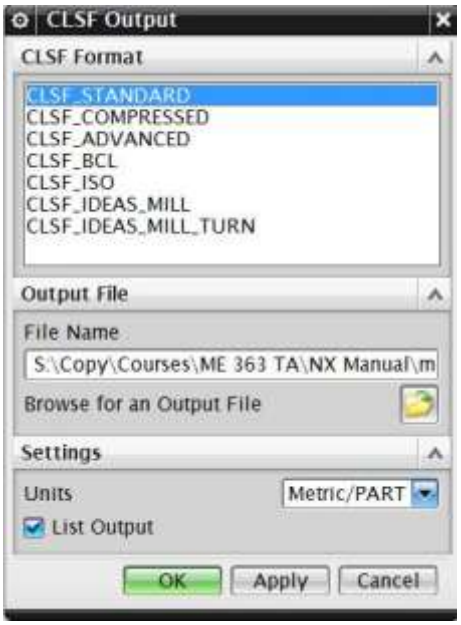
There are two steps involved in generating the final post-processed tool path.

1. Create the tool path data file, otherwise 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.

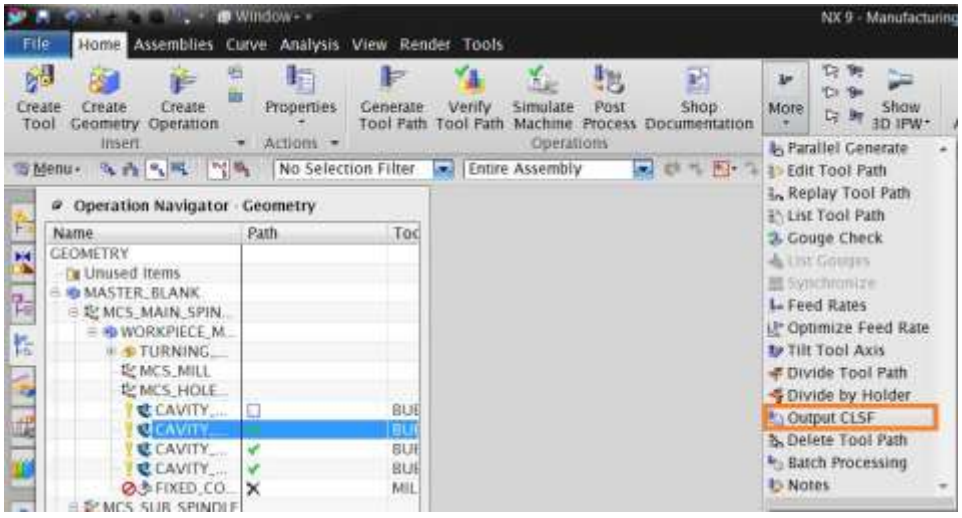
10.5.1 Creating CLSF

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.

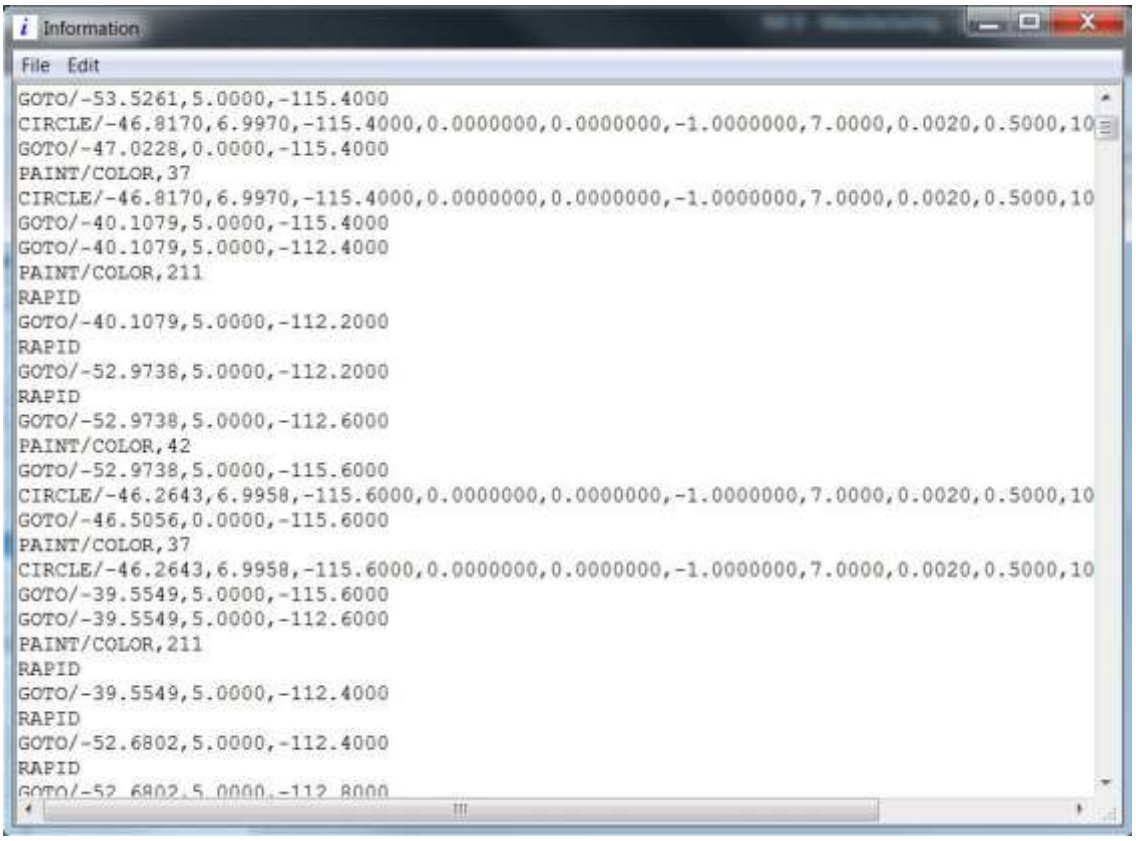


A window will pop up to select the CLSF Format.

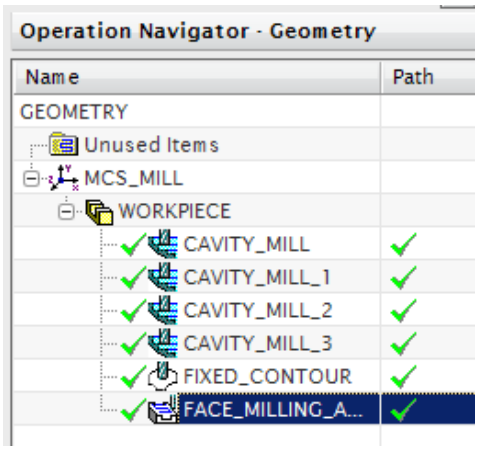
- Choose **CLSF_STANDARD** and enter a location for the file

➢ Choose **OK**

The CLSF file will be created. It will be similar to the figure below. The contents of the file contain the basic algorithm 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** (XXX.cls).



Any program that has been output to **CLSF** or post-processed will have a green checkmark next to it in the **Operation Navigator**.



10.5.2 Post-Processing

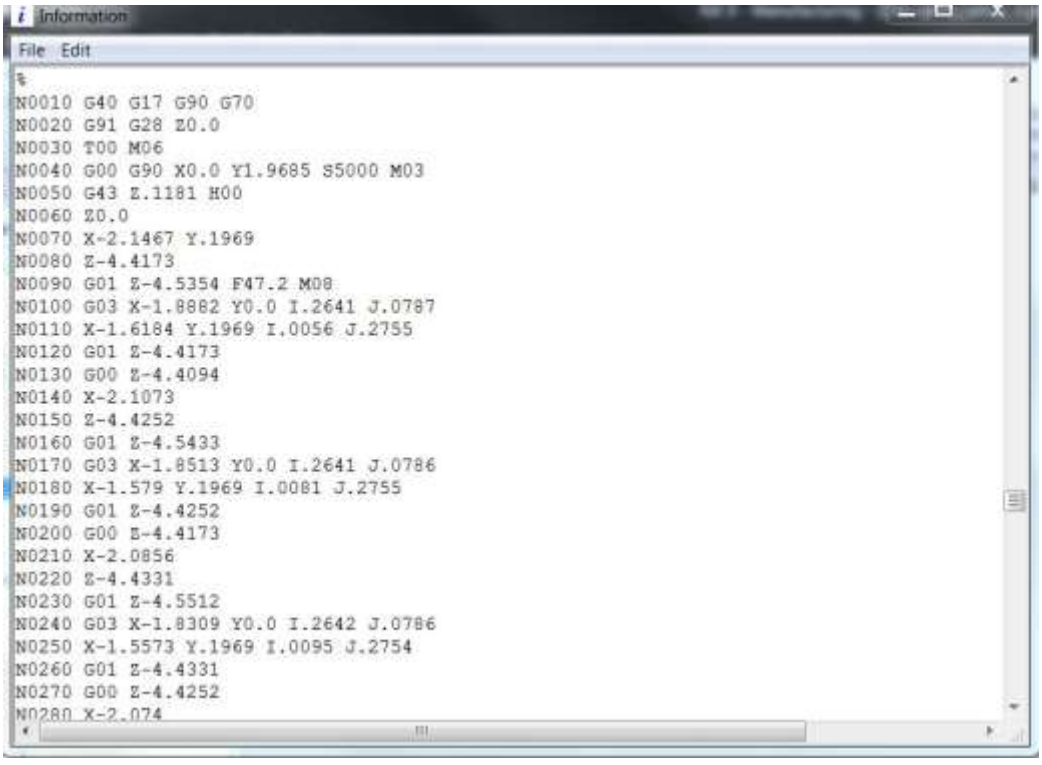
Click on a program in the **Operation Navigator** that you want to post process.

- Click **MENU** ⟶ **TOOLS → OPERATION NAVIGATOR → OUTPUT → POSTPROCESS** or from the **home tab** as shown below

- Select the **MILL_3_AXIS** machine and enter a location for the file

- Select **OK**

This will create the post-processed file for the desired machine. You can find the block numbers with G and M codes concerning the Machine controller type. The extension of the file is **.ptp** (XXX.ptp).



The final output (XXX.ptp) file can be transferred to the machine through DNC or punched tapes and the actual milling operation be done. This entire sequence starting from the transfer of the model into the Manufacturing module to the transfer of the files to the machine and cutting the raw piece into the final part is called Computer Aided Manufacturing.