



NX 10 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 solution. 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 NX 10. 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 an NX 10 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. The files of components are also available online to download and use. We first released the tutorial for Unigraphics 18 and later updated for NX 2 followed by the updates for NX 3, NX 5, NX 7 and NX 9. This write-up further updates to NX 10.

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 Amir Ghazanfari at ag4nc@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.

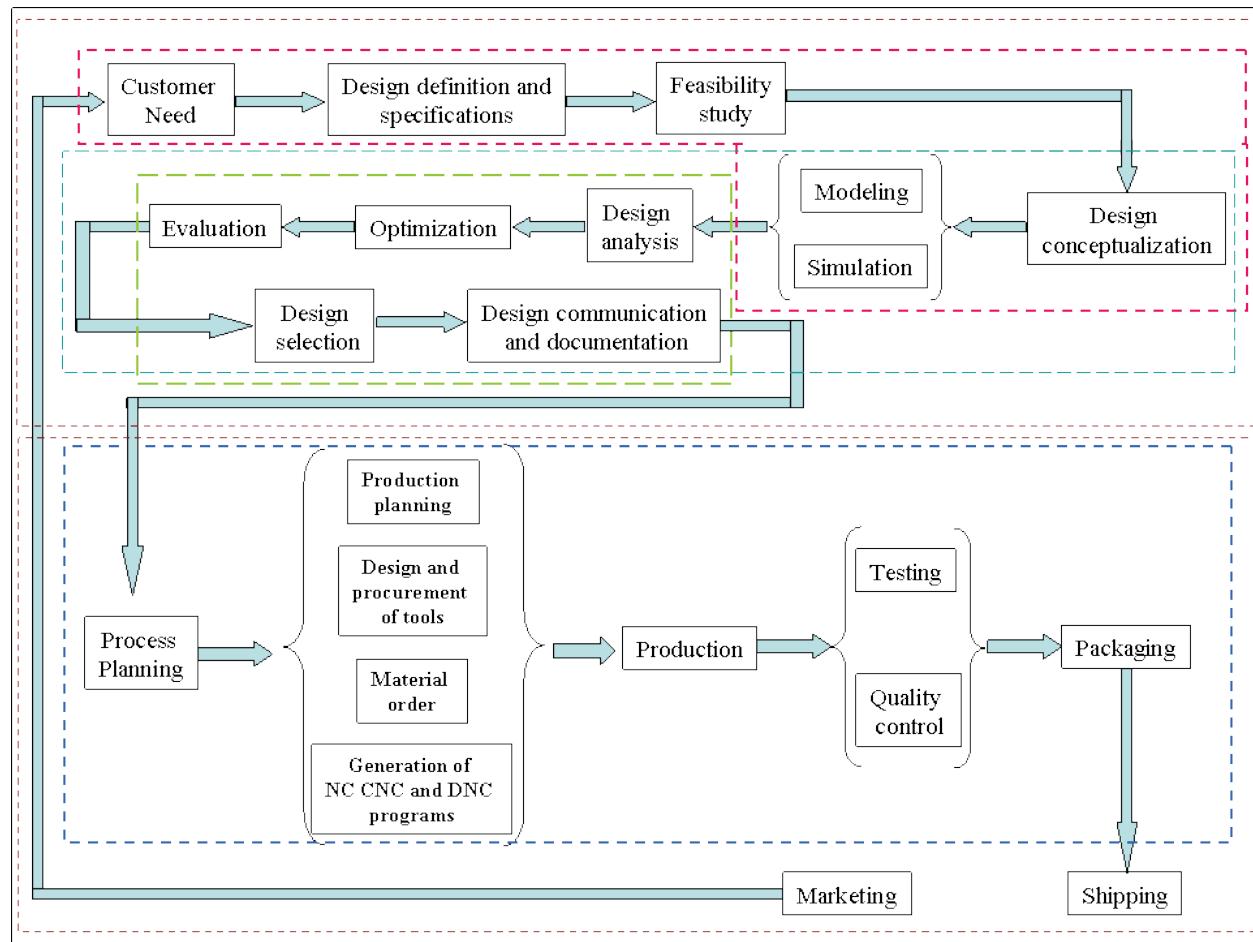
1.1 PRODUCT REALIZATION PROCESS

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 1800s. The real development of CAD/CAM systems started in the 1950s. CAD/CAM went through four major phases of development in the last century. The 1950s was known as the era of interactive computer graphics. MIT's Servo Mechanisms Laboratory demonstrated the concept of numerical control (NC) on a three-axis milling machine. Development in this era was slowed down by the shortcomings of computers at the time. During the late 1950s

the development of Automatically Programmed Tools (APT) began and General Motors explored the potential of interactive graphics.

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

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

In the 1980s, new theories and algorithms evolved and integration of various elements of design and manufacturing was developed. The major research and development focus 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.

- Solid Edge, 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, and COMSOL are packages mainly used for analysis of structures and fluids. Different software are used for different purposes.

- Geomagic and CollabCAD are some of the 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 NX 10 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 NX 10.

Chapter 2 includes the NX 10 essentials from starting a session to getting familiar with the NX 10 layout by practicing basic functions such as Print, Save, and Exit. It also gives a brief description of the Coordinate System, Layers, various toolboxes and other important commands, which will be used in later chapters.

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

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

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

Chapter 7 introduces free-form modeling. The method of modeling curves and smooth surfaces will be demonstrated.

Chapter 8 is capsulated into a brief introduction to Design Simulations available in NX 10 for the Finite Element Analysis.

Chapter 9 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 multiple axes and even advanced CNC machines.

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.

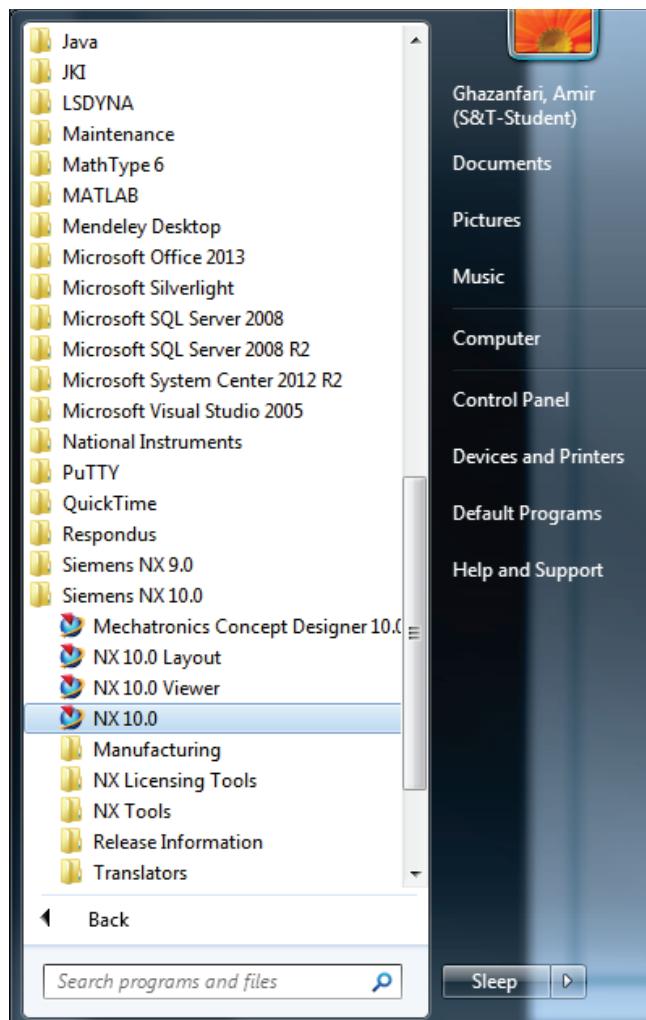
CHAPTER 2 – GETTING STARTED

We begin with starting of an NX 10 session. This chapter will provide the basics required to use any CAD/CAM package. You will learn the preliminary steps to start, to understand and to use the NX 10 package for modeling, drafting, etc. It contains five sub-sections a) Opening an NX 10 session, b) Printing, saving, and closing part files, c) getting acquainted with the NX 10 user interface d) Using layers and e) Understanding important commands and dialogs.

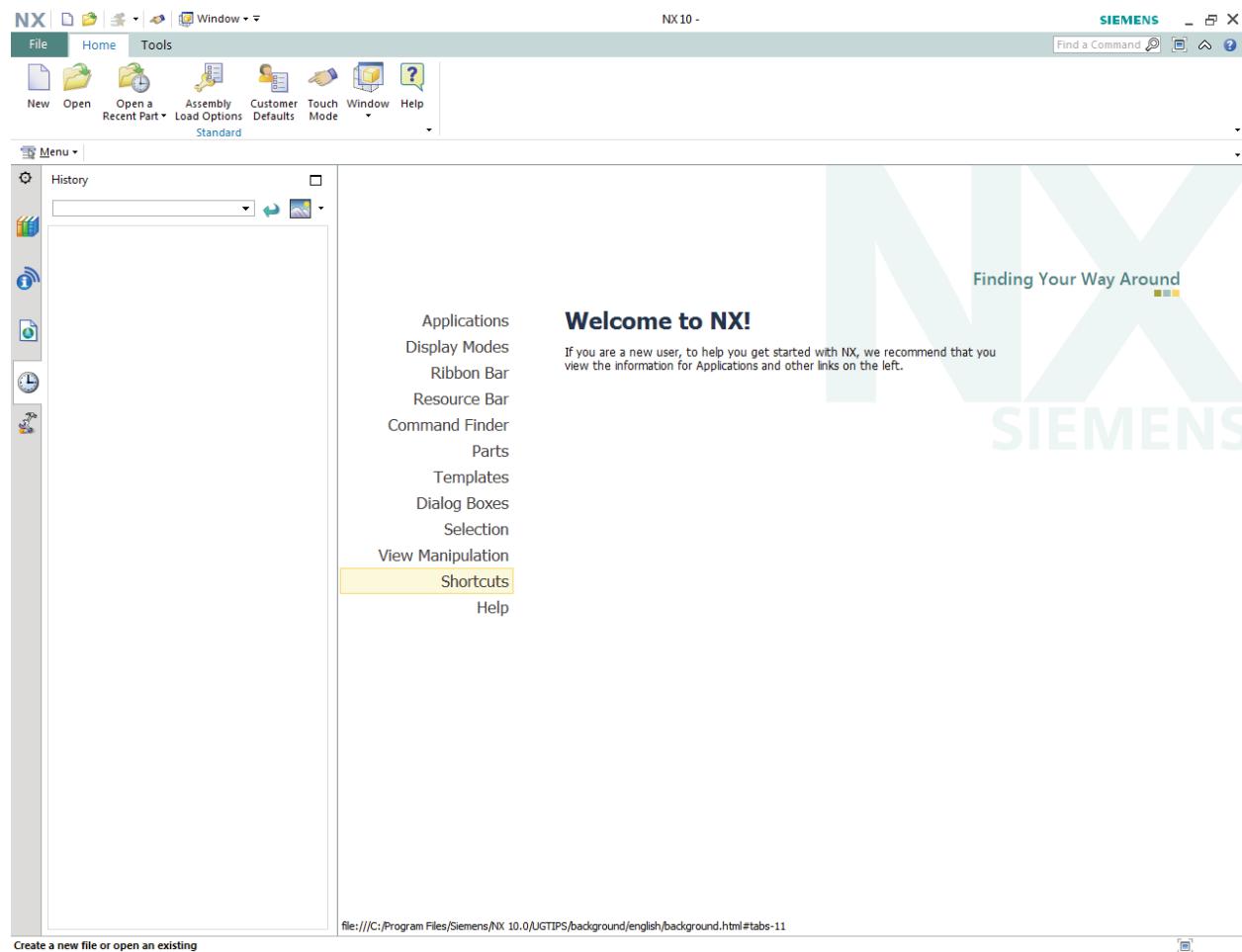
2.1 STARTING AN NX 10 SESSION AND OPENING FILES

2.1.1 Start an NX 10 Session

- From the Windows desktop screen, click on **Start** → **All Programs** → **Siemens NX 10** → **NX 10**



The main NX 10 Screen will open. This is the Gateway for the NX 10 software. The NX 10 blank screen looks like the figure shown below. There will be several tips displayed on the screen about the special features of the current version. The *Gateway* also has the *Standard Toolbar* that will allow you to create a new file or open an existing file. On the left side of the *Gateway* screen, there is a toolbar called the *Resource Bar* that has menus related to different modules and the ability to define and change the *Role* of the software, view *History* of the software use and so on. This will be explained in detail later in this chapter.



2.1.2 Open a New File

Let's begin by learning how to open a new part file in NX 10. To create a new file there are three options.

- Click on the **New** button on top of the screen

OR

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

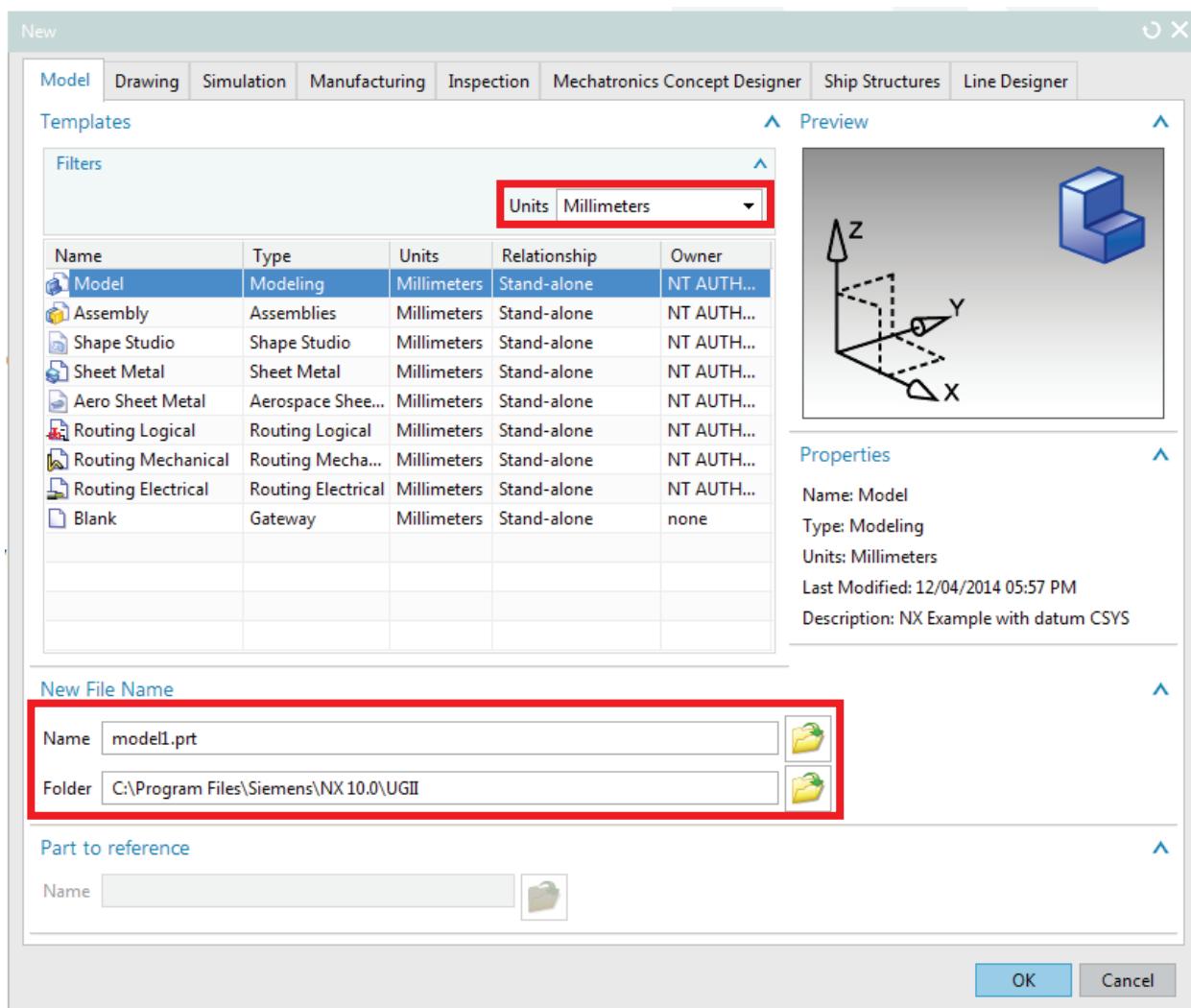
OR

- Press <Ctrl> + N

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

There are numerous types of files in NX 10 to select from the *Templates* dialogue box located at the center of the window. The properties of the selected file are displayed below the *Preview* on the right side. Since we want to work in the modeling environment and create new parts, only specify the units (inches or millimeters) of the working environment and the name and location of the file. The default unit is millimeters.

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



2.1.3 Open a Part File

There are several ways to open an existing file.

- Click on the **Open** or **Open a Recent Part** button on top of the screen

OR

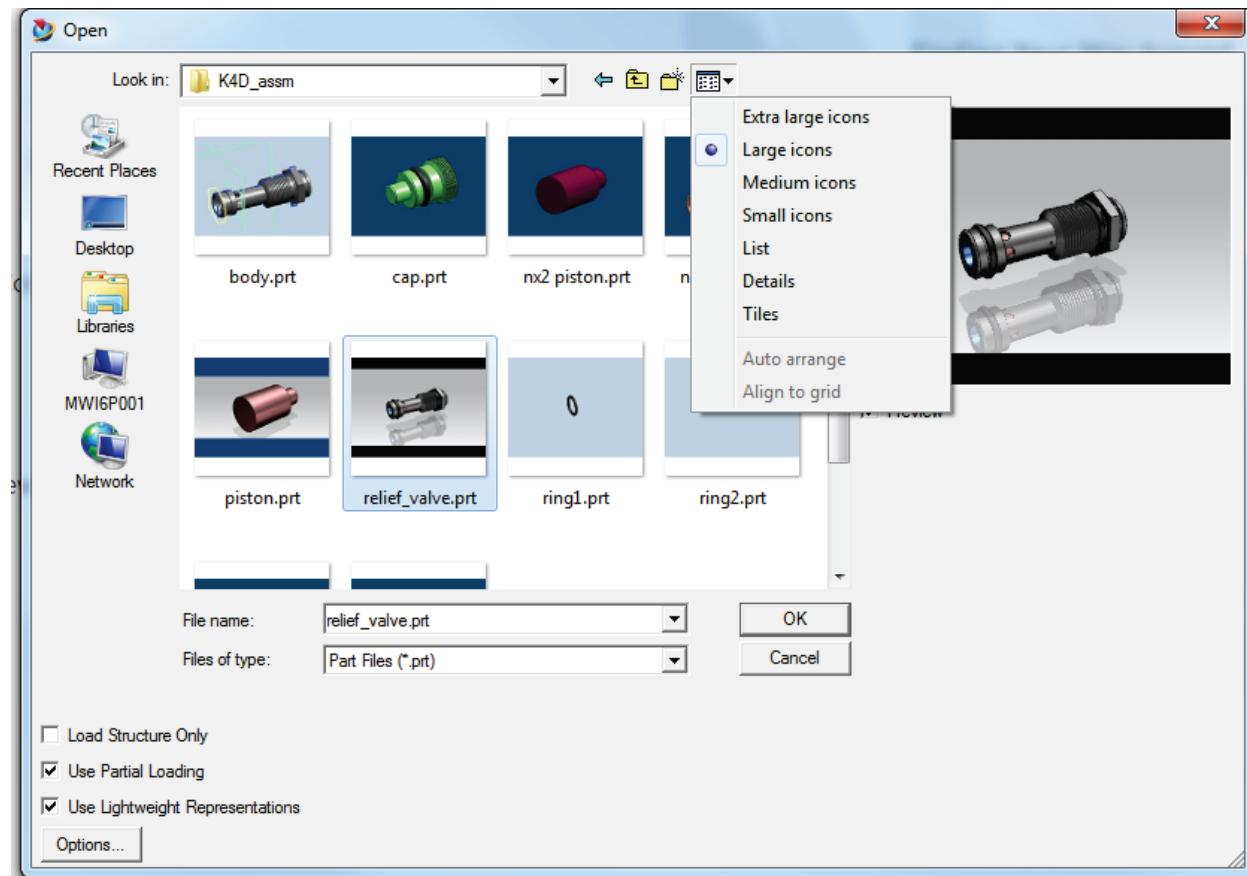
- Go through the **File** drop-down menu at the top-left of the screen and click **Open**

OR

- Press <Ctrl> + O

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

- Click **Cancel** to exit the window



2.2 PRINTING, SAVING AND CLOSING FILES

2.2.1 Print an NX 10 Image

To print an image from the current display,

- Click **File → Print**

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

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

- Click **Cancel** to exit the window

2.2.2 Save Part Files

It is imperative that you save your work frequently. If for some reasons, NX 10 shuts down and the part is not saved, all the work will be lost. To save the part files,

- Click **File → Save**

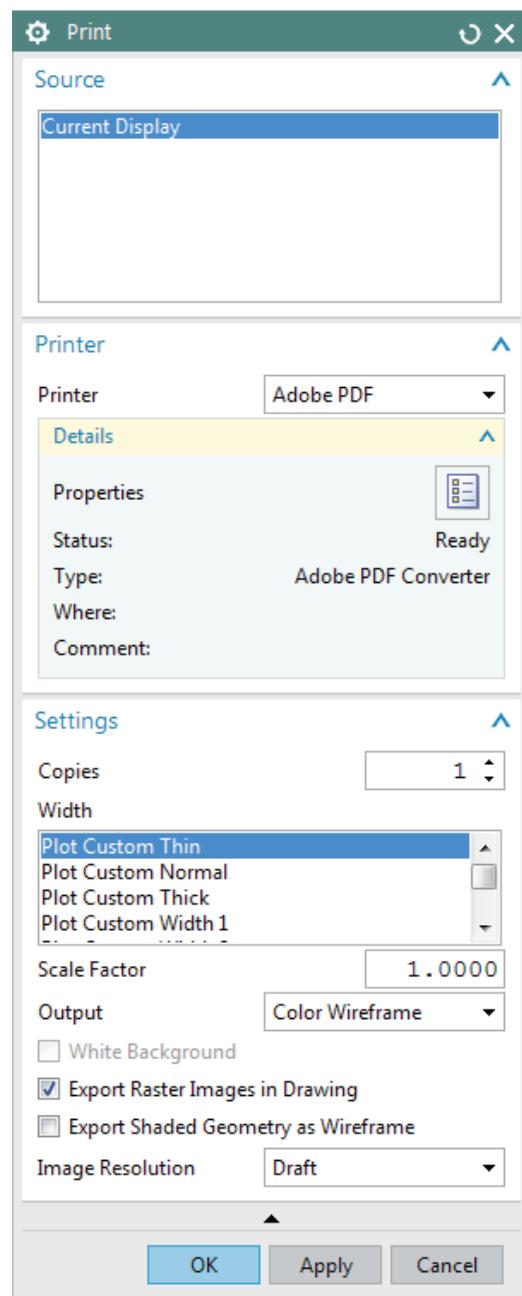
There are five options to save a file:

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

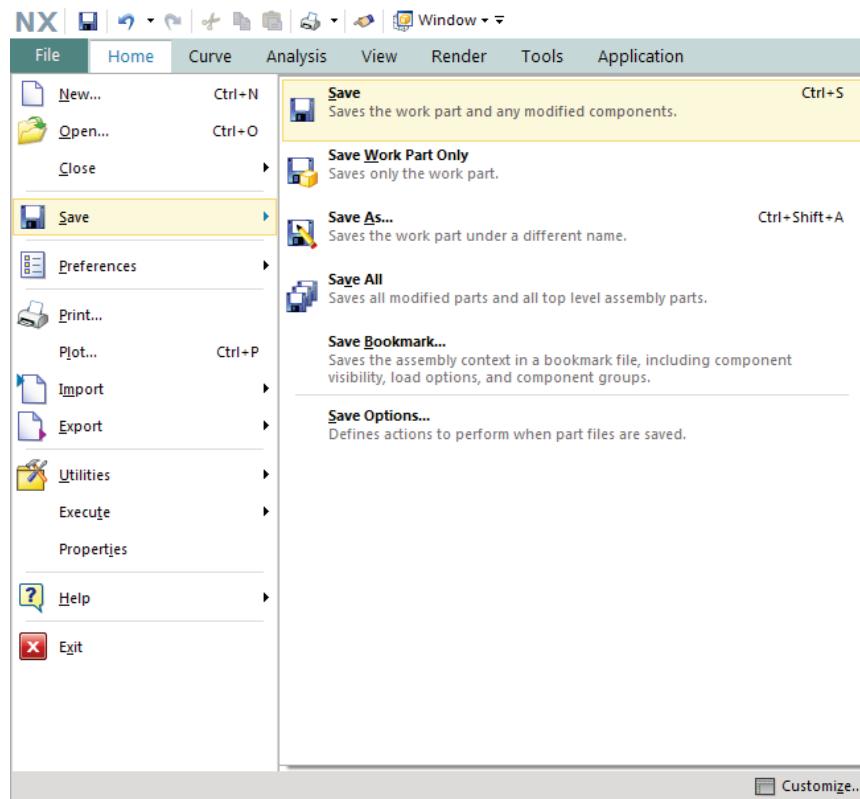
Save Work Part Only: This option will only save the active part on the screen.

Save As: This option allows you to save the part on screen using a different name and/or type. The default type is *.prt*. However, you can save your file as IGES (*.igs*), STEP 203 (*.stp*), STEP 214 (*.step*), AutoCAD DXF (*.dxf*), AutoCAD DWG (*.dwg*), CATIA Model (*.model*) and CATIA V5 (*.catpart*).

Save All: This option will save all the opened part files with their existing names.



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



2.2.3 Close Part Files

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

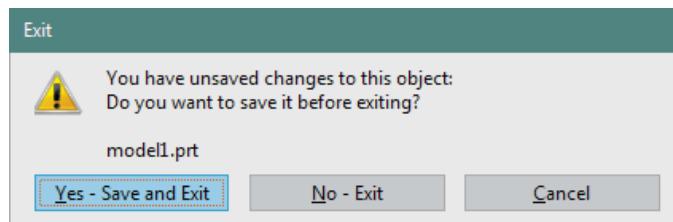
- Click File → Close

If you close a file, the file will be cleared from the working memory and any changes that are not saved will be lost. Therefore, remember to select *Save and Close*, *Save As and Close*, *Save All and Close* or *Save All and Exit*. In case of the first three options, the parts that are *selected* or *all parts* will be closed but the NX 10 session keeps on running.

Selected Parts...	Closes selected parts and keeps the session running.
All Parts	Closes all parts and keeps the session running.
Save and Close	Saves and closes the work part.
Save As and Close...	Saves the work part under a different name and closes it.
Save All and Close	Saves and closes all parts loaded in the current session.
Save All and Exit	Saves all modified parts and ends the session.
Close and Reopen Selected Parts...	Updates selected modified parts with the versions stored on disk.
Close and Reopen All Modified Parts	Updates all modified parts with the versions stored on disk.

2.2.4 Exit an NX 10 Session

- Click **File → Exit**



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

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

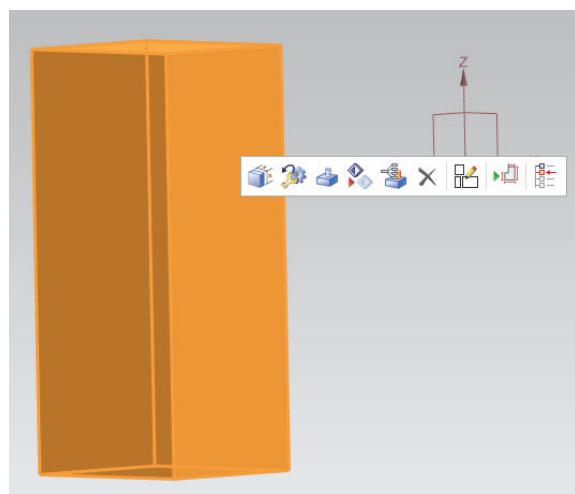
2.3 NX 10 INTERFACE

The user interface of NX 10 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 to entering values and naming files.

2.3.1 Mouse Functionality

2.3.1.1 Left Mouse Button (MB1)

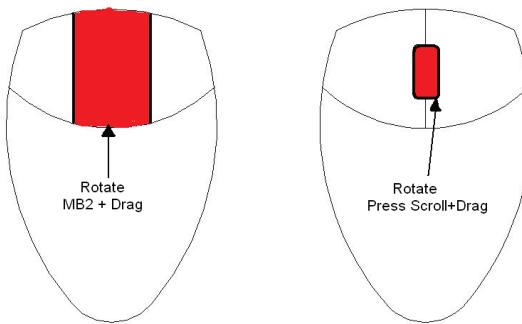
The left mouse button, named Mouse Button 1 (MB1) in NX, is used for *Selection* of icons, menus, and other entities on the graphic screen. Double clicking MB1 on any feature will automatically open the *Edit* Dialog box. Clicking MB1 on an object enables the user to have quick access to several options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



2.3.1.2 Middle Mouse Button (MB2)

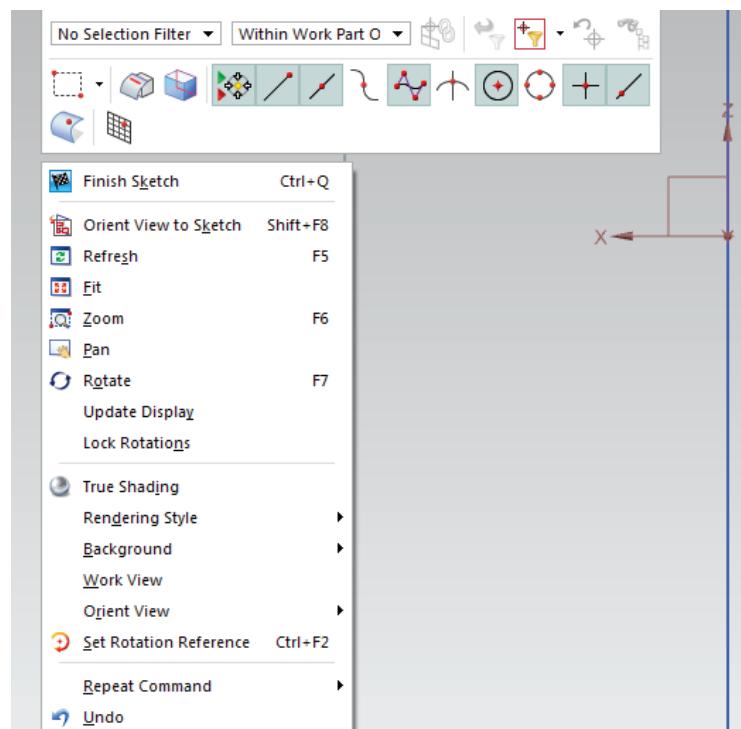
The middle mouse button (MB2) or the scroll button is used to *Rotate* the object by pressing, holding and dragging. The model can also be rotated about a single axis. To rotate about the axis horizontal to the screen, place the mouse pointer near the right edge of the graphic screen and rotate. Similarly,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

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

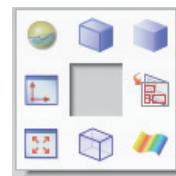


2.3.1.3 Right Mouse Button (MB3)

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



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



2.3.1.4 Combination of Buttons

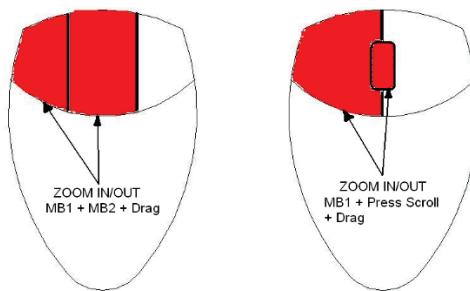
Zoom In /Out:

- Press and hold both **MB1** and **MB2** simultaneously and drag

OR

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

OR

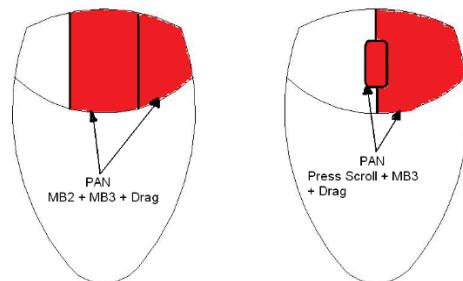


Pan:

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

OR

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

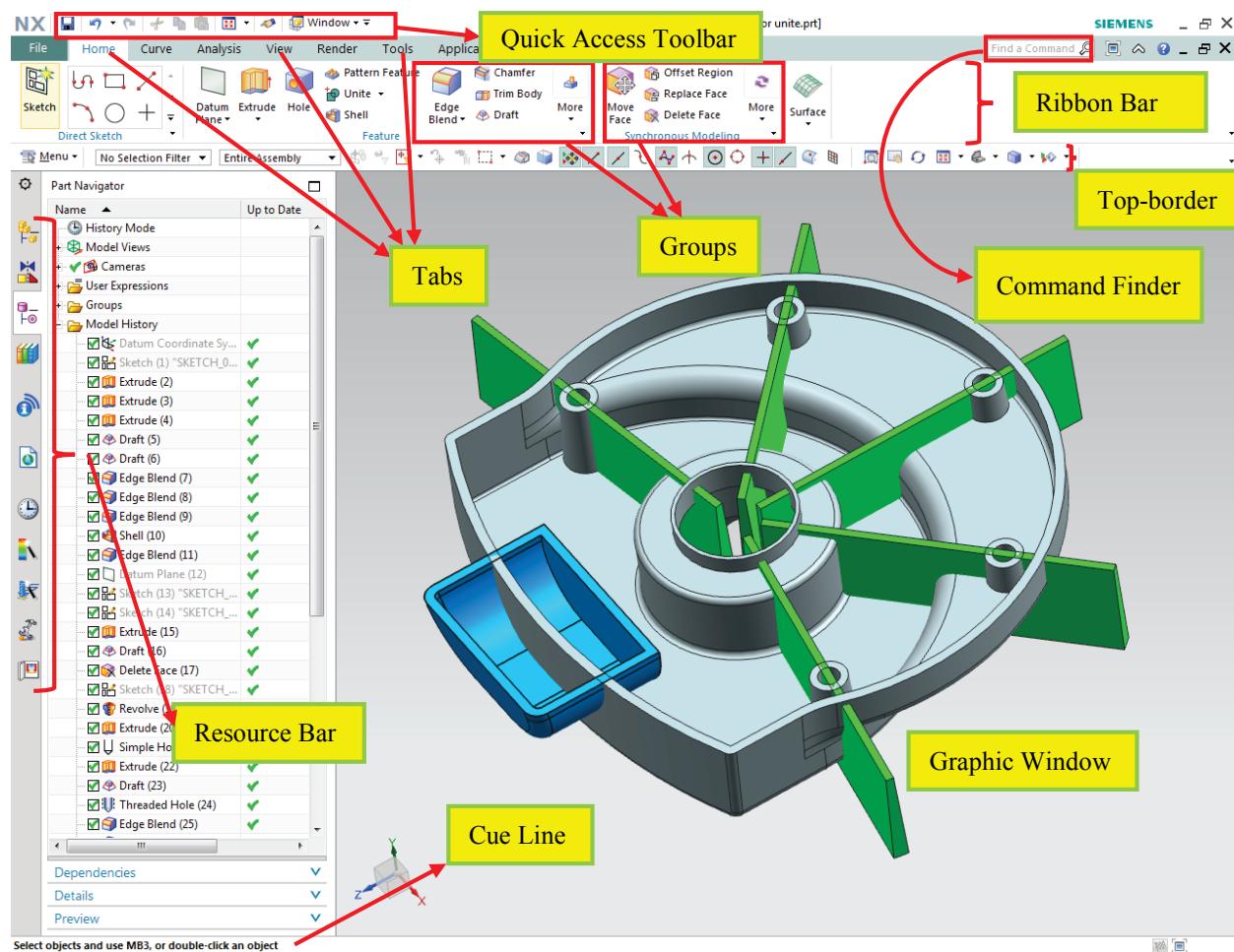


Shortcut to menus:

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

2.3.2 NX 10 Gateway

The following figure shows the typical layout of the NX 10 window when a file is opened. This is the Gateway of NX 10 from where you can select any module to work on such as modeling, manufacturing, etc. It has to be noted that these toolbars may not be exactly on the same position of the screen as shown below. The toolbars can be placed at any location or position on the screen. Look out for the same set of icons.



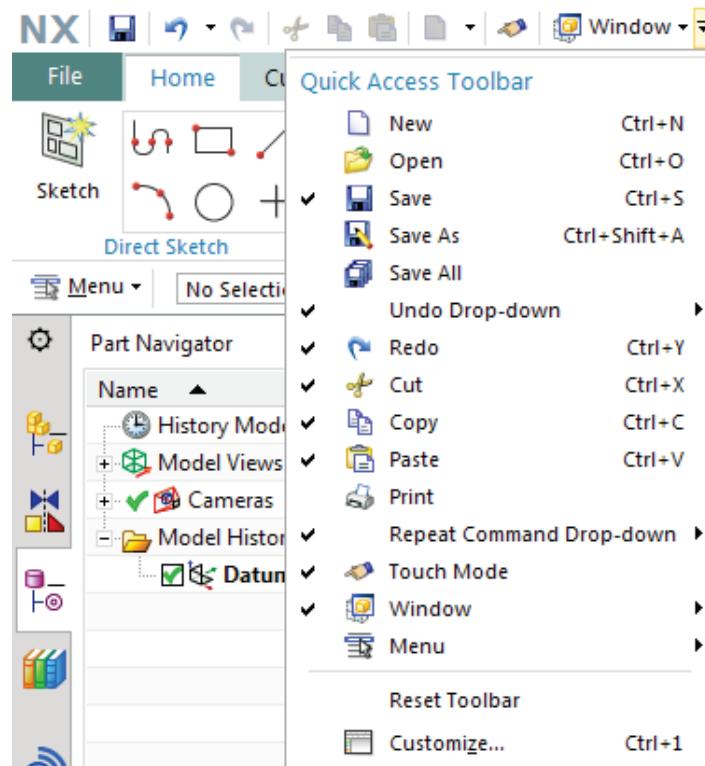
2.3.2.1 Ribbon Bar

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

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

2.3.2.2 Quick Access Toolbar

The quick access toolbar has most commonly used buttons (save, undo, redo, cut, copy, paste and recent commands) to expedite the modeling process. You may easily customize these buttons as shown in the figure below.



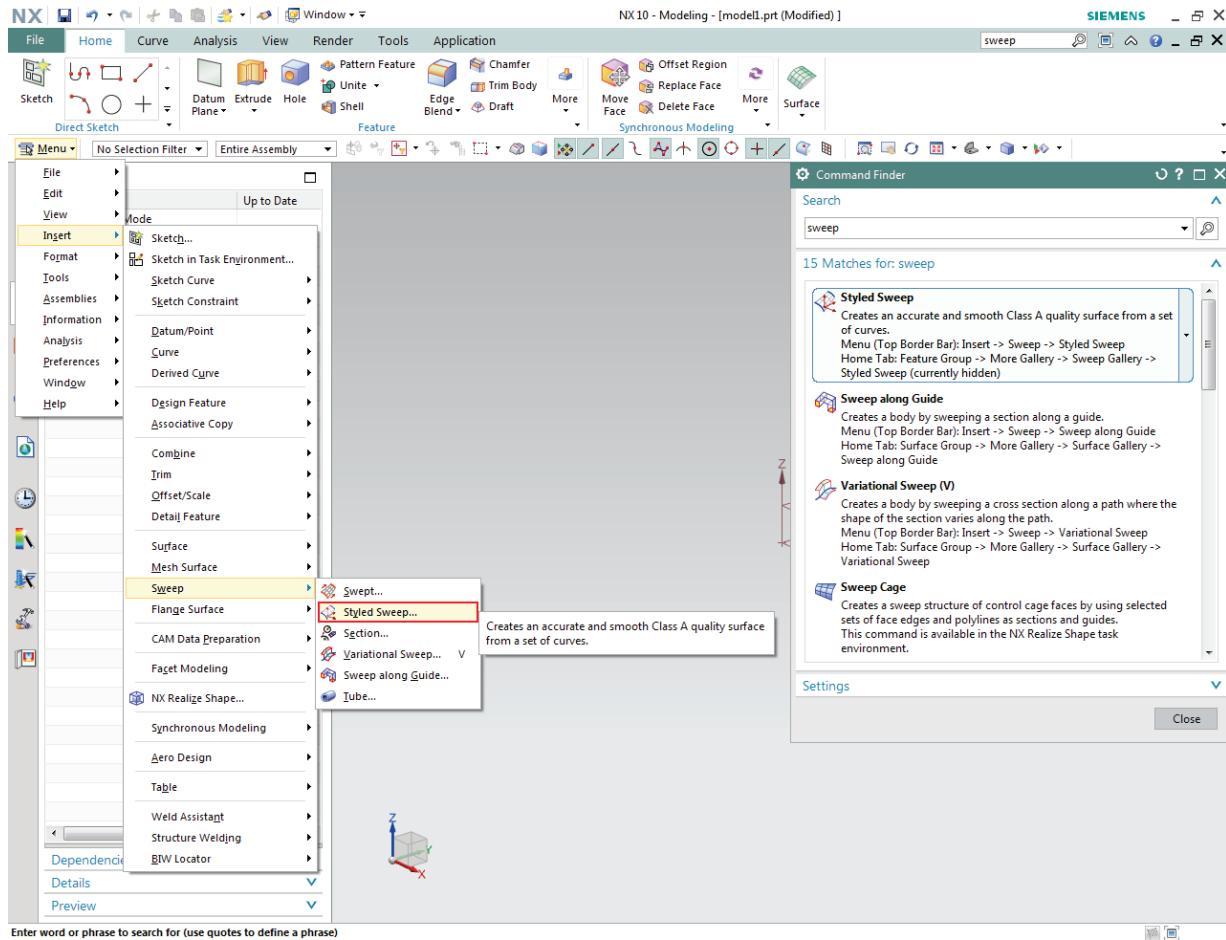
2.3.2.3 Command Finder

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

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

OR

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



2.3.2.4 Top-border

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

2.3.2.5 Resource Bar

The *Resource Bar* features icons for a number of pages in one place using very little user interface space. NX 10 places all navigator windows (*Assembly*, *Constraint* and *Part*) in the *Resource Bar*, as well as the *Reuse Library*, *HD3D Tools*, *Web Browser*, *History Palette*, *Process Studio*,

Manufacturing Wizards, Roles and System Scenes. Two of the most important widows are explained below.

Part Navigator

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

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

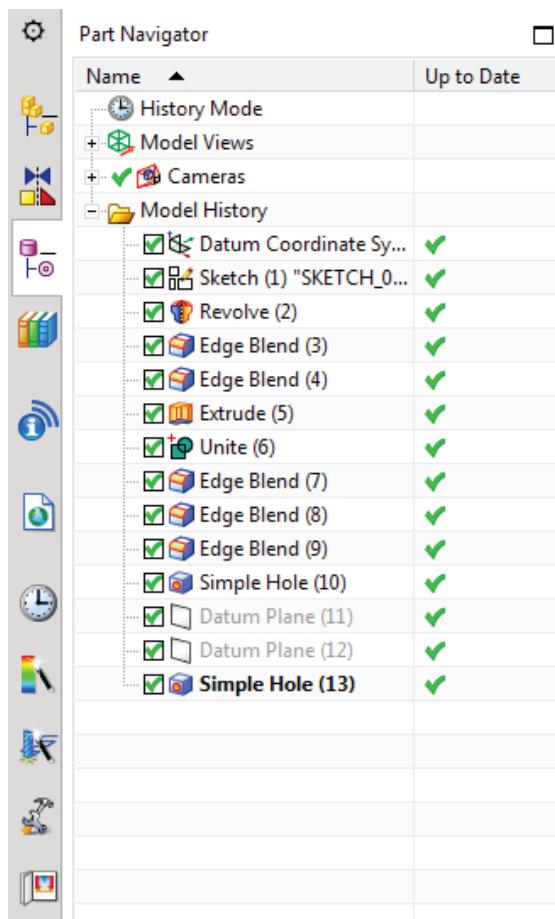
The *Part Navigator* is available for all NX applications and not just for modeling. However, you can only perform feature-editing operations when you are in the Modeling module. Editing a feature in the *Part Navigator* will automatically update the model. Feature editing will be discussed later.

History

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

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.

2.3.2.6 Cue Line

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

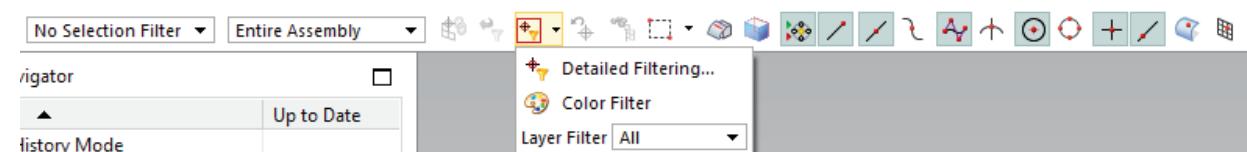
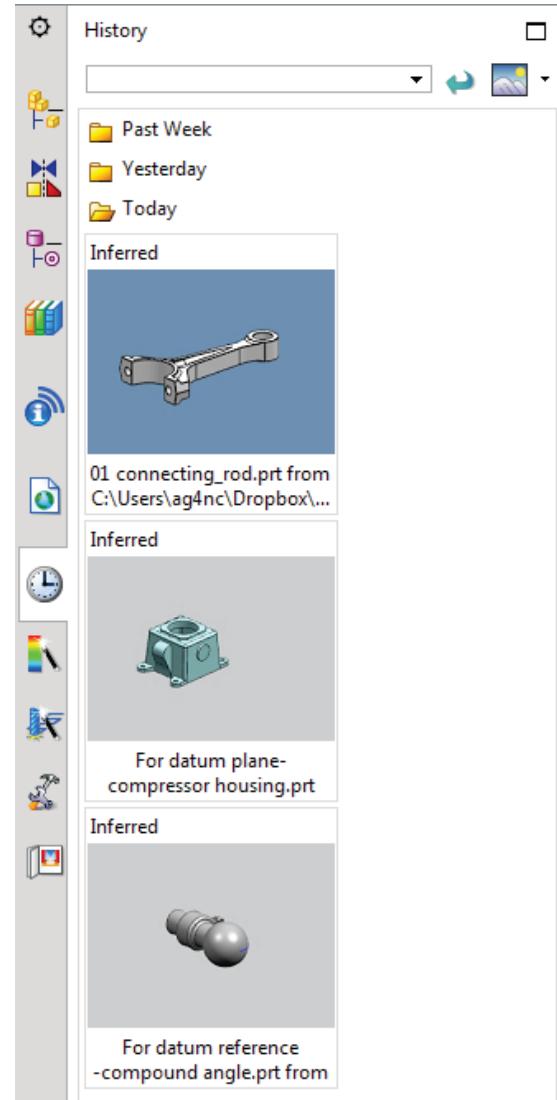
The *Progress Meter* is displayed in the *Cue Line* when the system performs a time-consuming operation such as loading a large assembly. The meter shows the percentage of the operation that has been completed. When the operation is finished, the system displays the next appropriate cue.

2.3.3 Geometry Selection

You can filter the selection method, which facilitates easy selection of the geometry in a close cluster. In addition, you can perform any of the feature operation options that NX 10 intelligently provides depending on the selected entity. Selection of items can be based on the degree of the entity like, selection of *Geometric* entities, *Features* and *Components*. The selection method can be opted by choosing one of the icons in the *Selection Toolbar*.

2.3.3.1 Feature Selection

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

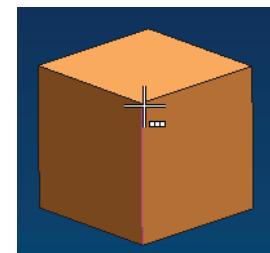
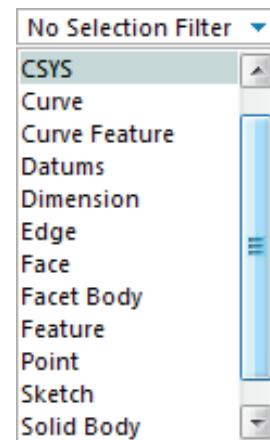


Besides that, the filtering of the features can be further narrowed down by selecting one of the desired options in the drop-down menu as shown in the figure. For example, selecting *Curve* will highlight only the curves in the screen. The default is *No Selection Filter*.

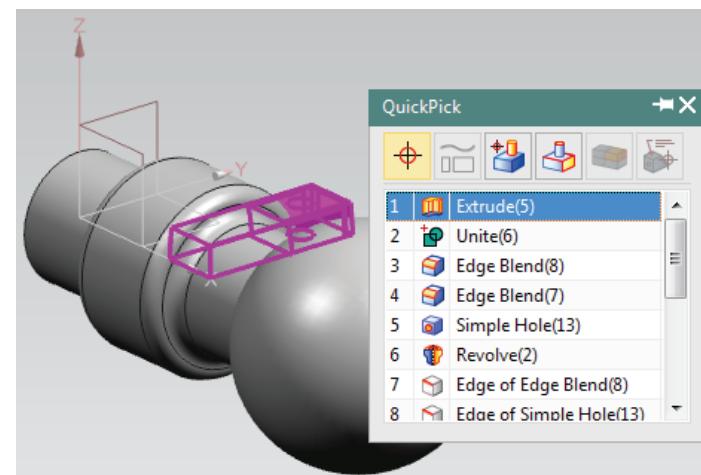
2.3.3.2 General Object Selection

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

If you want to select an entity that is hidden behind the displayed geometry, place the mouse cursor roughly close to that area on the screen such that the cursor ball occupies a portion of the hidden geometry projected on the screen. After a couple of seconds, the ball cursor turns into a *plus* symbol as shown in the 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, NX 10 will highlight the corresponding entity on the screen in a magenta color.



Confirmation dialog box as shown in the following figure below. This *QuickPick* menu consists of the list of entities captured within the ball of the cursor. The entities are arranged in ascending order of the *degree* of the entity. For example, edges and vertices are assigned lower numbers while solid faces are given higher numbers. By moving the cursor on the numbers displayed, NX 10 will highlight the corresponding entity on the screen in a magenta color.



2.3.4 User Preferences

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

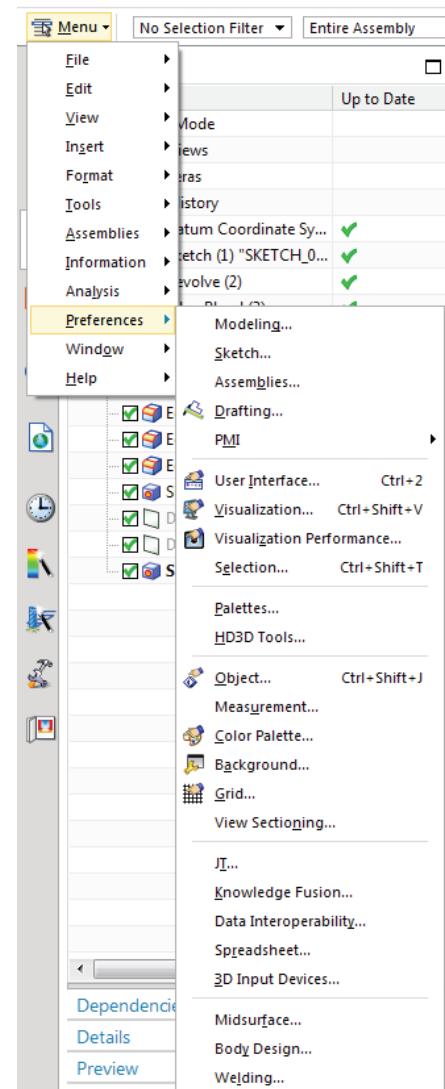
User Preferences are used to define the display parameters of new objects, names, layouts, and views. You can set the layer, color, font, and width of created objects. You can also design layouts and views, control the display of object and view names and borders, change the size of the selection ball,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

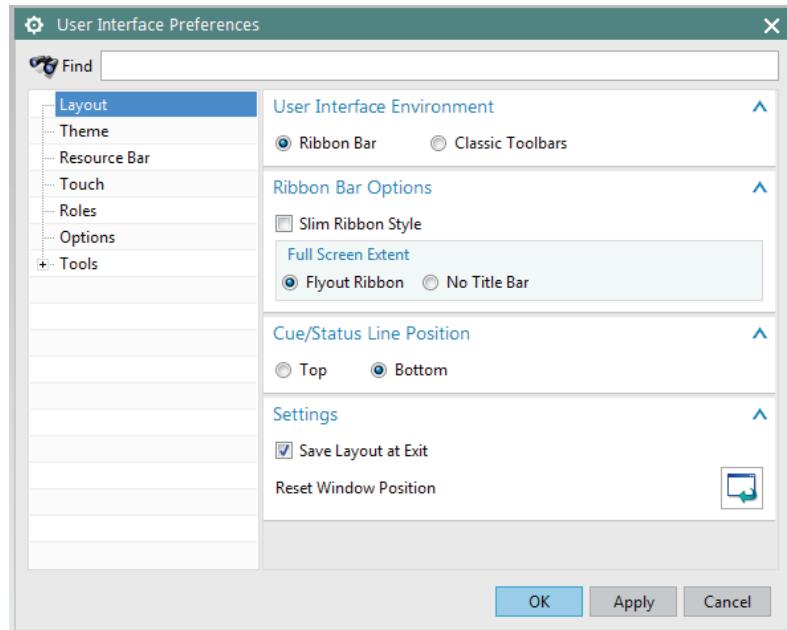
2.3.4.1 User Interface

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

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

- The *Layout* tab allows you to select the *User Interface Environment*
- The *Touch* tab lets you use touch screens
- The *Options* tab allows you, among others, to set the precision level (in the *Information Window*)
- The *Journal* tab in the *Tools* allows you to use several programming languages
- The *Macro* tab in the *Tools* allows you to set the pause while displaying animation



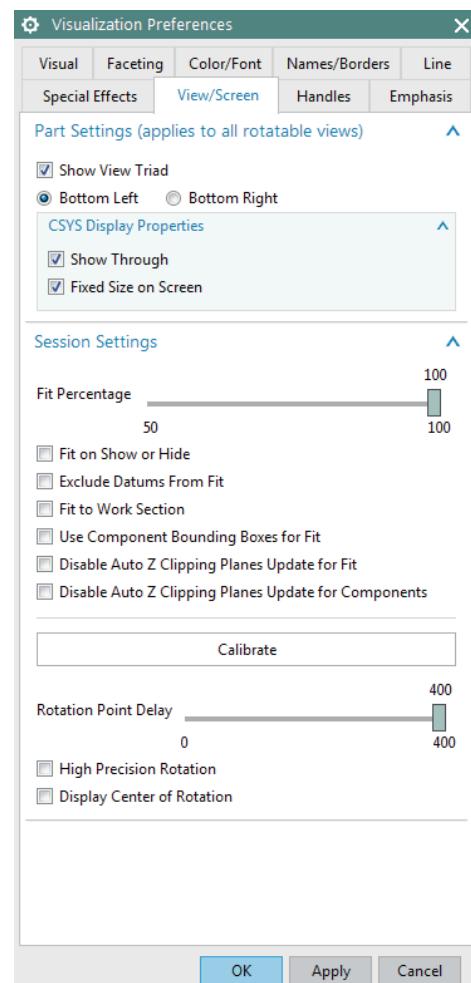


2.3.4.2 Visualization

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

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

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



- Click on **Preferences → 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.

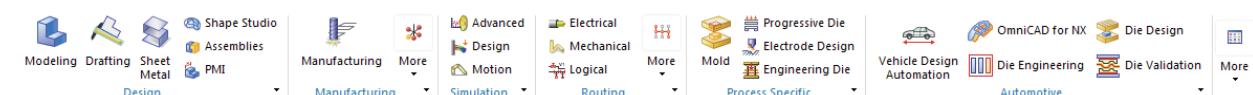
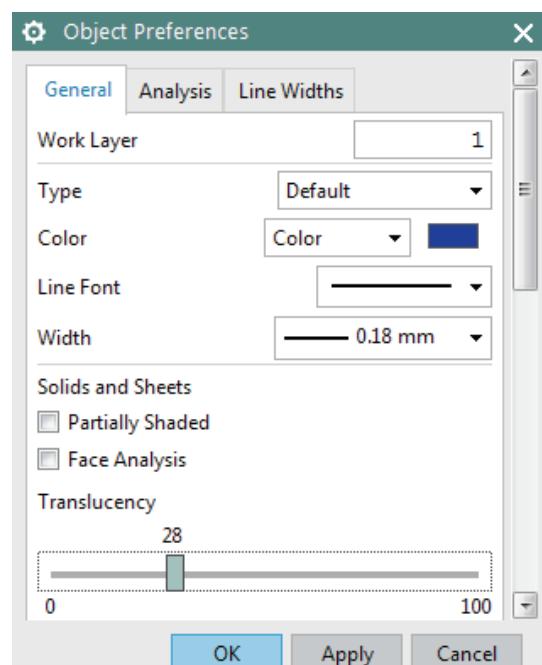
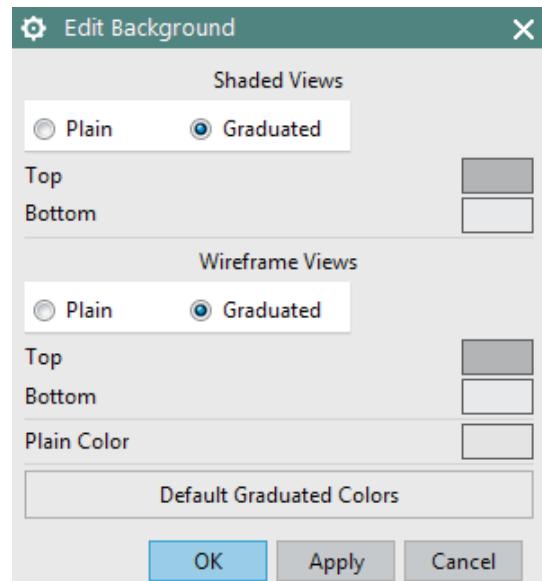
You can change and observe the *Color* and *Translucency* of objects.

- Click **Preferences → Object**

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

2.3.5 Applications

Applications can be opened using the *File* option located at the top left corner of the main window OR the *Applications* tab above the *Ribbon bar*. You can select the type of application you want to run. For example, you can select *Modeling*, *Drafting*, *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*. We will introduce some of these *Application* in the next chapters.



2.4 LAYERS

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

2.4.1 Layer Control

With NX 10, you can control whether objects are visible or selectable by using *Layers*. A *Layer* is a system-defined attribute such as color, font, and width that all objects in NX 10 must have. There are 256 usable layers in NX 10, 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*

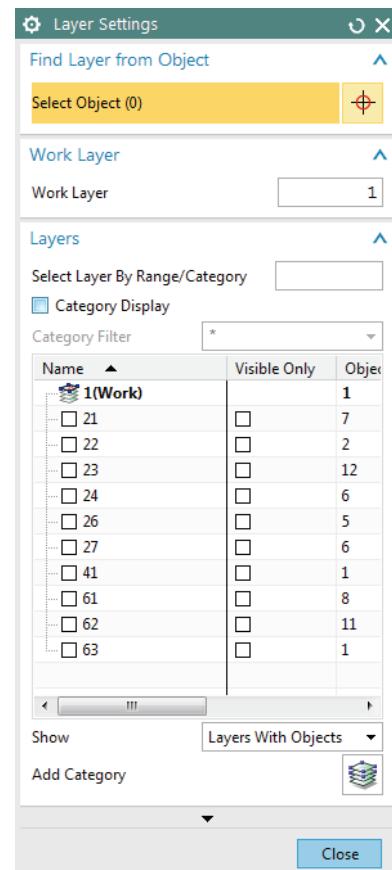
The *Work Layer* is the layer that objects are created ON and is always visible and selectable while it remains the *Work Layer*. *Layer 1* is the default *Work Layer* when starting a new part file. When the *Work Layer* is changed to another type of layer, the previous *Work Layer* automatically becomes *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 **View →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.



2.4.2 Commands in Layers

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

- Choose **File → New**

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

- Choose **Menu → Insert → Design Feature → Cone**
- Choose **Diameter and Height** under Type
- Click **OK**
- Right-click on the screen and choose **Orient View → Trimetric**
- Right-click on the screen and choose **Rendering Style → Shaded**

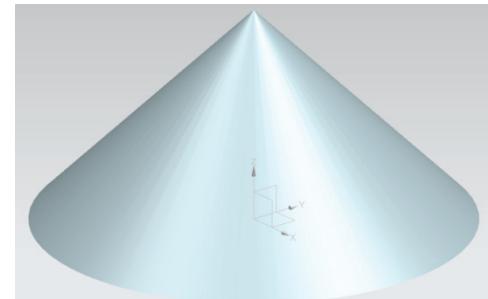
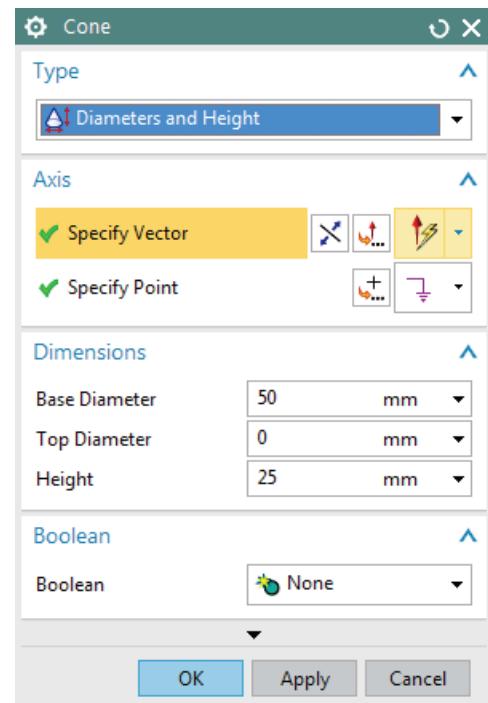
You will be able to see a solid cone similar to the picture on right.

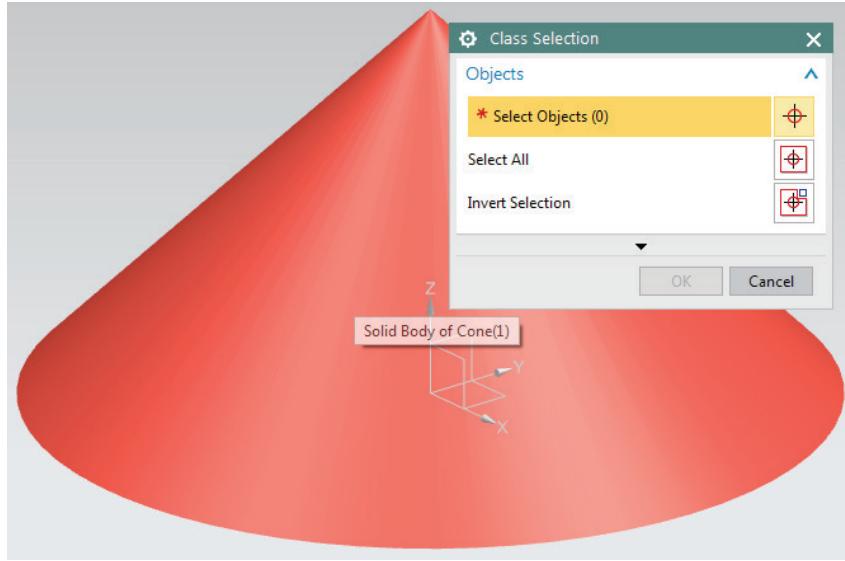
Now let us practice some *Layer Commands*.

- Choose **View → Move to Layer**

You will be asked to select an object

- Move the cursor on to the *Cone* and click on it so that it becomes highlighted
- Click **OK**



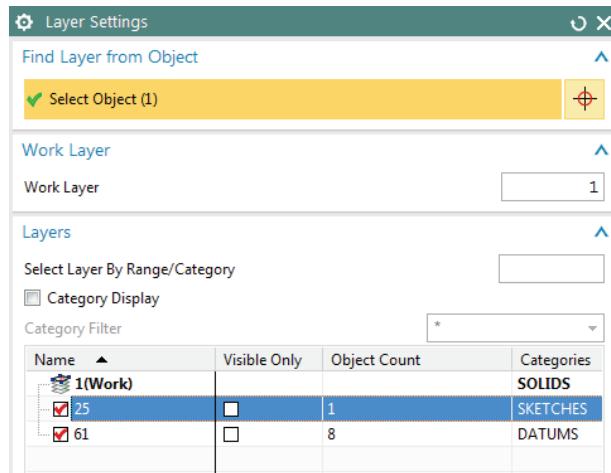
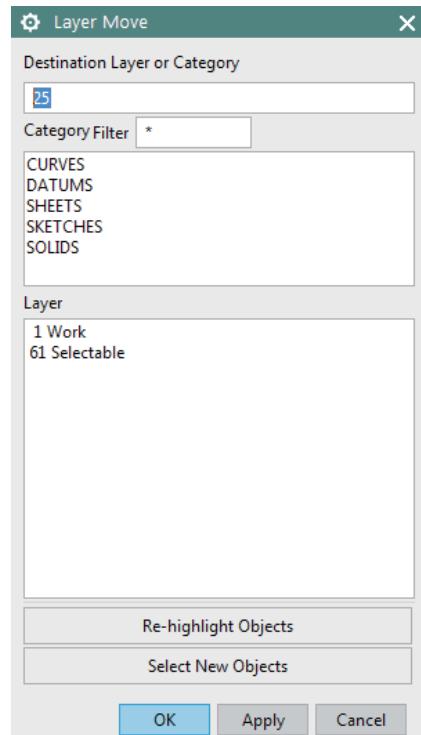


- In the **Destination Layer or Category** space at the top of the window, type 25 and Click **OK**

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

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

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

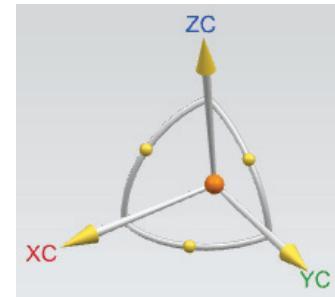


2.5 COORDINATE SYSTEMS

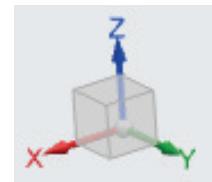
There are different coordinate systems in NX. A three-axis symbol is used to identify the coordinate system.

2.5.1 Absolute Coordinate System

The *Absolute Coordinate System* is the coordinate system from which all objects are referenced. This is a fixed coordinate system and the locations and orientations of every object in NX 10 modeling space are related back to this system. The *Absolute Coordinate System* (or Absolute CSYS) also provides a common frame of reference between part files. An absolute position at $X=1$, $Y=1$, and $Z=1$ in one part file is the same location in any other part file.

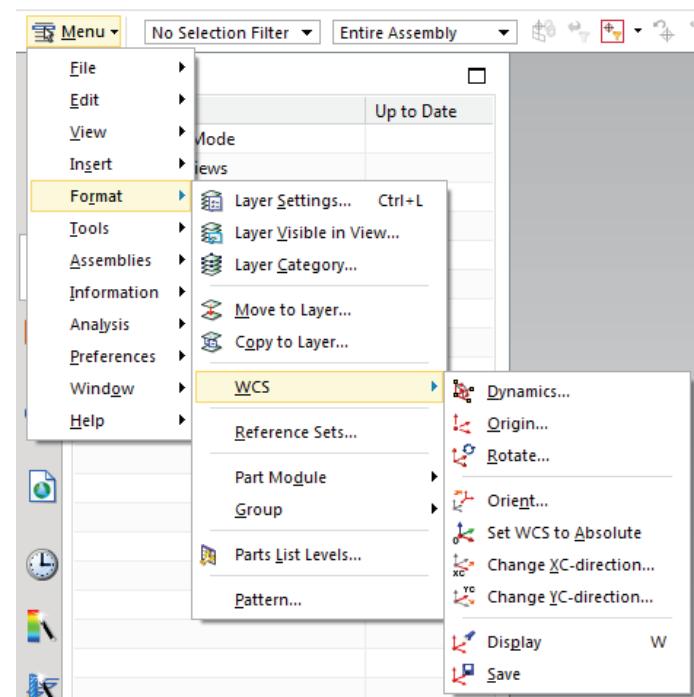


The *View Triad* on the bottom-left of the *Graphics* window is ONLY a visual indicator that represents the ORIENTATION of the *Absolute Coordinate System* of the model.



2.5.2 Work Coordinate System

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



2.5.3 Moving the WCS

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

- Choose **Menu → Format → WCS**

2.5.3.1 Translate the WCS

This procedure will move the WCS origin to any point you specify, but the

orientation (direction of the axes) of the WCS will remain the same.

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

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

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

2.5.3.2 Rotate the WCS

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

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

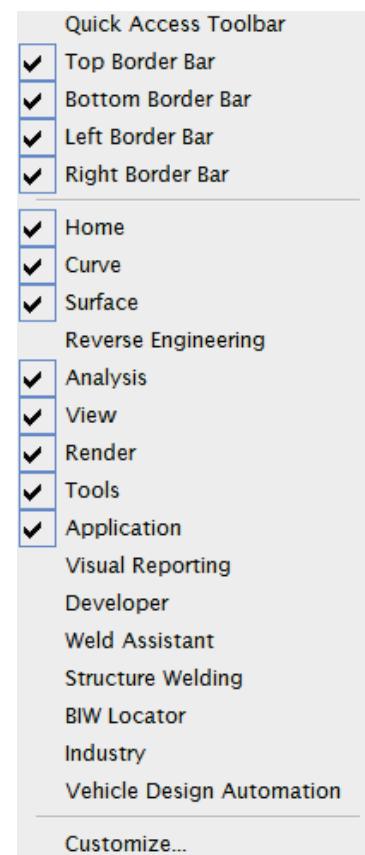
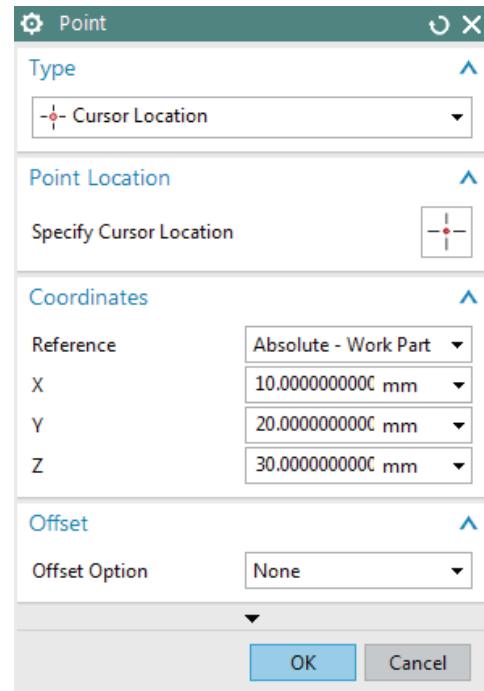
The dialog shows six different ways to rotate the WCS around an axis. These rotation procedures follow the right-hand rule of rotation. You can also specify the angle to which the WCS be rotated.

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

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

2.6 TOOLBARS

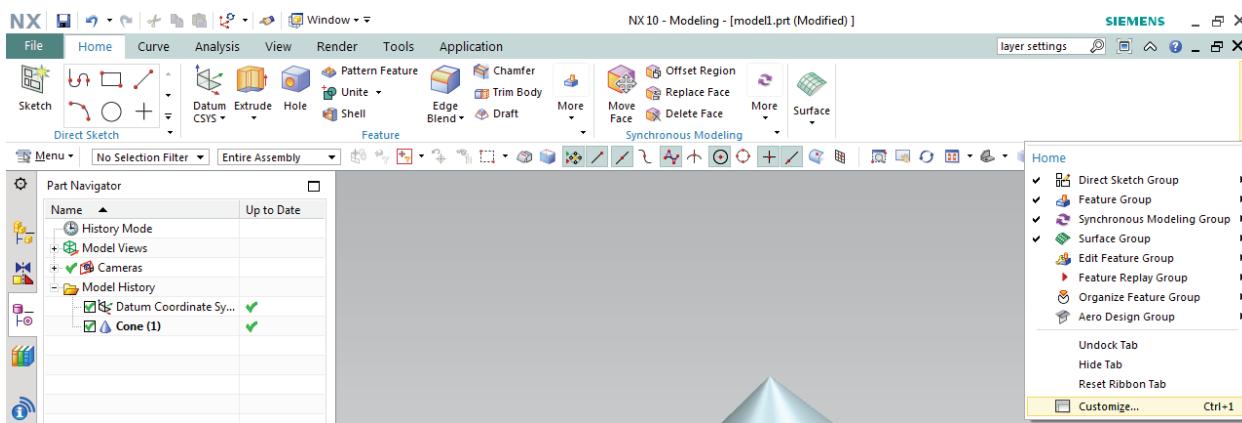
Toolbars contain icons, which serve as shortcuts for many functions. The figure on the right shows the main Toolbar items normally displayed. However, you can find many more icons for different feature commands, based on the module selected and how the module is customized.



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

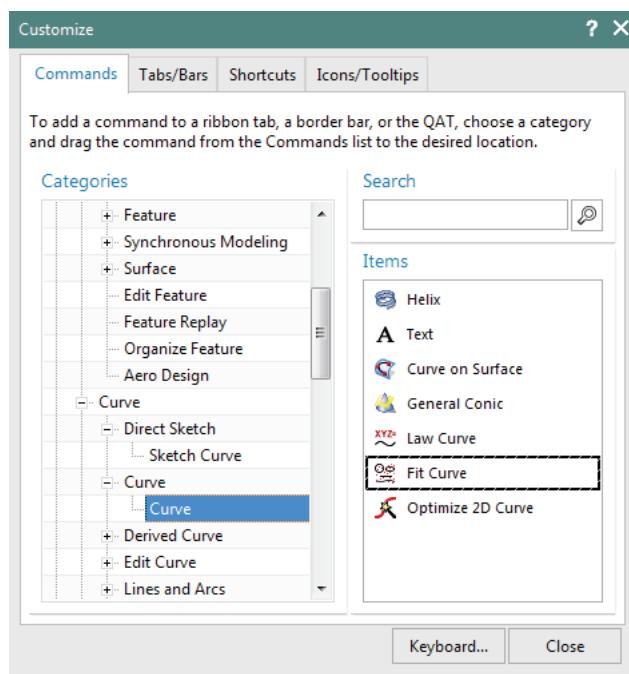
Normally, the default setting should be sufficient for most operations but during certain operations, you might need additional toolbars. If you want to add buttons pertaining to the commands and toolbars,

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



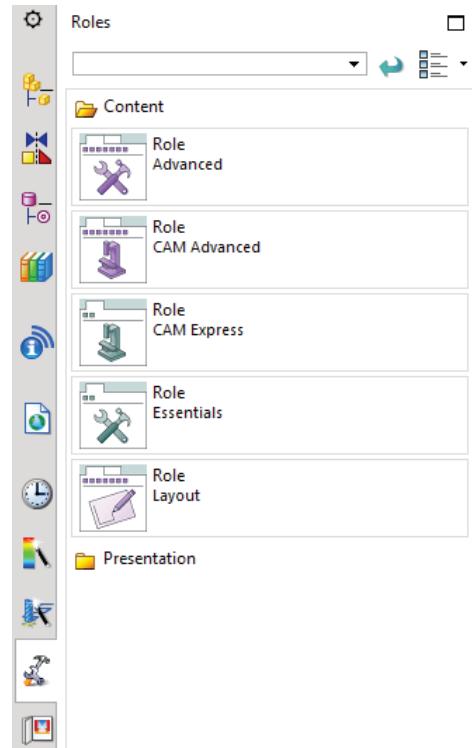
This will pop up a *Customize* dialog window with all the Toolbars and commands pertaining to each Toolbar under *Commands* tab. To add a command,

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



You can customize the settings of your NX 10 interface by clicking on the *Roles* tab on the *Resource Bar*.

The *Roles* tab has different settings of the toolbar menus that are displayed on the NX 10 interface. It allows you to customize the toolbars you desire to be displayed in the Interface.



CHAPTER 3 – TWO DIMENSIONAL SKETCHING

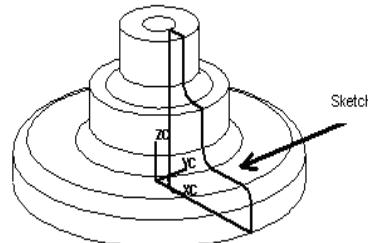
In this chapter, you will learn how to create and edit sketches in NX 10. You can directly create a sketch on a *Plane* in *Modeling* application. In most cases, *Modeling* starts from a 2D sketch and then *Extrude*, *Revolve* or *Sweep* the sketch to create solids. Many complex shapes that are otherwise very difficult to model can easily be drawn by sketching. In this chapter, we will see some concepts of sketching and then proceed to sketch and model some parts.

3.1 OVERVIEW

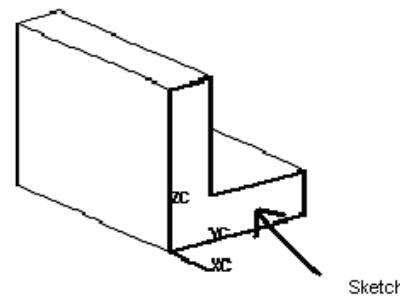
An NX 10 sketch is a named set of curves joined in a string that when swept, form a solid. The sketch represents the outer boundary of that part. The curves are created on a plane in the sketcher. In the beginning, these curves are drawn without any exact dimensions. Then, *Dimensional Constraints* as well as *Geometric Constraints* are applied to fully constrain the sketch. These will be discussed in detail later in this chapter.

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

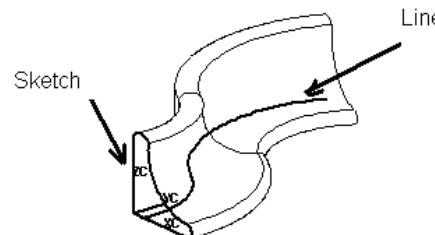
- A sketch can be revolved



- A sketch can be extruded



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



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

The advantages of using sketching to create parts are:

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

3.2 SKETCHING ENVIRONMENT

In NX 10 you can create sketch using two ways. The first method creates the *Sketch* in the current environment and application. For this,

- Choose **Menu → Insert → Sketch**

In the other method you can create Sketch using

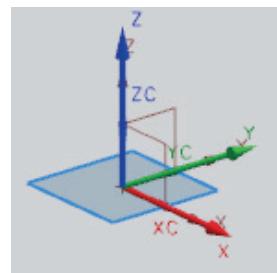
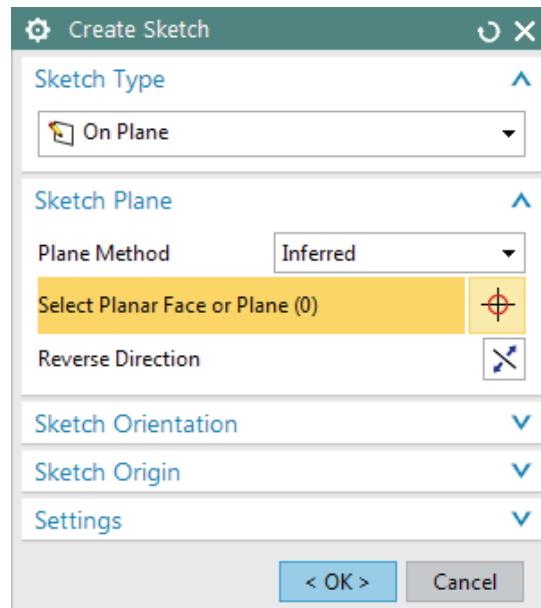
- Choose **Sketch** in **Home** toolbar

In either case, a dialog box pop-ups asking you to define the *Sketch Plane*. The screen will display the sketch options. You can choose the *Sketch Plane*, direction of sketching and type of plane for sketching.

When you create a sketch using the *Create Sketch* dialog box, you can choose the plane on which the sketch can be created by clicking on the coordinate frame as shown. This will highlight the plane you have selected. The default plane selected is XC-YC. However, you can choose to sketch on another plane. If there are any solid features created in the model beforehand, any of the flat surfaces can also be used as a sketching plane.

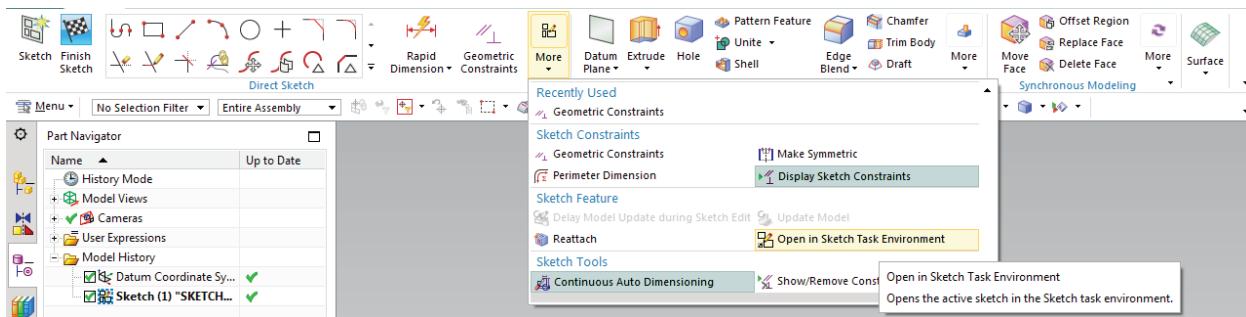
- Choose the **XC-YC** plane and click **OK**

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



The main screen will change to the *Sketching Environment*. The XY plane is highlighted as the default plane for sketching. This is the basic sketch window. There is also a special *Sketch Task Environment* in NX 10 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
- Click on **Open in Sketch Task Environment** as shown below

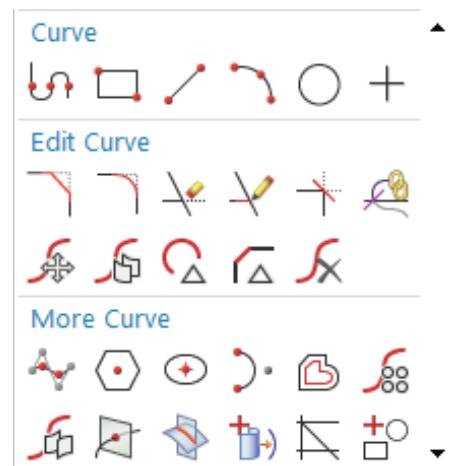


There are three useful options next to the *Finish Flag*. You can change the name of the sketch in the box. The next one is *Orient to Sketch* which orients the view to the plane of the sketch. If the model file is rotated during the process of sketching, click on this icon to view the sketch on a plane parallel to the screen. *Reattach* attaches the sketch to a different planar face, datum plane, or path, or changes the sketch orientation. It allows you to reattach the sketch to the desired plane without recreating all the curves, dimensions, and constraints.



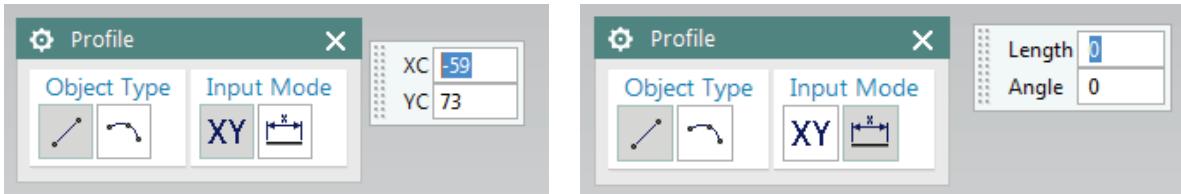
3.3 SKETCH CURVE TOOLBAR

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



Profile

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

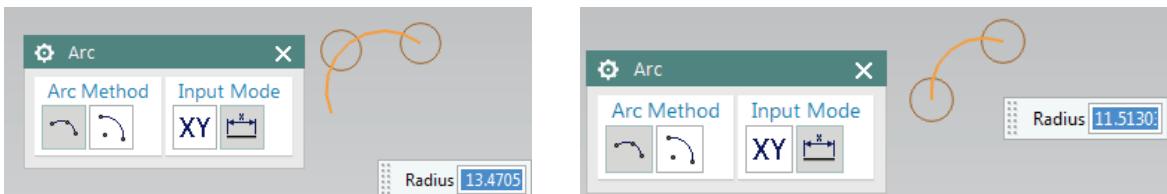


Line

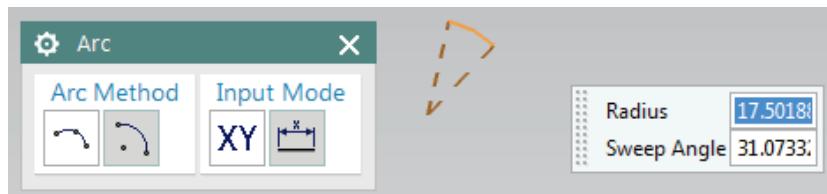
This option will selectively create only straight lines.

Arc

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



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



Circle

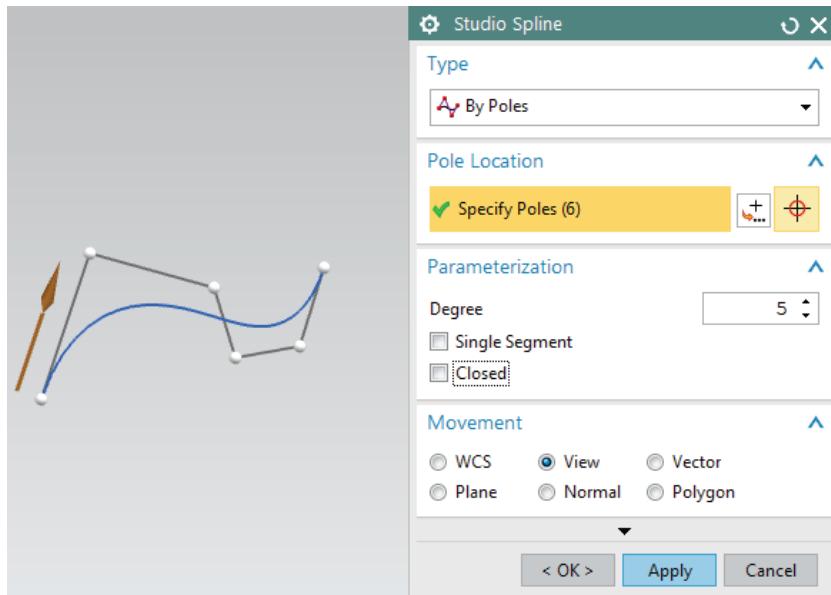
Creating a circle is similar to creating an arc, except that circle is closed.

Quick Trim

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

Studio Spline

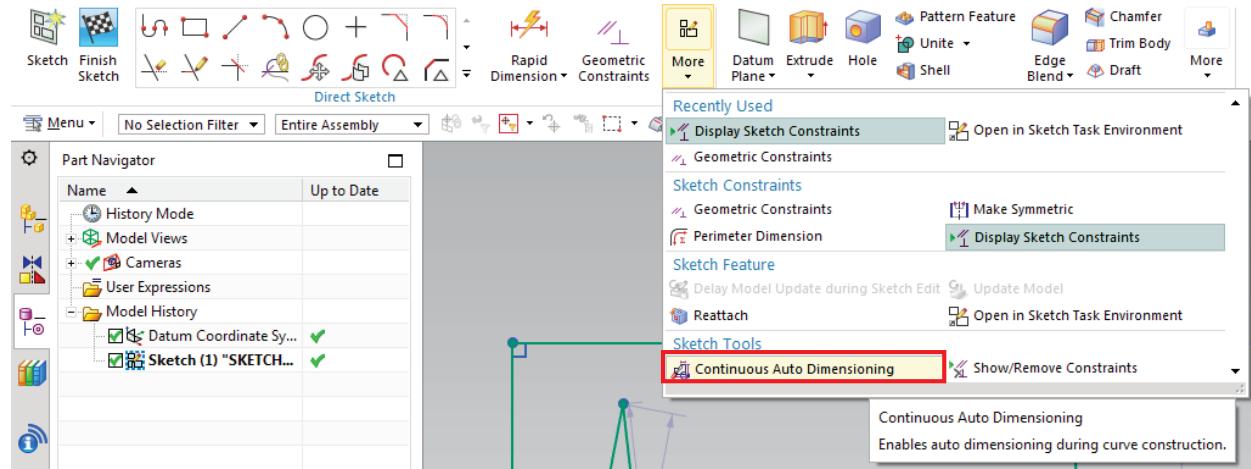
You can create basic spline curves (B-spline and Bezier) with poles or through points with the desired degree of the curve. The spline will be discussed in detail in the seventh chapter (Freeform Features).



3.4 CONSTRAINTS TOOLBAR

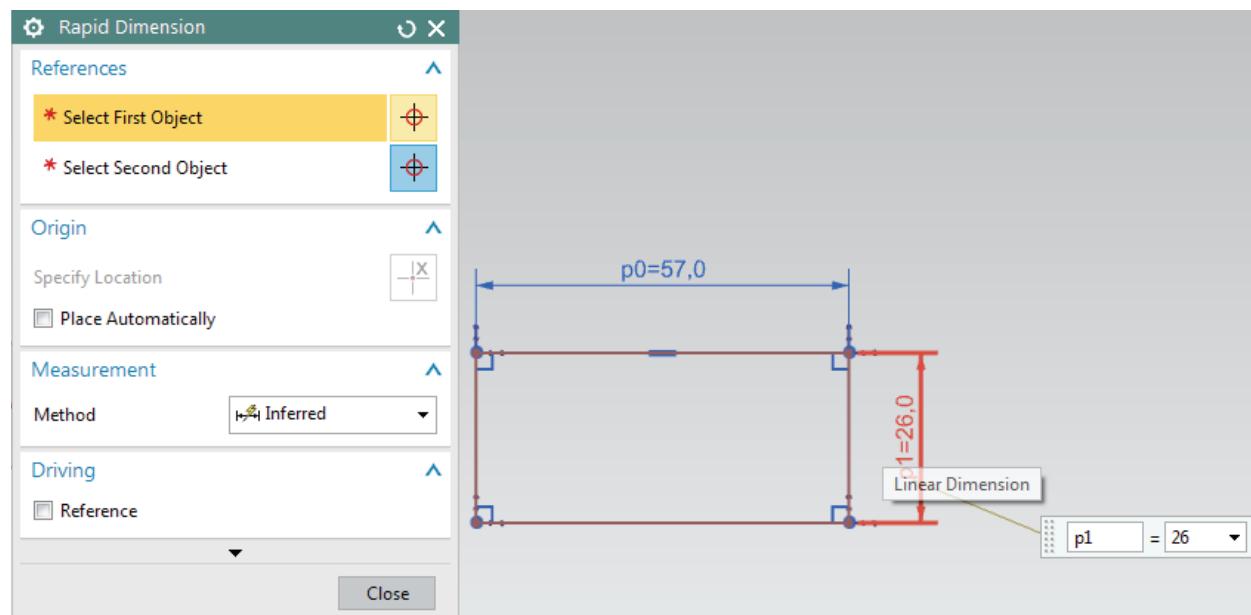
All the curves are created by picking points. For example, a straight line is created with two points. In a 2D environment, any point has two degrees of freedom, one along X and another along Y axis. The number of points depends on the type of curve being created. Therefore, a curve entity has twice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

In NX 10 smart constraints are applied automatically, i.e. automatic dimensions or geometrical constraints are interpreted by NX 10. You can turn this option off by clicking on *Continuous Auto Dimensioning* as shown below. The following paragraphs show how to manually apply constraints.



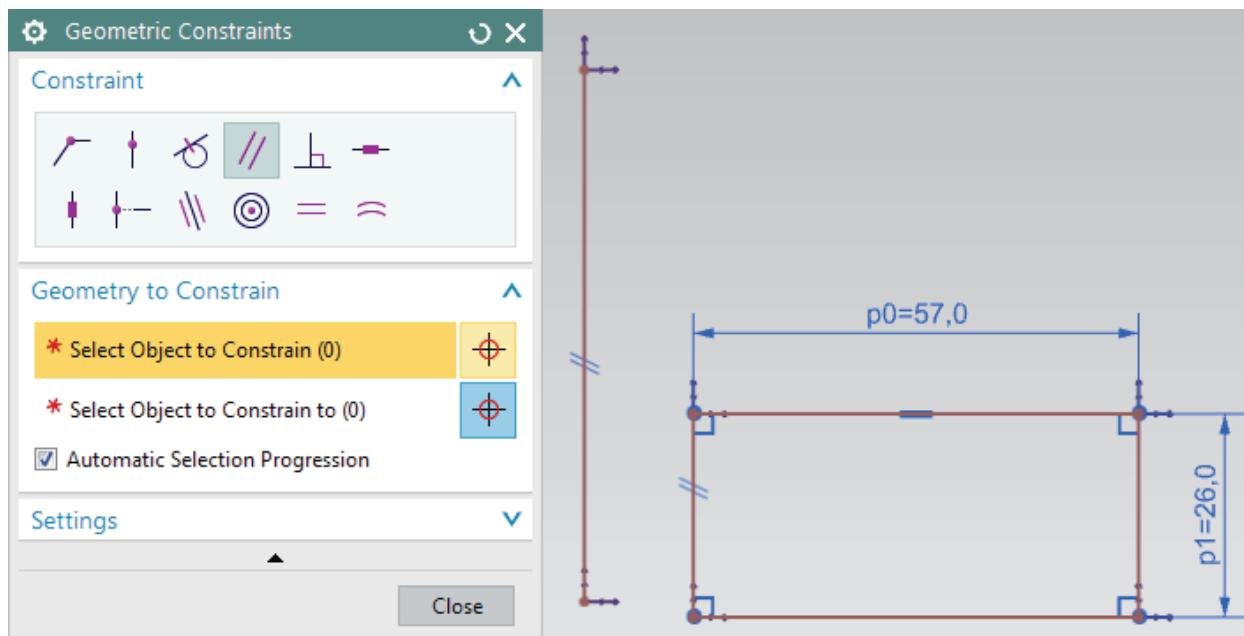
Dimensional Constraints

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



Geometric Constraints

Besides the dimensional constraints, some geometric constraints can be given to eliminate the degrees of freedom. They include 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. As an example, a constraint is applied on the line in the below picture to be parallel to the left side of the rectangle (the line was originally at an angle with the rectangle).



Display Sketch 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.

The number of degrees of freedom that are not constrained are displayed in the *Status Line*. All these should be removed by applying the constraints to follow a disciplined modeling.

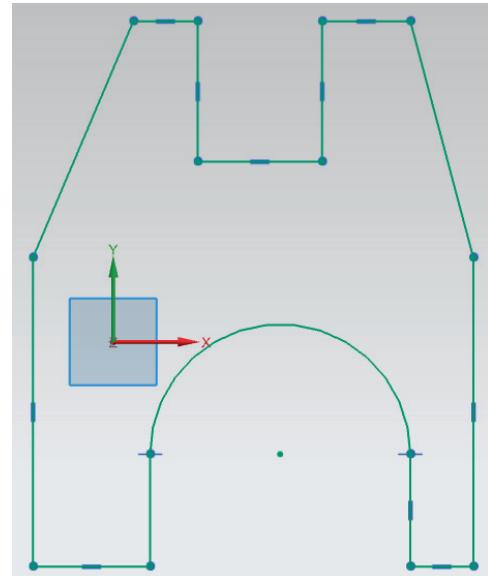
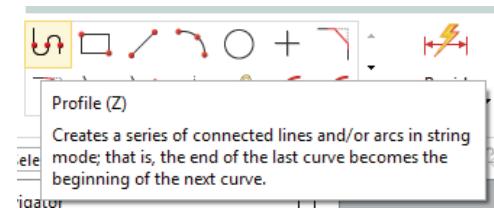
3.5 EXAMPLES

3.5.1 Arbor Press Base

- Create a new file and save it as **Arborpress_base.prt**
- Click on the **Sketch** button and click **OK**
- Choose **Menu → Insert → Sketch Curve → Profile** or click on the **Profile** icon in the **Direct Sketch** group (remember to deactivate **Continuous Dimensioning**)
- Draw a figure similar to the one shown on right. While making continuous sketch, click on the **Line** icon on the **Profile** dialog box to create straight lines and the **Arc** icon to make the semicircle. (Look at the size of the XY plane in the figure. Use that perspective for the approximate zooming).

Once the sketch is complete, we 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

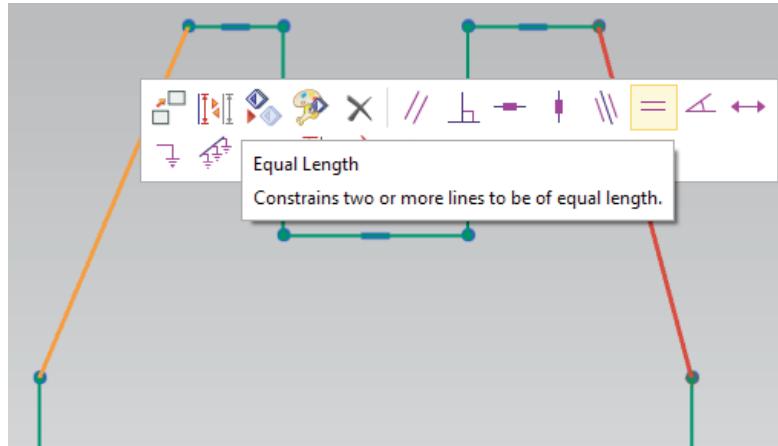


Now we start by constraining between an entity in the sketch and a datum or a fixed reference. First, place the center of the arc at the origin. This creates a reference for the entire figure. We can use the two default X and Y axes as a datum reference.

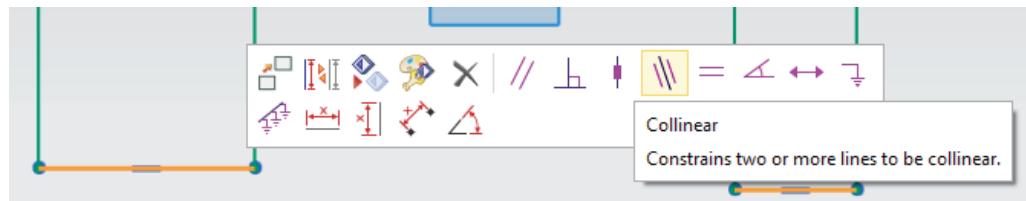
- Select **Point on Curve**  constraint
- Select the Y-axis and then the center of the arc
- Repeat the same procedure to place the center of the arc on the X-axis

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

- Select the two slanted lines and make them **Equal Length**



- Similarly select the two long vertical lines and make them **Equal Length**
- Select the bottom two horizontal lines and make them **Collinear** and then click on the same lines and make them **Equal 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.



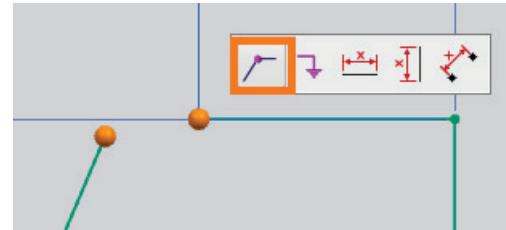
- Select the circular arc and one of the two vertical lines connected to its endpoints
- Select the **Tangent** icon

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.

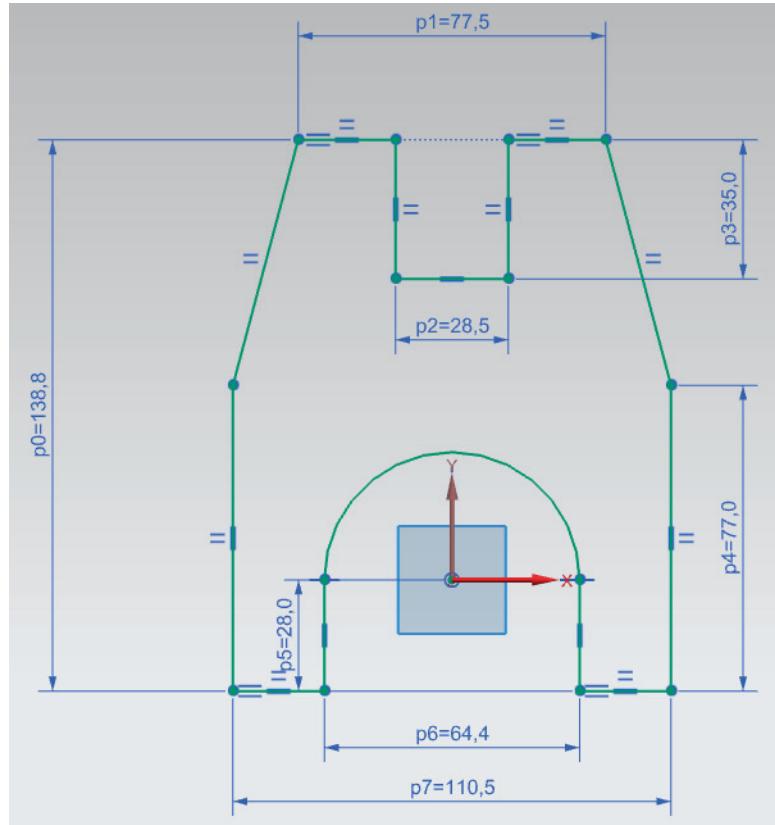
- Select the two vertical lines and make them **Equal**
- Similarly select the two small horizontal lines at the top of the profile and make them **Collinear and Equal**
- Similarly select the two vertical lines and make them **Equal**

Note: At times 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.



So far, we have created all the *Geometric* constraints. Now we have to create the *Dimensional* constraints. If there is any conflict between the *Dimensional* and *Geometric* constraints, those entities will be highlighted in yellow.

- Choose the **Rapid Dimension** icon  in the **Constraints** toolbar
- Add on all the dimensions as shown in the following figure without specifying the values



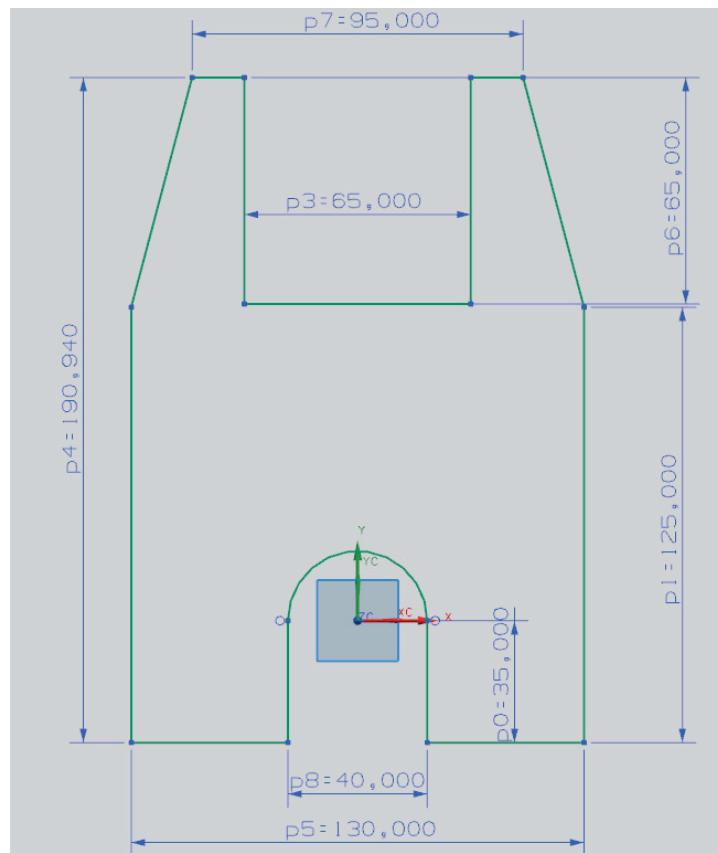
For example, to create a dimension for the top two corners,

- Click somewhere near the top of the two diagonal lines to select them

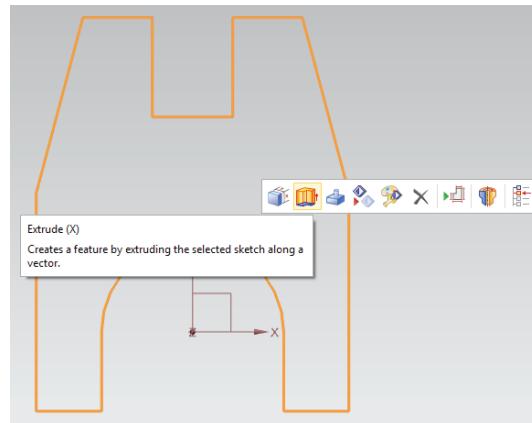
While dimensioning, if you find the dimensions illegible, but do not worry about editing the dimensions now.

Now we 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. Once enough number of dimensions are provided, sketch color changes indicating it is fully defined.

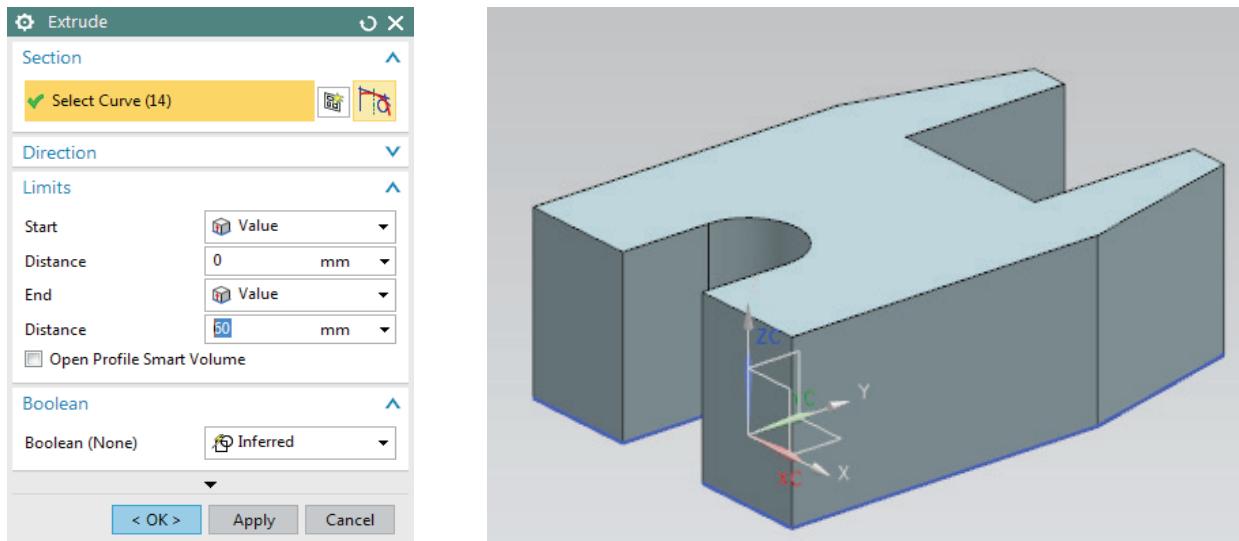
- Edit the values as shown in the figure below. Double click on each dimension to change the values to the values as shown in figure below



- Click on the **Finish Flag**  on the top left corner or bottom right of the screen when you are finished
- Click on the sketch and select **Extrude** (this **Feature** is explained in details in the next sections)



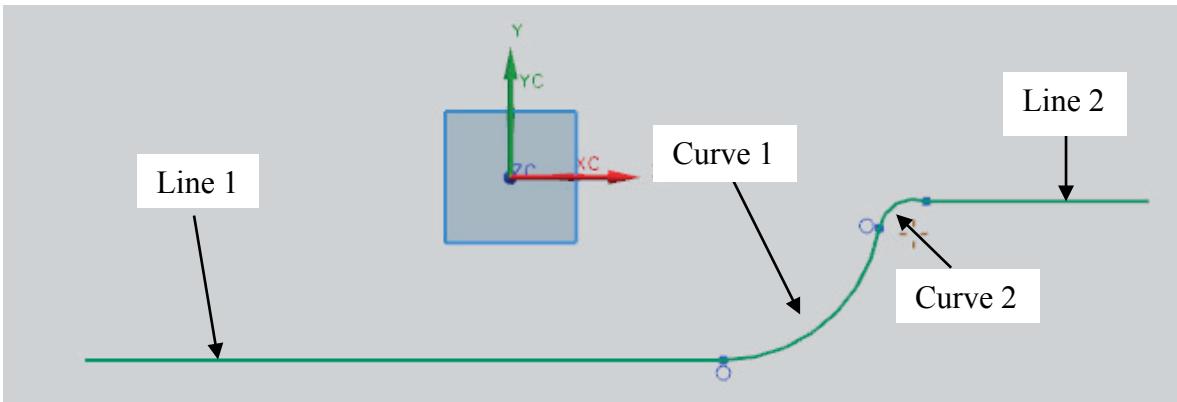
- Extrude this sketch in the Z-direction by **60 mm**



- Save and close the file

3.5.2 Impeller Lower Casing

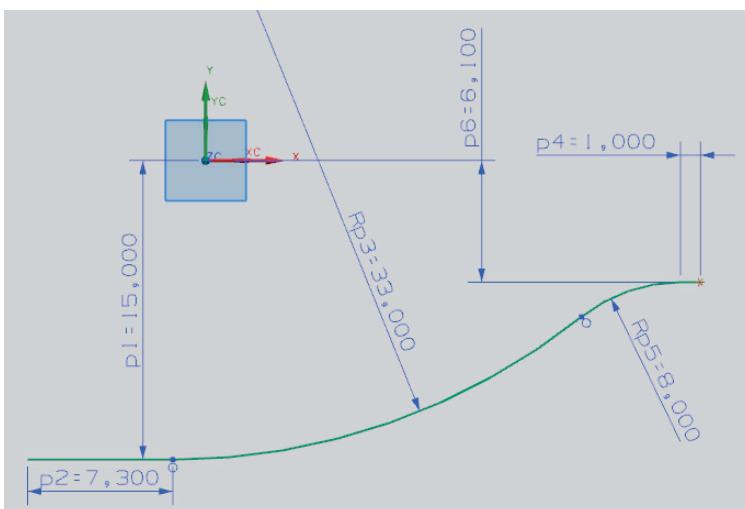
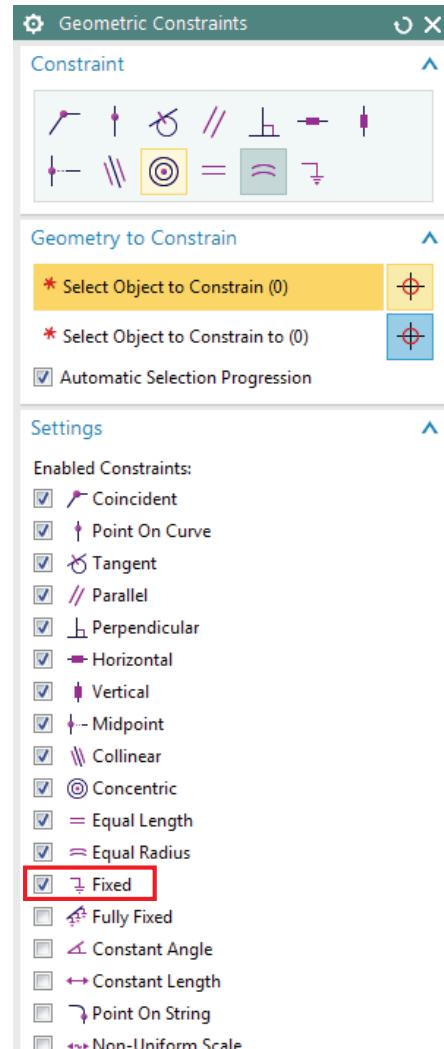
- Create a new file in **inches** and save it as **Impeller_lower_casing.prt**
- Click **Menu → Insert → Sketch In Task Environment** or click **Sketch In Task Environment** icon from the ribbon bar
- Set the sketching plane as the **XC-YC** plane
- Make sure the **Profile** window is showing and draw the following curve



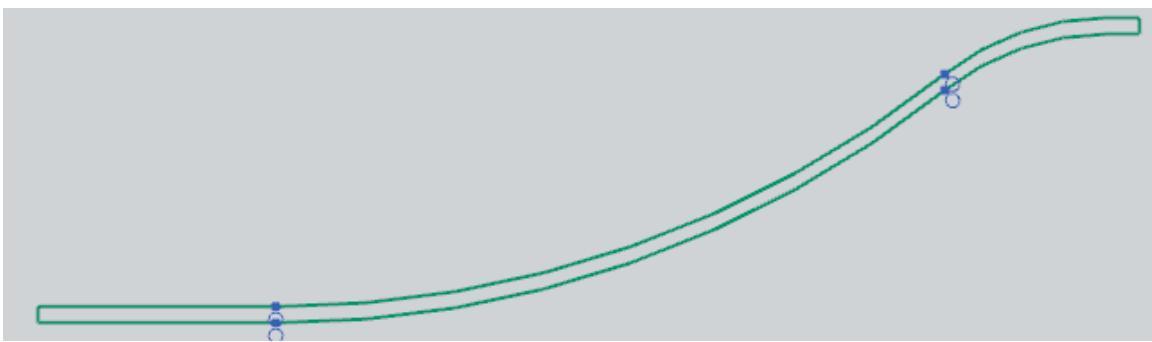
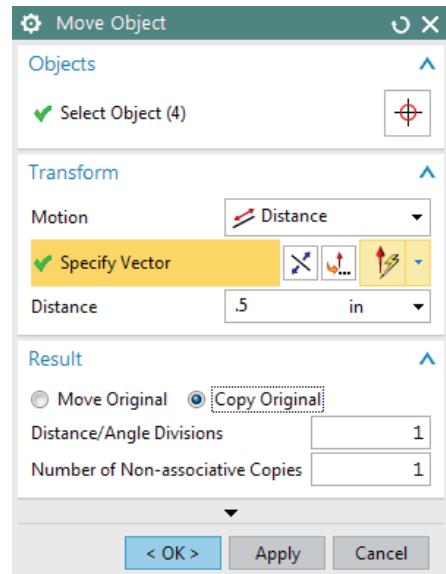
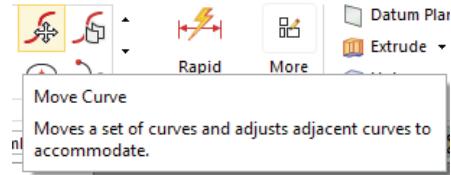
- Create a point at the origin $(0, 0, 0)$ by clicking the plus sign in the **Direct Sketch** group

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 (if you cannot see this icon, click on **Settings** and check it as shown on the right)
- Make all of the curve-lines and curve-curve joints **Tangent**
- Apply the dimensional constraints as shown in the figure below



- Select all the dimensions.
- Right click and **Hide** the dimensions
- Choose **Menu →Edit →Move Object** or choose **Move Curve** from the ribbon bar
- 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



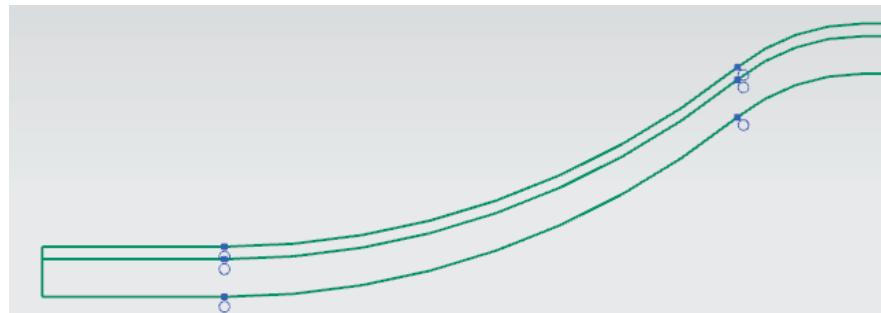
The sketch is ready.

- Choose **Edit →Move Object** or choose **Move Curve** from the ribbon bar
- Select the outer curve. 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.

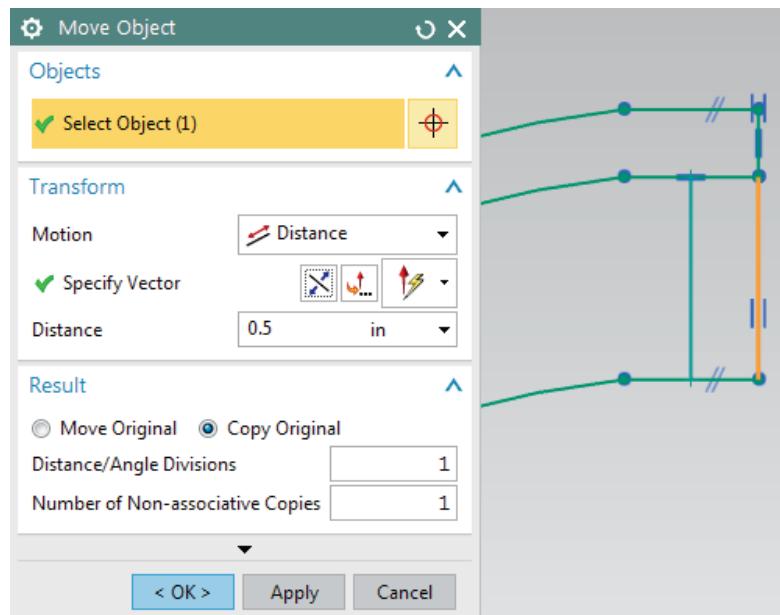
- Using **straight lines** join this curve with the inside curve of the casing

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 line as shown in the figure below



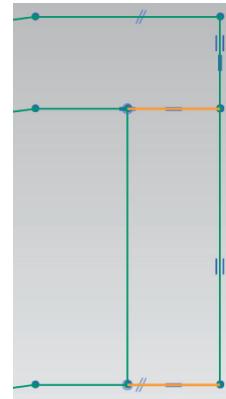
- **Move** the lower curve in the **XC**-direction by **-0.5** inches. This is the same as translating it in the **negative XC-direction** by **0.5** inches

- Using **straight lines** join the two lines as shown in the figure on right side

- Click on the Finish Flag 

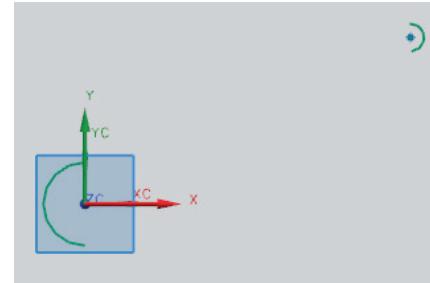
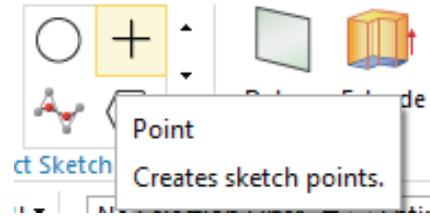
- Save and Close the file

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



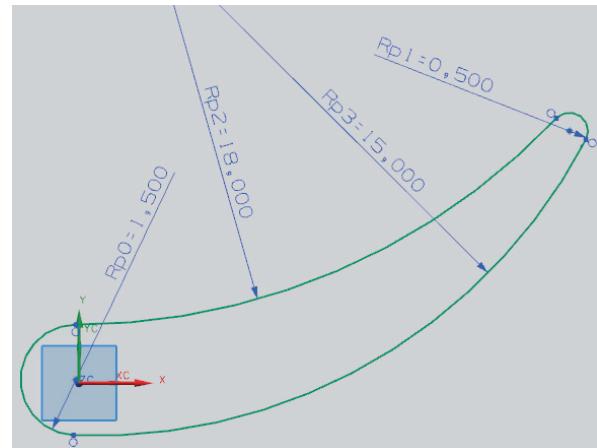
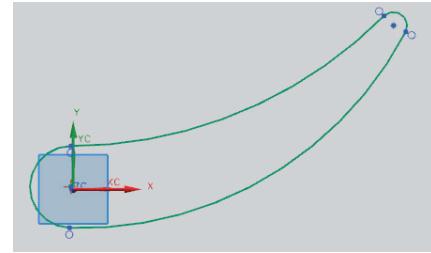
3.5.3 Impeller

- Create a new file in **inches** and save it as **Impeller_impeller.prt**
- Click on **Sketch**
- Set the sketching plane as the XC-YC plane and click **OK**
- Click on **Menu → Insert → Datum/Point → Point** or click **Point** from **Direct Sketch** group in the ribbon bar
- 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
- Then click on the **Rapid Dimension** icon
- Give the **Radius** dimensions for each arc. Edit dimensions so that the two arcs on the end are 1.5 and 0.5 inches and the two middle arcs are 18 and 15 inches as shown in the figure below
- Click on the **Finish Flag** 
- Save and Close the file.

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



CHAPTER 4 – THREE DIMENSIONAL MODELING

This chapter discusses the basics of three dimensional modeling in NX 10. We will discuss what a feature is, what the different types of features are, what primitives are and how to model features in NX 10 using primitives. This will give a head start to the modeling portion of NX 10 and develop an understanding of the use of *Form Features* for modeling. Once these features are introduced, we will focus on *Feature Operations* which are functions that can be applied to the faces and edges of a solid body or features you have created. These include taper, edge blend, face blend, chamfer, trim, etc. After explaining the feature operations, the chapter will walk you through some examples.

In NX 10, *Features* are a class of objects that have a defined parent. Features are associatively defined by one or more parents and the order of their creation and modification retain within the model, thus capturing it through the *History*. Parents can be geometrical objects or numerical variables. Features include primitives, surfaces and/or solids and certain wire frame objects (such as curves and associative trim and bridge curves). For example, some common features include blocks, cylinders, cones, spheres, extruded bodies, and revolved bodies.

Commonly *Features* can be classified as 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

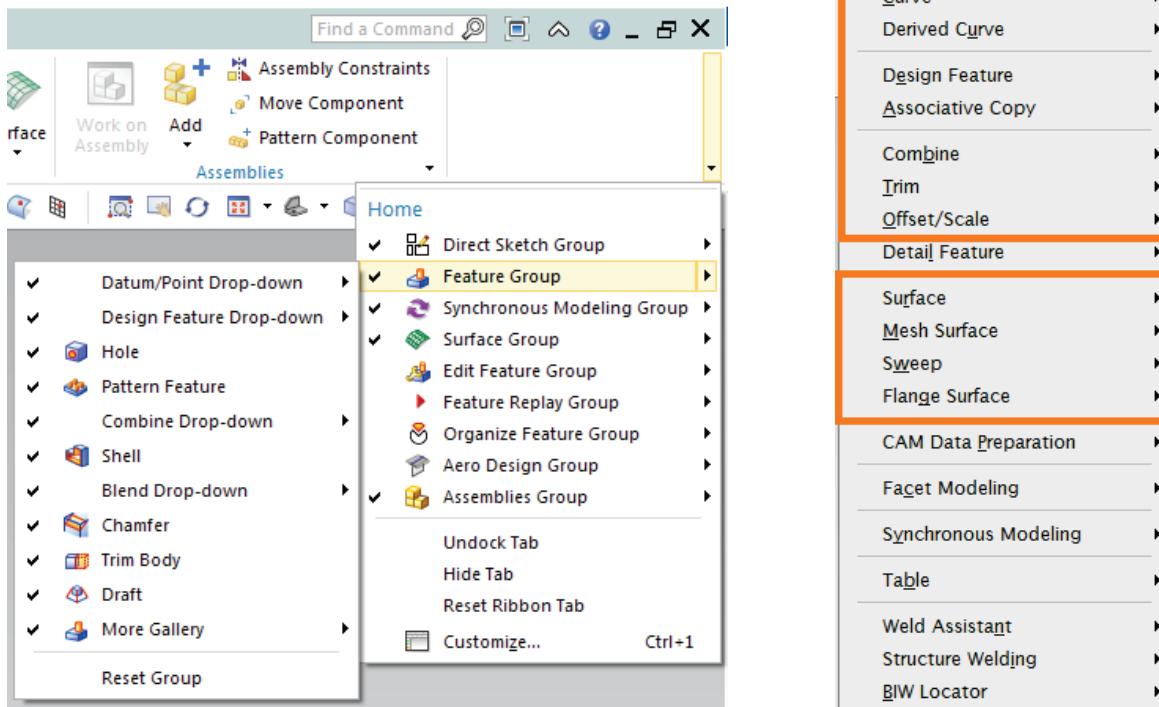
4.1 TYPES OF FEATURES

There are six types of *Form Features*: Primitives, Reference features, Swept features, Remove features, Extract features, and User-defined features. Similar to previous versions, NX 10 stores all the *Form Features* under the *Insert* menu option. The form features are also available in the *Form Features Toolbar*.

- Click **Insert** on the **Menu**

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

- Click on the drop down arrow in **Home Toolbar**
- Choose **Feature Group**



4.1.1 Primitives

They let you create solid bodies in the form of generic building shapes. Primitives include:

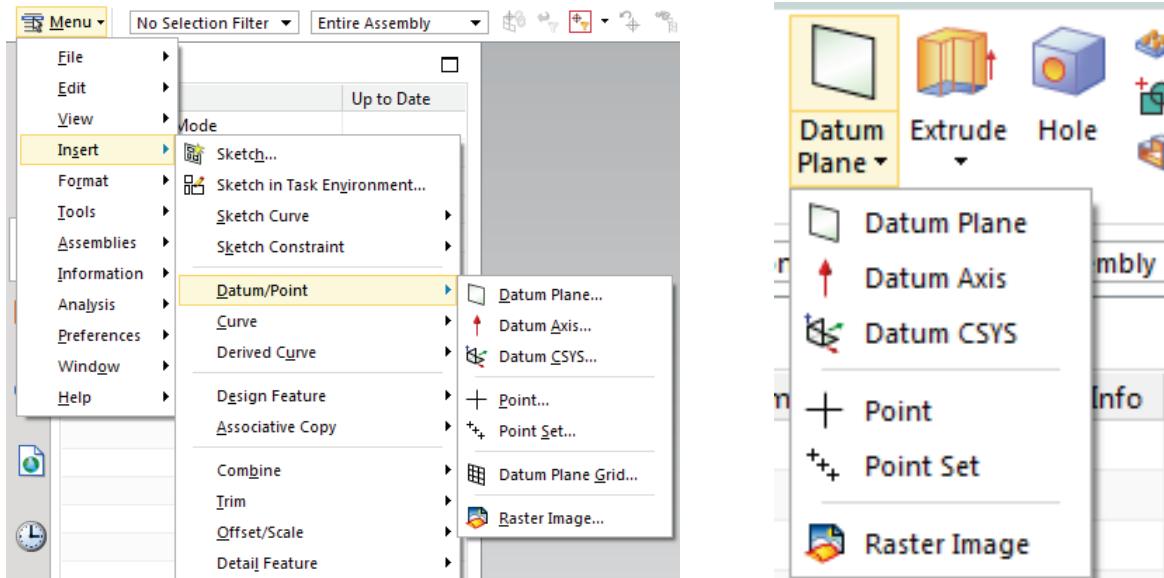
- Block
- Cylinder
- Cone
- Sphere

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

4.1.2 Reference Features

These features 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** or click on **Datum Plane** in **Feature** group in the ribbon bar to view the different **Reference Feature** options: **Datum Plane**, **Datum Axis**, **Datum CSYS**, and **Point**



4.1.3 Swept Features

These features 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 **Extrude** in **Feature** group in the ribbon bar

OR

- Click on **Insert → Sweep** or click on **More** in **Feature** group in the ribbon bar to find all the options available including **Sweep**

4.1.4 Remove Features

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

- Click on **Insert → Design Feature**

Remove Features include:

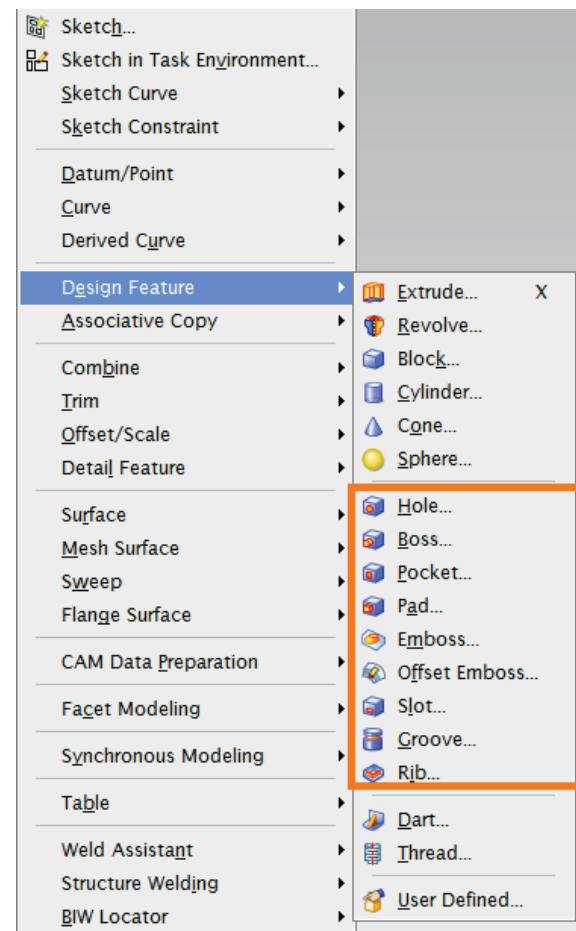
- Hole
- Pocket
- Slot
- Groove

4.1.5 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
- Solid to Shell
- Thicken Sheet
- Bounded plane
- Sheet from curves

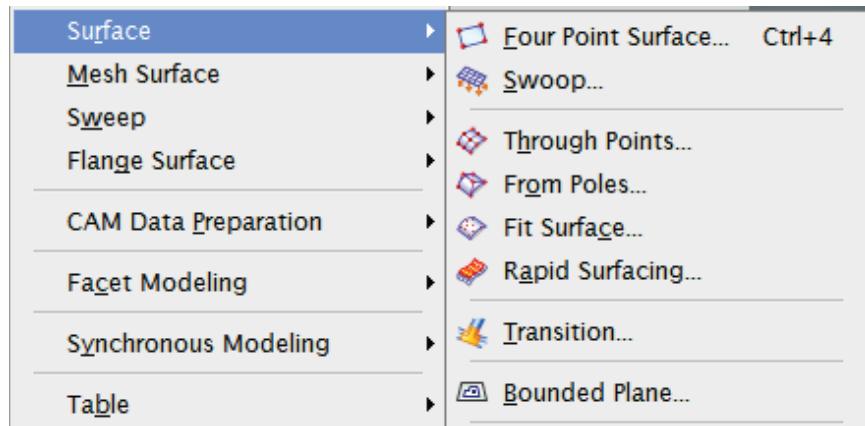
- Click on **Insert → Associative Copy → Extract** for Extract options or click on **More** in **Feature** group in the ribbon bar to find **Extract Geometry**



- Click on **Insert → Offset/Scale** for **Solid to Shell** and **Thicken Sheet** Assistant or click on **More** in **Feature** group in the ribbon bar to find **Offset/Scale** options



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



4.1.6 User-Defined features

These features 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**

4.2 PRIMITIVES

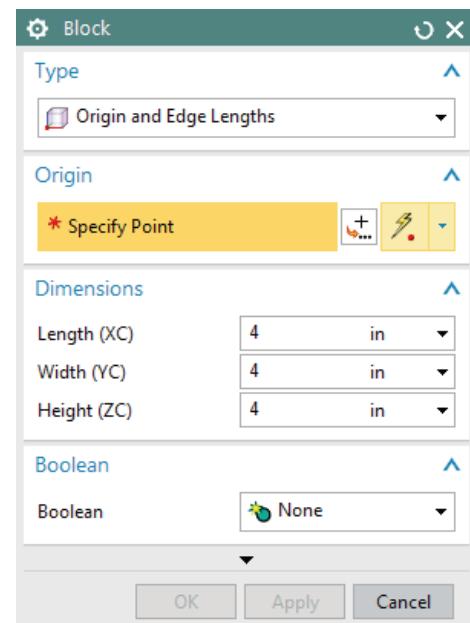
Primitive features are base features from which many other features can be created. The basic primitives are blocks, cylinders, cones and spheres. Primitives are non-associative which means they are not associated to the geometry used to create them.

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

4.2.1 Model a Block

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

The *Block* window appears. There are three main things to define a block. 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 and Edge Lengths
 - Height and Two Points
 - Two Diagonal Points
- Make sure the **Origin and Edge Lengths** method is selected

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

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

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

- Click **OK**

The *Block* window will reappear again.

- Type the following dimensions in the window

Length (XC) = **65** inches

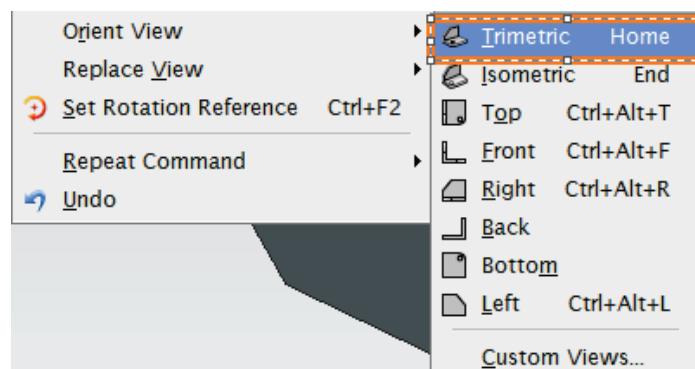
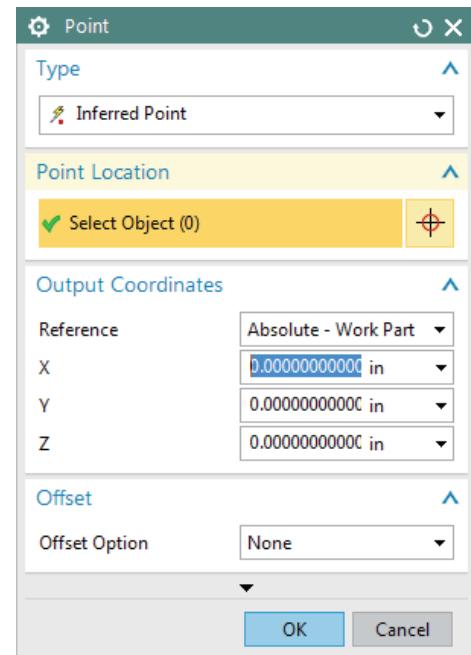
Width (YC) = **85** inches

Height (ZC) = **20** inches

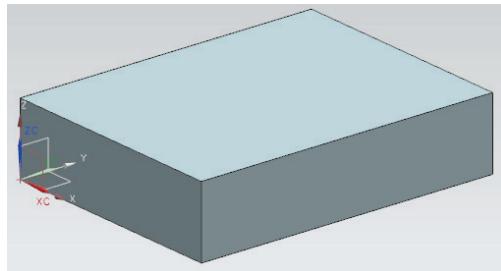
- Click **OK**

If you do not see anything on the screen,

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



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



4.2.2 Model a Shaft

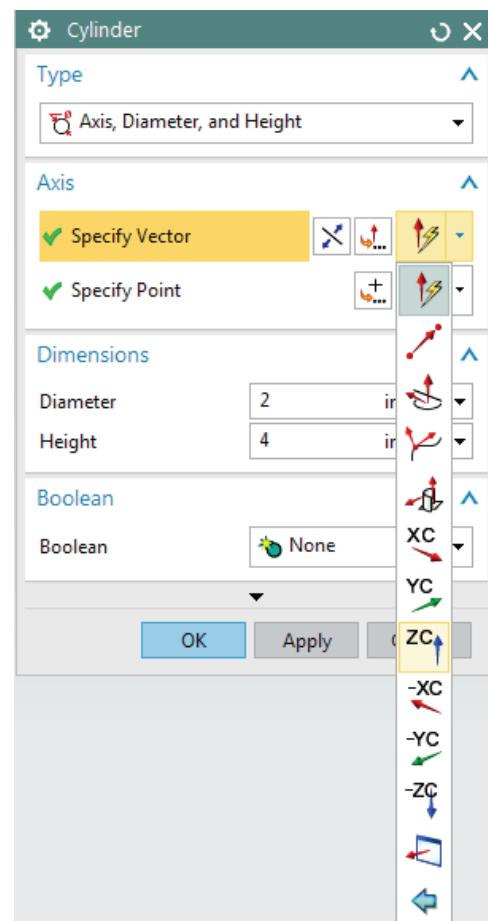
We will now model a shaft having two cylinders and one cone joined together.

- Create a new file and save it as **Impeller_shaft.prt**
- Choose **Insert → Design Feature → Cylinder** or click on **More** in **Feature** group in the ribbon bar to find **Cylinder** in **Design Feature** section

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

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

- Axis, Diameter, and Height
- Arc and Height
- Select **Axis, Diameter, and Height**
- Click on the **Vector Constructor** icon next to **Specify Vector** and select the **ZC** Axis icon
- Click on the **Point Dialog** icon next to **Specify Point** to set the origin of the cylinder
- Set all the **XC**, **YC**, and **ZC** coordinates to be 0



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

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

Diameter = 4 inches

Height = 18 inches

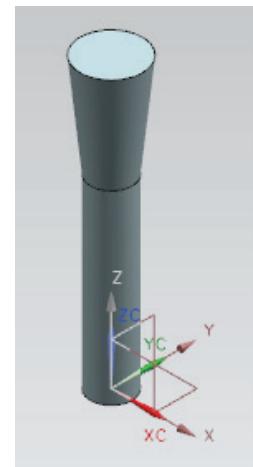
- Click **OK**
- Right-click on the screen, choose **Orient View → Isometric**

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



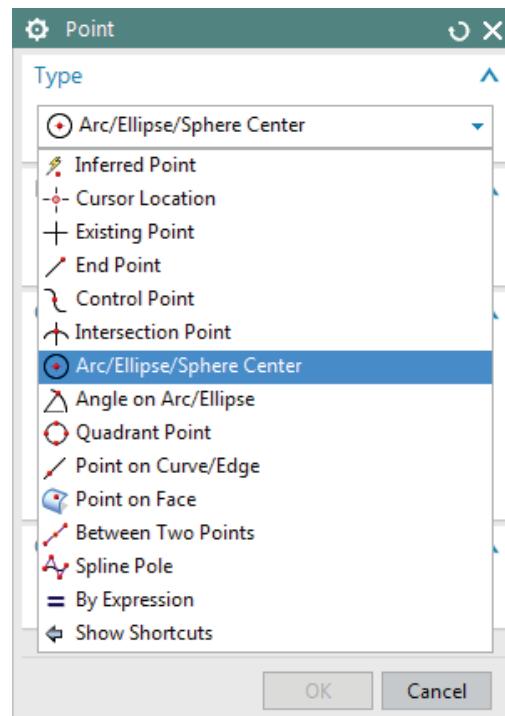
- Choose **Insert → Design Feature → Cone** or click on **More** in **Feature** group in the ribbon bar to find **Cone** in **Design Feature** section

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



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

- Select **Diameters and Height**
- Click on the Vector Constructor icon next to **Specify Vector**
- Choose the **ZC-Axis** icon so the vector is pointing in the positive Z direction
- Click on the **Point Constructor** icon next to **Specify Point** to set the origin of the cylinder.



The *Point Constructor* window will appear next.

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

OR

- For the **Output 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

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

- Click **OK**

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

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



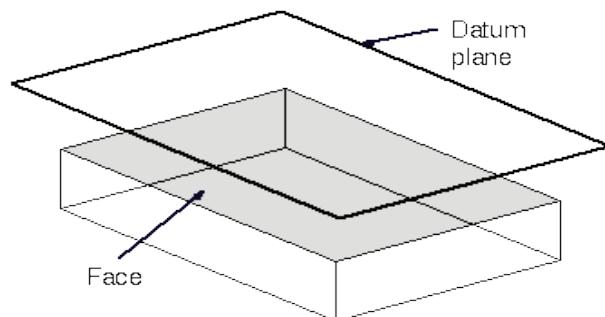
- Repeat the same procedure as before to create another **Cylinder**. The vector should be pointing in the positive *ZC-direction*. On the **Point Constructor** window, again click on the **Center** icon and construct it at the center point of the base of the cone. The cylinder should have a diameter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

The complete shaft will look as shown on the right. Remember to save the model.

4.3 REFERENCE FEATURES

4.3.1 Datum Plane

Datum Planes are reference features that can be used as a base feature in building a model. They assist in creating features on cylinders, cones, spheres, and revolved solid bodies which do not have a planar surface and also aid in creating



features at angles other than normal to the faces of the target solid. We will follow some simple steps to practice *Reference Features*. For starters, we will create a *Datum Plane* that is offset from a face as shown in the figure below.

- 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*.

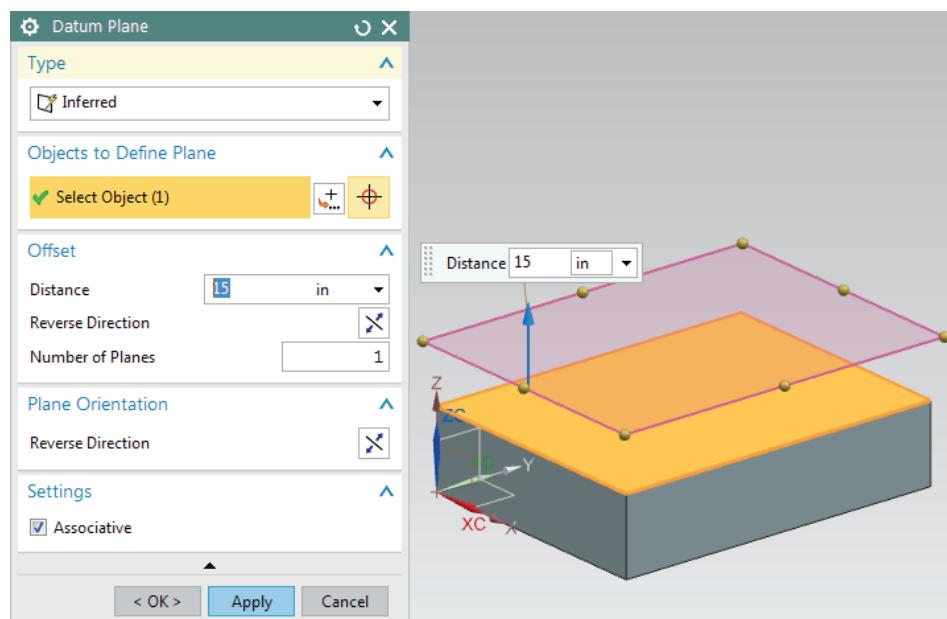


The *Datum Plane* window allows you to choose the method of selection. However, NX 10 is smart enough to judge the method depending on the entity you select if you keep in *Inferred* option, which is also the *Default* option.

- 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.

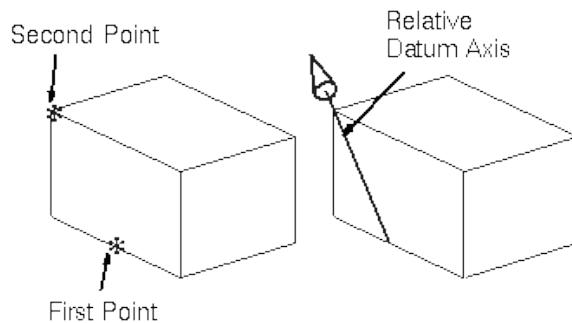
- Insert the **Offset Distance** value as **15** inches in the dialog box and click **OK**



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

4.3.2 Datum Axis

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

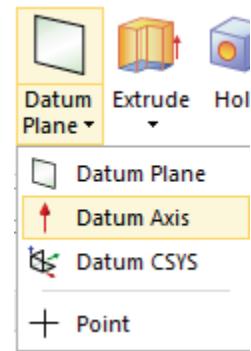


- Choose **Insert → Datum/Point → Datum Axis**

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

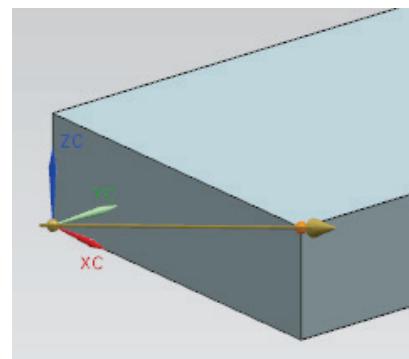
The next window allows you to choose the method of selecting the axis. However, NX 10 can judge which method to use depending on the entity you select.

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



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

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

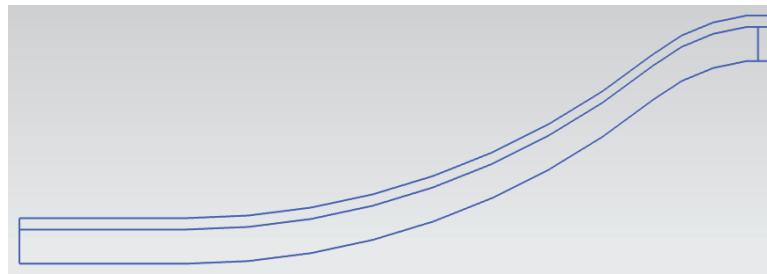


4.4 SWEPT FEATURES

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

- Open the **Impeller_lower_casing.prt**

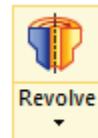
In the previous section, we finished the two dimensional sketching of this part and it should look similar to the below figure.



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

OR

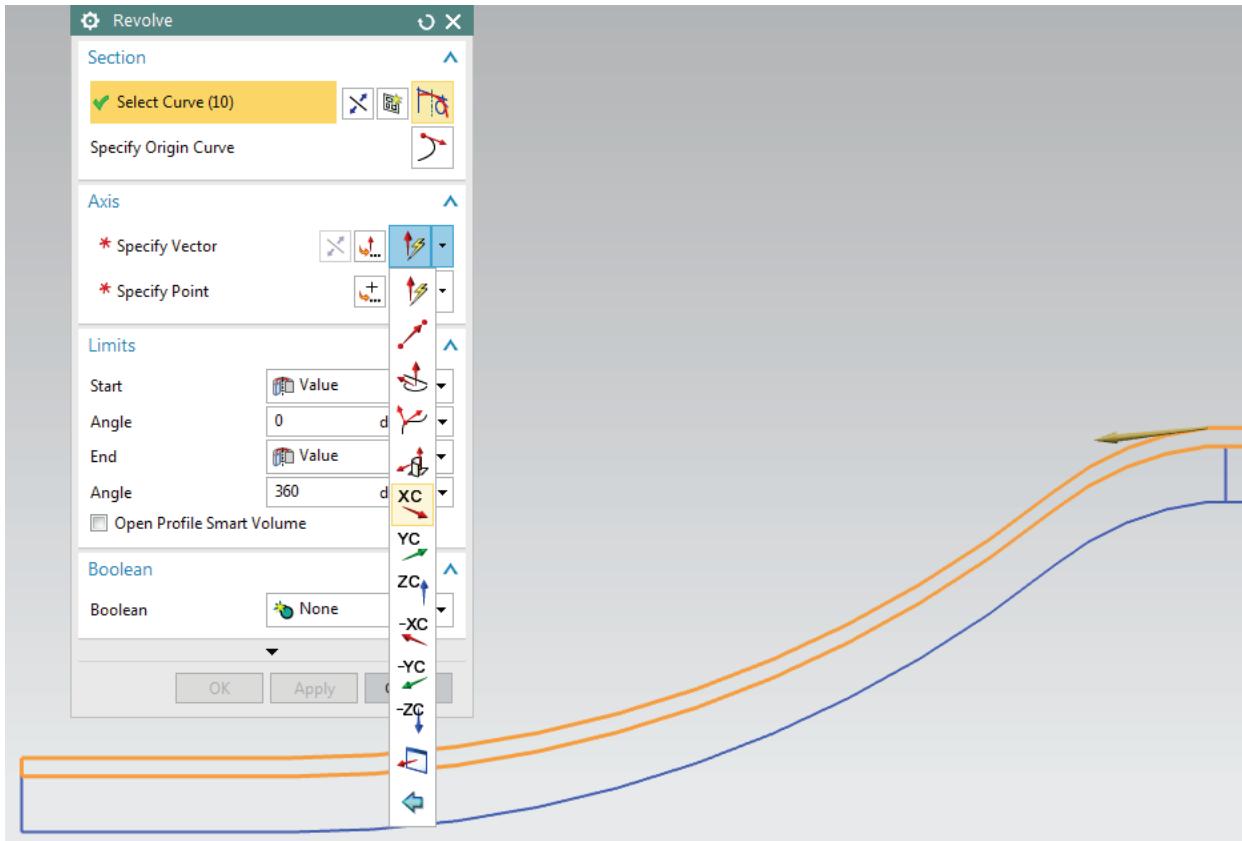
- Click on the **Revolve** button in the **Feature Group**



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



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



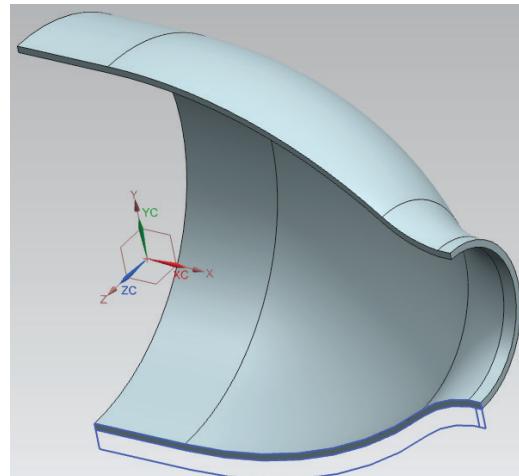
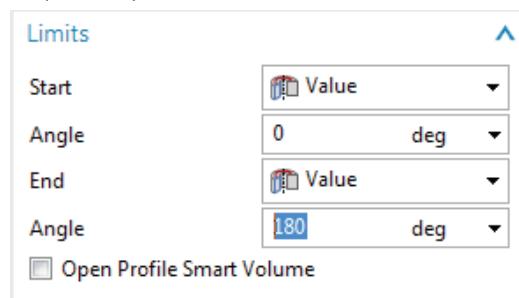
- In the **Specify Point** option, enter the coordinates **(0, 0, 0)** so the curve revolves around **XC-axis** with respect to the origin
- Keep the **Start Angle** as **0** and enter **180** as the value for the **End Angle**
- Click **OK**

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

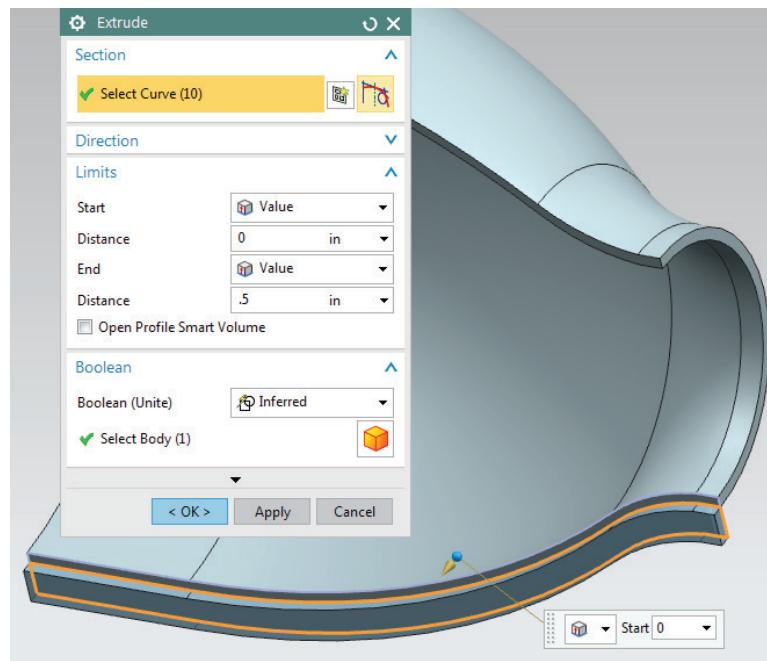
- Click on **Insert → Design Feature → Extrude**

OR

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



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

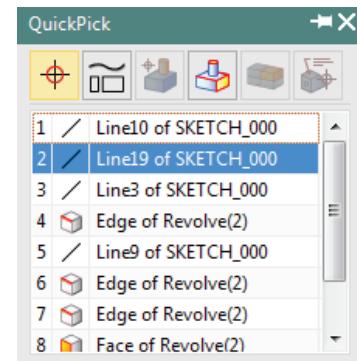


Note: In case you are not able to select the proper lines, left-click and hold the mouse button and you will see a dialog box pop-up as shown which will provide you the options of which curve to select.

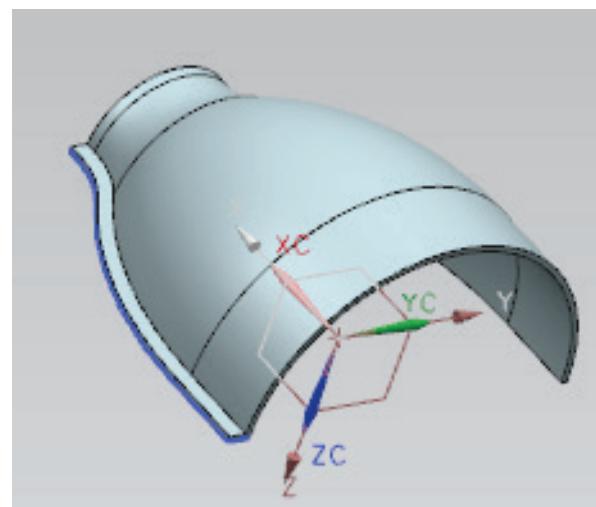
- Extrude this piece in the **negative Z-direction** by **0.5** inches

The final solid will be seen as follows.

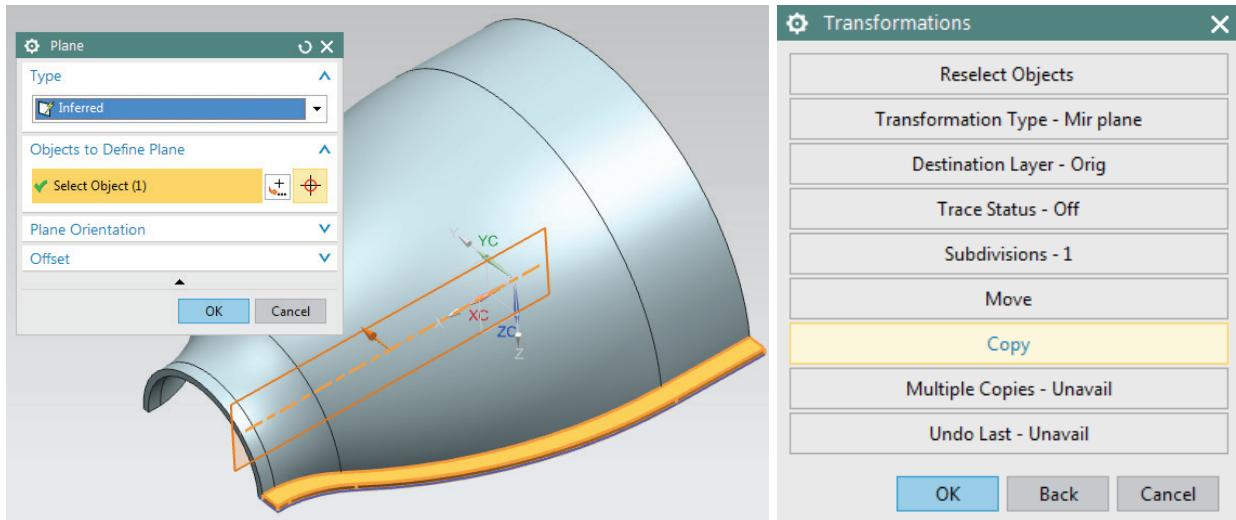
We will now use the *Mirror* option to create an edge on the other side.



- Choose **Edit → Transform**
- Select the solid edge as shown. For this you will have to change the Filter in the dialog box to **Solid Body**
- Click **OK**
- Choose **Mirror Through a Plane**
- Select the **Center Line** as shown below

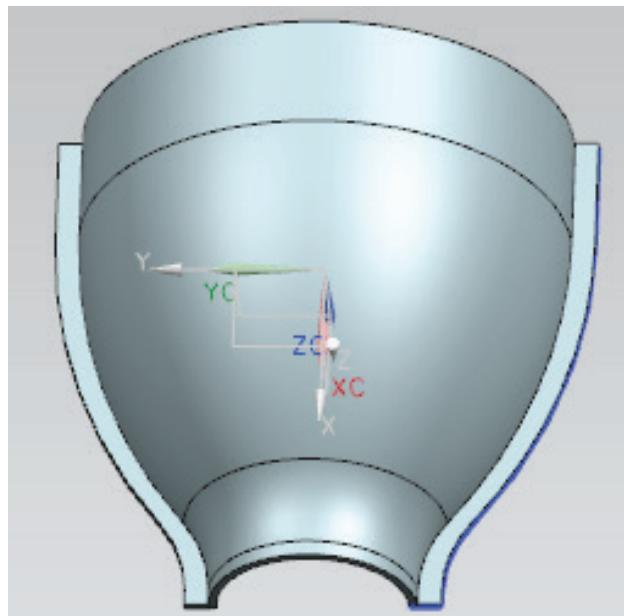


- Click **OK**
- Select **Copy**

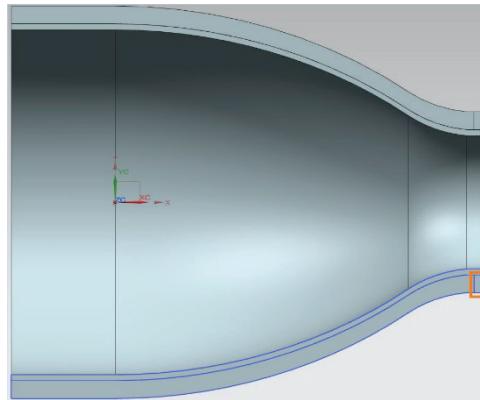


- Click **Cancel**

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



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

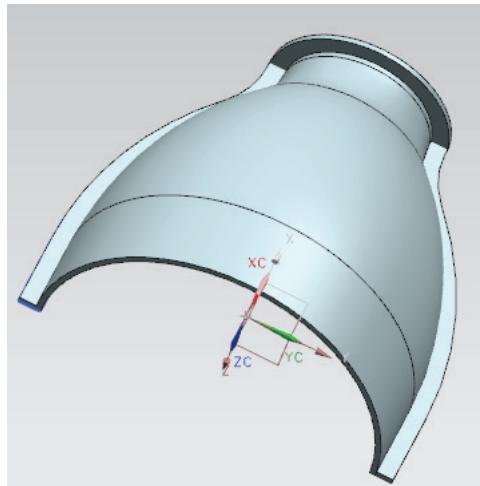


- Click on **Insert → Design Feature → Revolve**

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

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

This will form the edge as shown below.



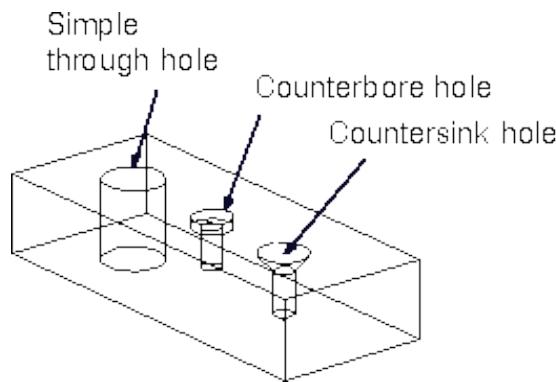
The lower casing is complete. Save the model.

4.5 REMOVE FEATURES

Remove 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.

4.5.1 General Hole

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



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

OR

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

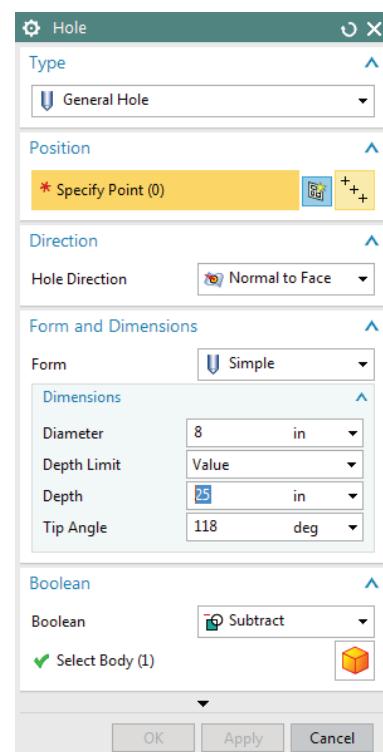
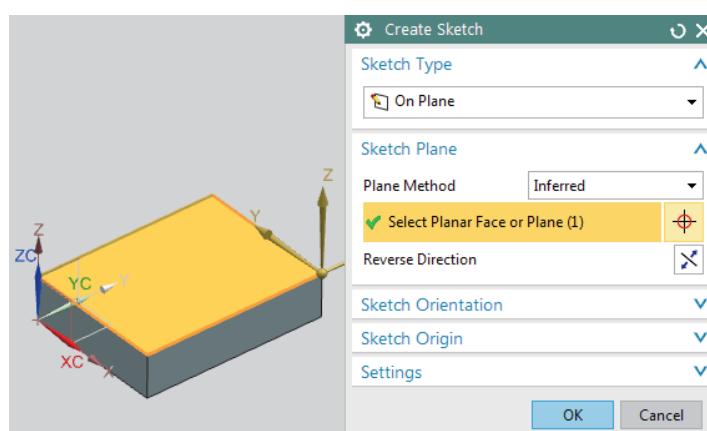


The *Hole* window will open. There are various selections that need to be done prior to making the holes. First you need to select the *Type* of the hole.

- Select the default **General Hole**

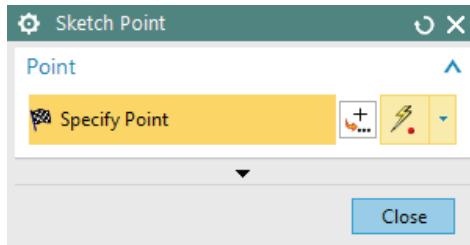
Next, you need to define the points at which you need to make the holes.

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



- Click **OK**

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



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

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

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

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



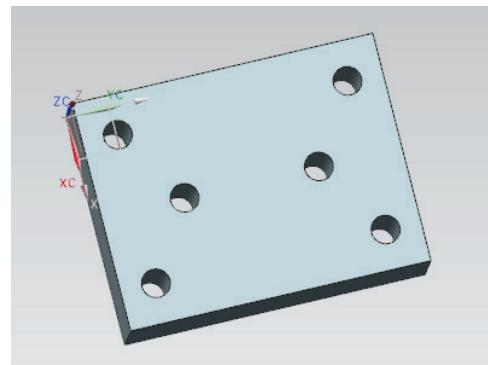
- In the **Form** dialog, choose the default option of **Simple Hole**

- Enter the following values in the **Dimensions** window

Diameter = **8** inches

Depth = **25** inches

Tip Angle = **118** degrees



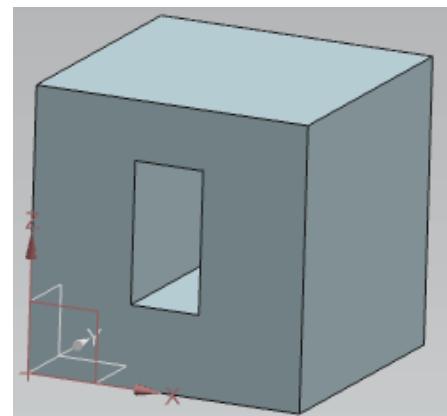
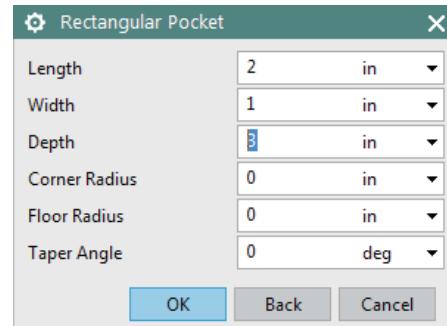
- Choose **Subtract** in the **Boolean** dialog box and click **OK**

Make sure to save the model.

4.5.2 Pocket

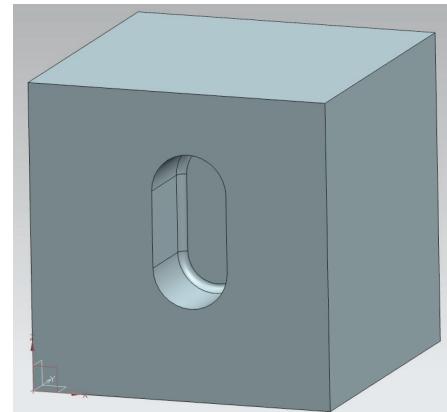
This creates a cavity in an existing body.

- Create a **Block** using default values
- Choose **Insert → Design Features → Pocket**
- Select **Rectangular**
- Select the **Face** that you want to create the **Pocket** on it
- Select a **Vertical Face** to use as the reference for dimensioning
- Enter the dimensions of the **Pocket** as shown
- Change the **Positioning** if you want



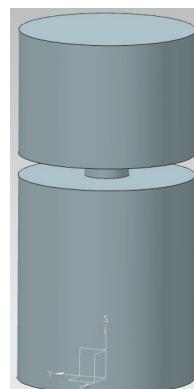
4.5.3 Slot

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. An example is shown on the right.



4.5.4 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. An example is shown on the right.



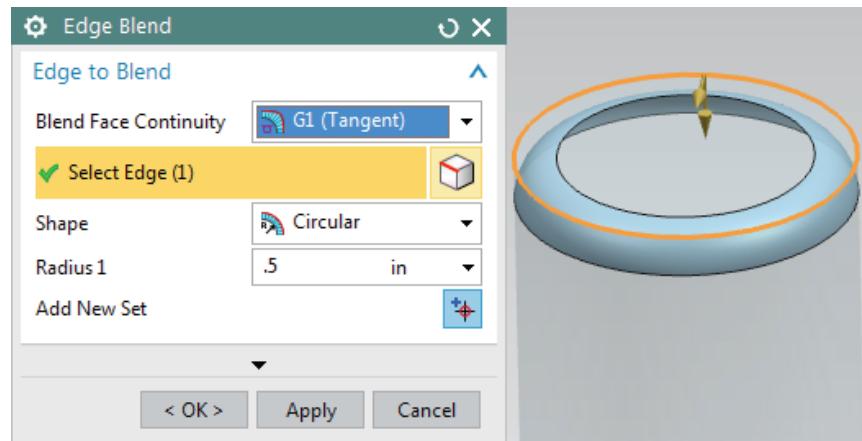
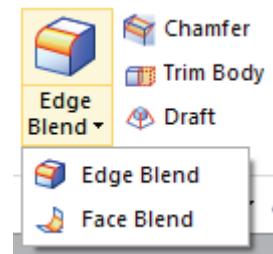
Note: *Pocket*, *Slot*, and *Groove* features are not commonly used in practice. All the models created using these features can be modeled using *2D Sketches* and *Extrude/Revolve*.

4.6 FEATURE OPERATIONS

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

4.6.1 Edge Blend

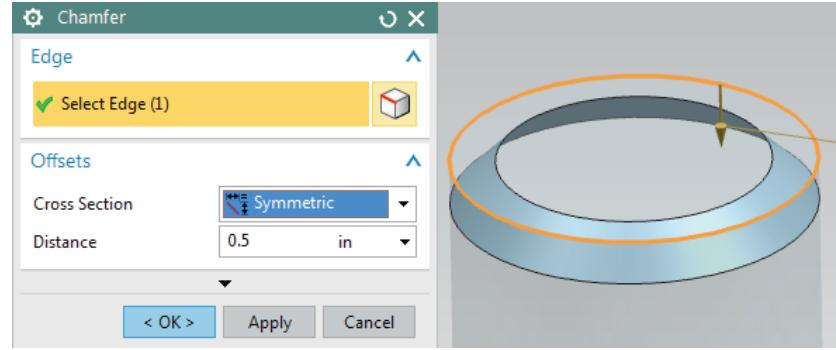
An *Edge Blend* is a radius blend that is tangent to the blended faces. This feature modifies a solid body by rounding selected edges. This command can be found under *Insert → Detail Feature → Edge Blend*. You can also click on its icon in the *Feature Group*. You need to select the edges to be blended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



Similar to *Edge Blend* you can also do a *Face Blend* by selecting two faces.

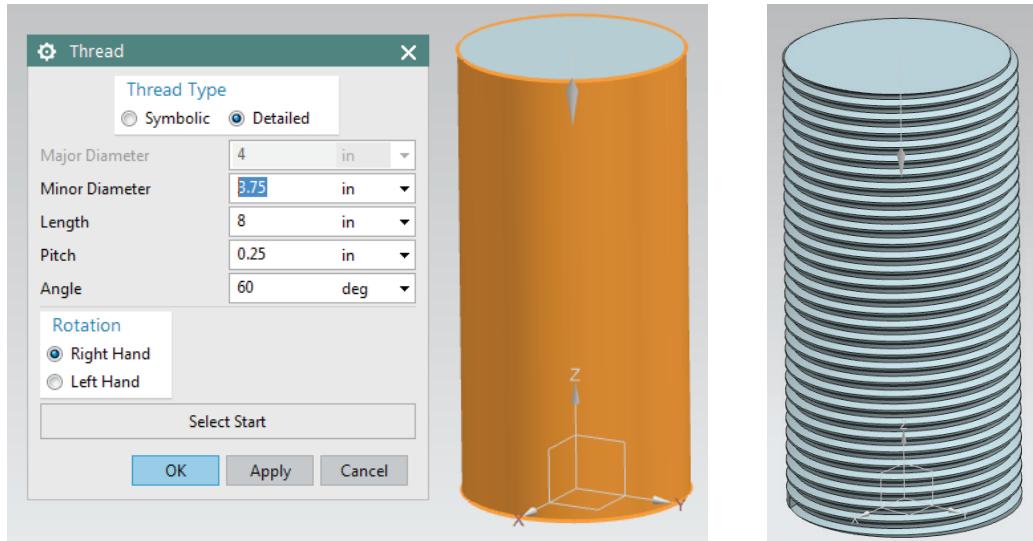
4.6.2 Chamfer

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 be found under *Insert → Detail Feature → Chamfer*. You can also click on its icon in the *Feature Group*. You need to select the edges to be chamfered and define the *Distance* of the *Chamfer* as shown below.

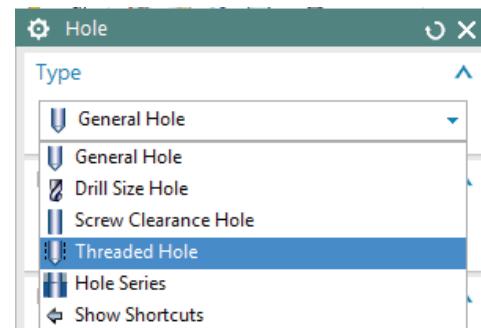


4.6.3 Thread

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 handed, external or internal, on cylindrical faces such as *Holes*, *Bosses*, or *Cylinders*. It also lets you select the method of creating the threads such as cut, rolled, milled or ground. You can create different types of threads such as metric, unified, acme and so on. To use this command, go to *Insert → Design Feature → Thread*. An example of a *Detailed Thread* is shown below.

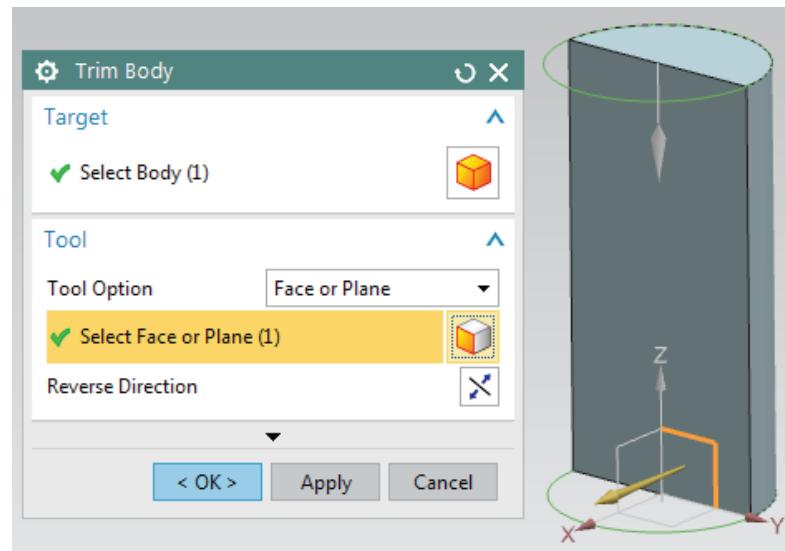


For *Threaded Holes*, it is recommended to use the *Threaded Hole* command instead of the *Thread* command: *Insert → Design Feature → Hole*



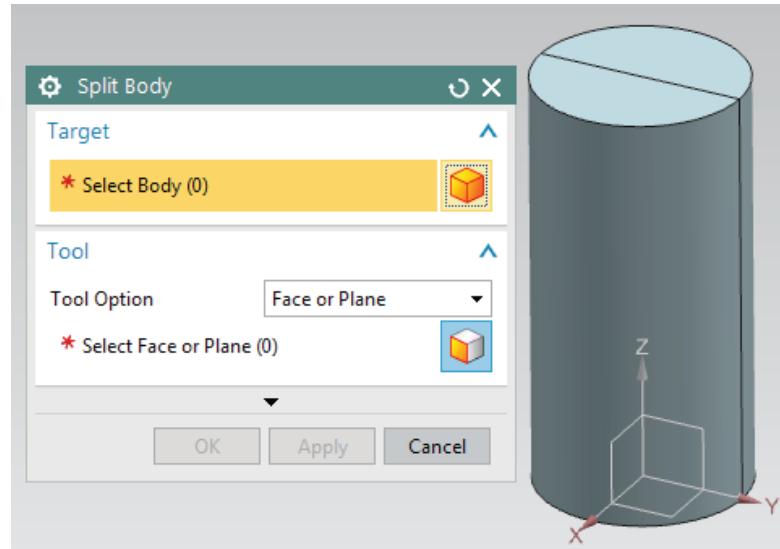
4.6.4 Trim Body

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



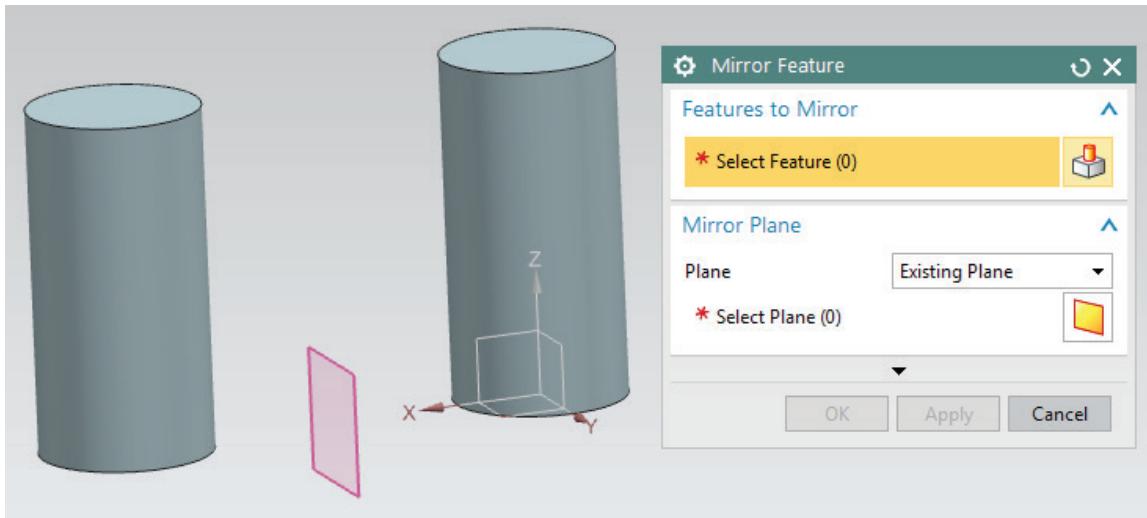
4.6.5 Split Body

A solid body can be split into two similar to trimming it. It can be done by a plane or a sheet body. To use this command, go to *Insert* → *Trim* → *Split Body* or click on its icon in the *Feature Group*. An example is shown on the right.



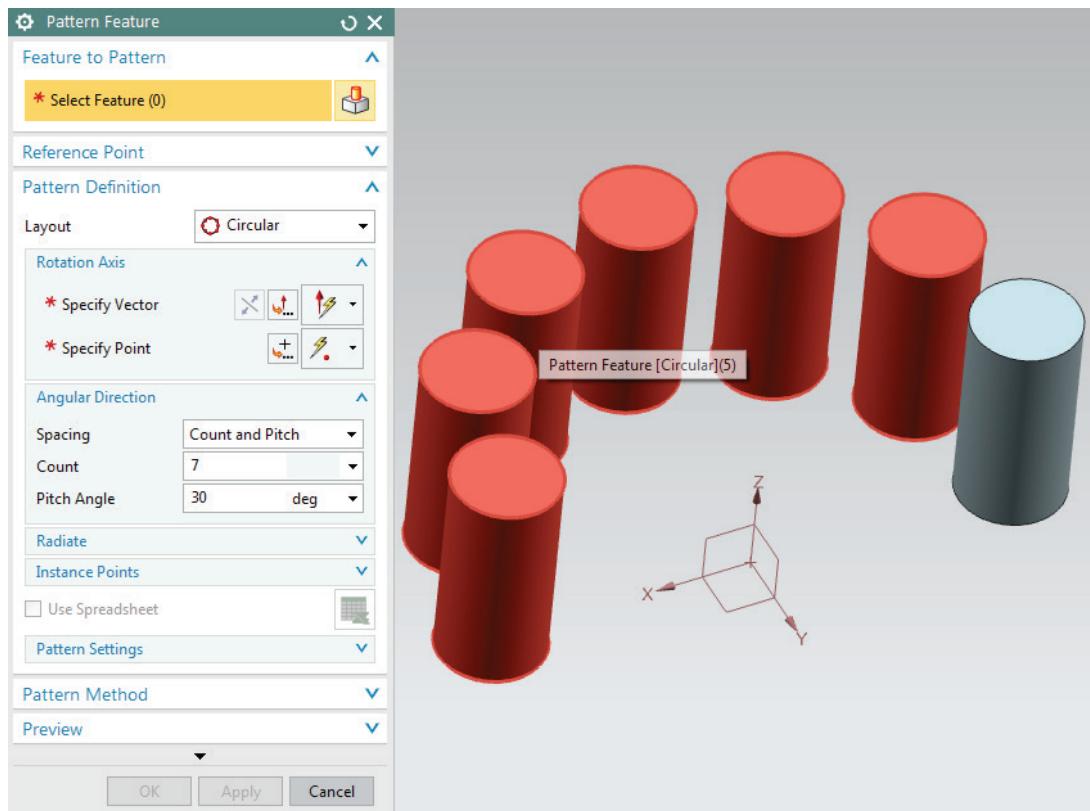
4.6.6 Mirror

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



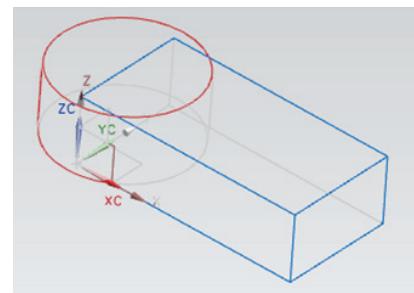
4.6.7 Pattern

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



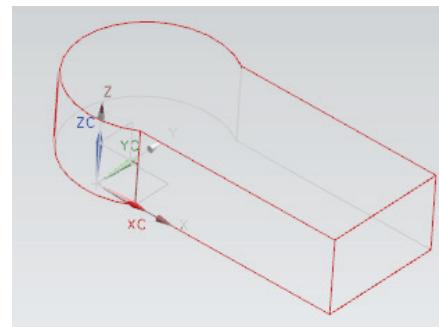
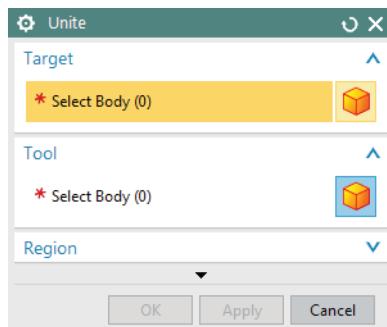
4.6.8 Boolean Operations

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



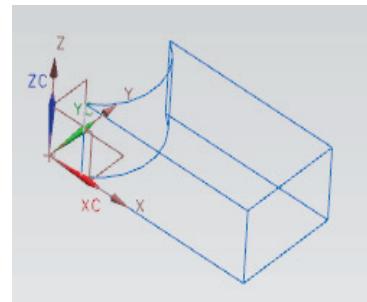
4.6.8.1 Unite

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



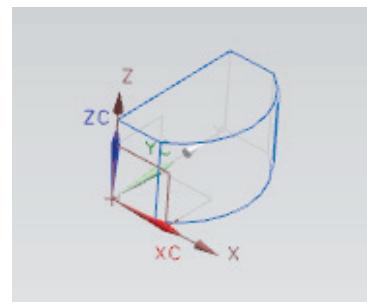
4.6.8.2 Subtract

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



4.6.8.3 Intersect

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

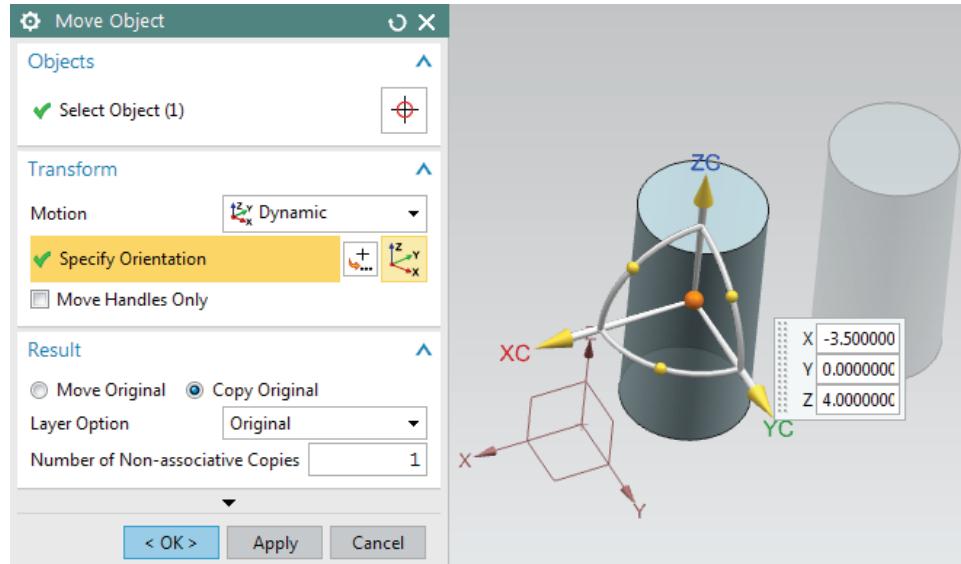


4.6.9 Move

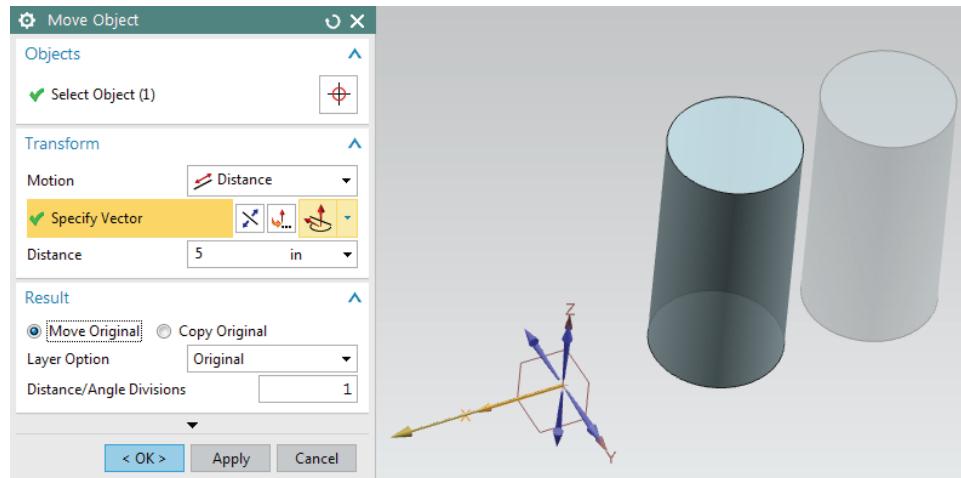
If you want to *Move* an object with respect to a fixed entity,

- Click on **Edit → Move Object**

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



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



- Click on **Specify Vector** and select the direction.
- Type **5** in the **Distance** box. This will translate the cylinder a distance of 5 inches along X-Axis
- Click **OK**

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*.

4.7 EXAMPLES

4.7.1 Hexagonal Screw

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

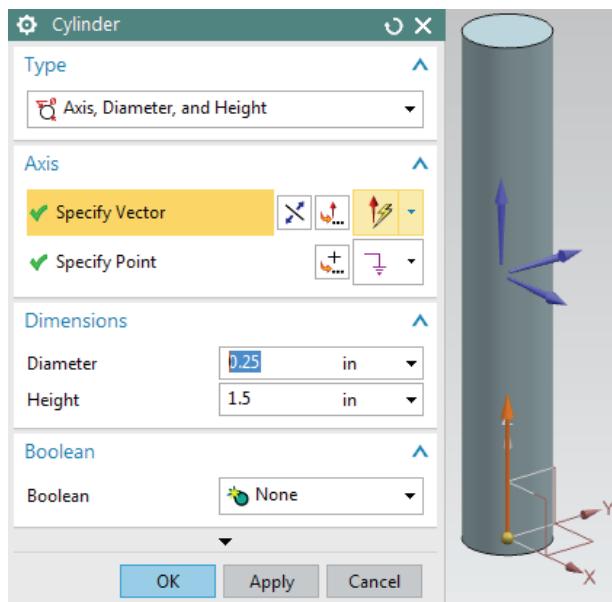
- Choose **Insert → Design Feature → Cylinder**

- The cylinder should be pointing in the **Positive ZC-Direction** with the center set at the **Origin** and with the following dimensions:

Diameter = **0.25** inches

Height = **1.5** inches

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



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

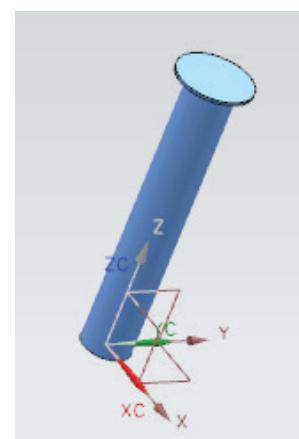
Diameter = **0.387** inches

Height = **0.0156** inches

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

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

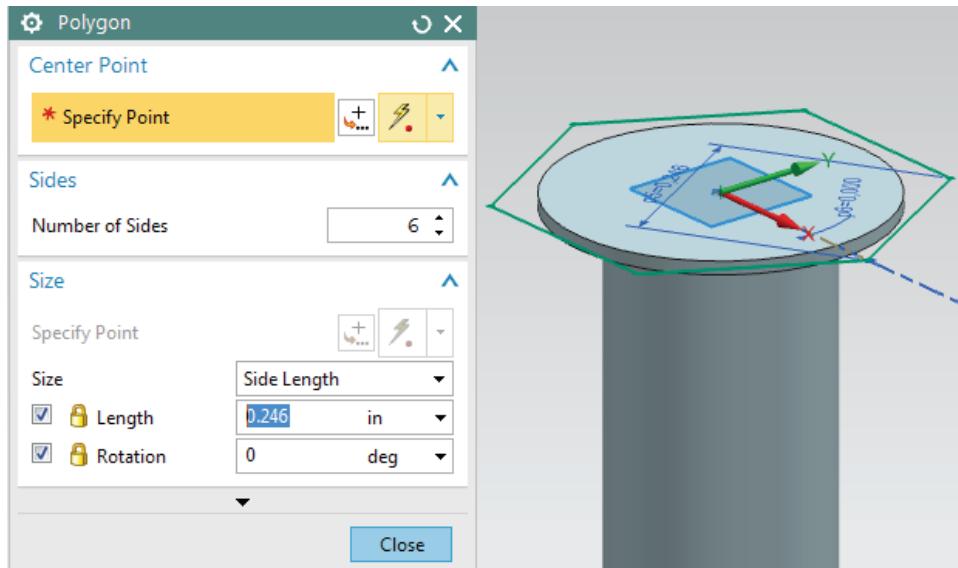
- Choose **Insert → Curve → Polygon**



- Select the center of the top circle as the **Center Point**
- On the **Sides** window, type **6** for the **Number of Sides**

There are three ways to draw the polygon.

- *Inscribed Radius*
 - *Circumscribed Radius*
 - *Side Length*
 - Choose **Side Length** and enter the following dimensions:
- Length = **0.246** inches
 Rotation = **0.00** degree
- Click **OK**

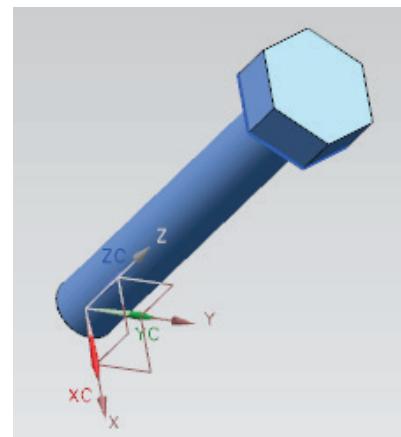


Now we will extrude this polygon.

- Choose **Insert → Design Feature → Extrude**
- Choose the **Hexagon** to be extruded
- Enter the **End Distance** as **0.1876** inches

The model looks like the following after extrusion.

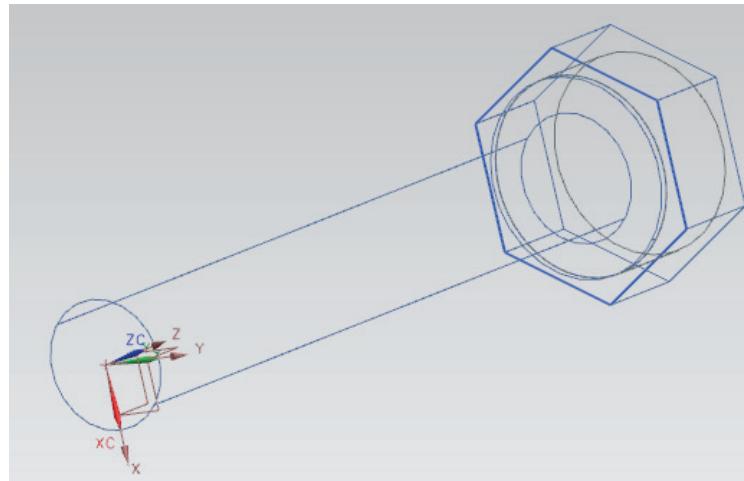
- On top of the cylinder that has a diameter of 0.387 inches, insert another cylinder with the following dimensions.



Diameter = **0.387** inches

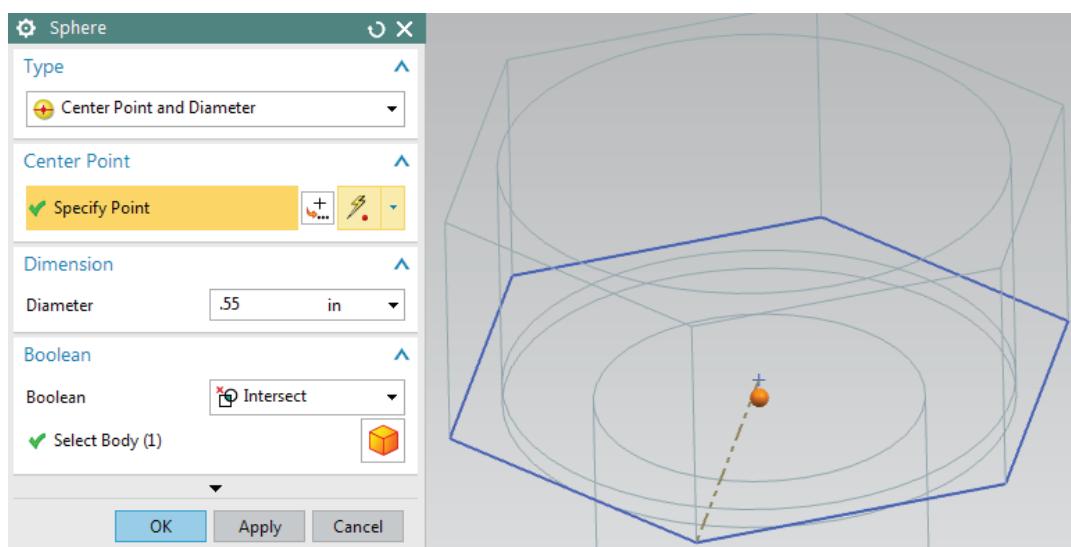
Height = **0.1875** inches

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



We will now use the feature operation *Intersect*.

- Choose **Insert → Design Feature →Sphere**
- Choose **Center Point and Diameter**
- 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



- Give **0.55** as the **Diameter**
- Choose **Intersect** in the **Boolean** dialog box

It will ask you to select the *Target Solid*

- Choose the hexagonal head
- Click **OK**

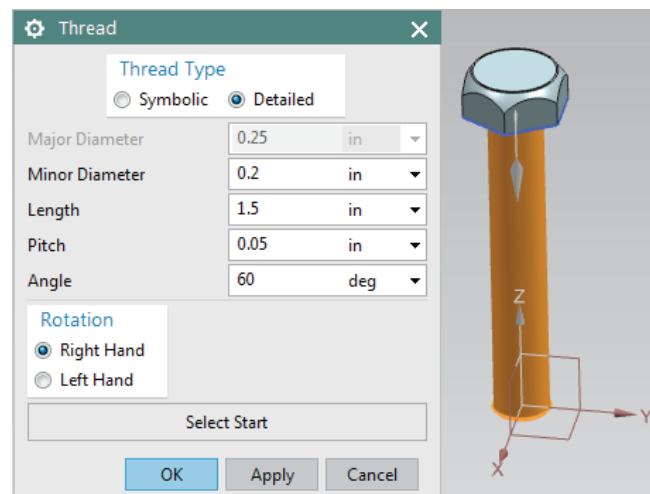
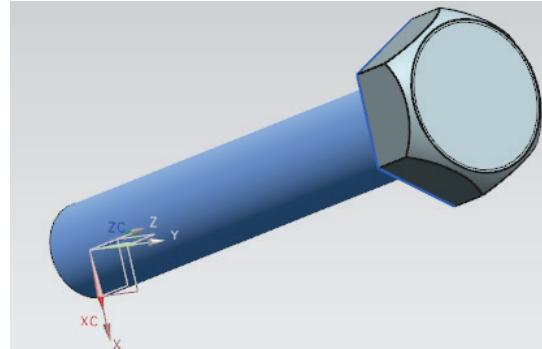
This will give you the hexagonal bolt as shown. Now we will add *Threading* to the hexagonal bolt.

- Choose **Insert → Design Feature → Thread**

- Click on the **Detailed** radio button
- Keep the **Rotation** to be **Right Hand**
- Click on the bolt shaft (the long cylinder below the hexagon head)

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

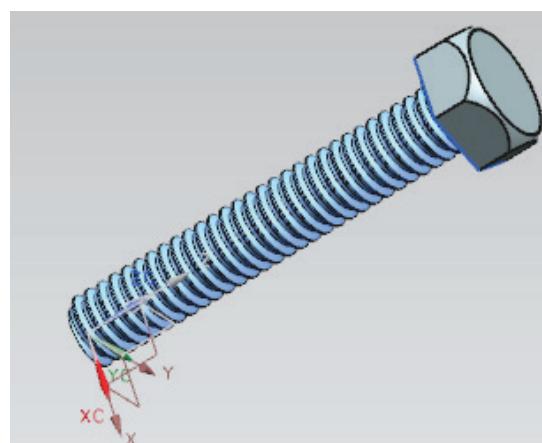
- Click **OK**



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

4.7.2 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 each side measuring **0.28685** inches and constructed at the **Origin**
- Choose **Insert → Design Feature → Extrude**

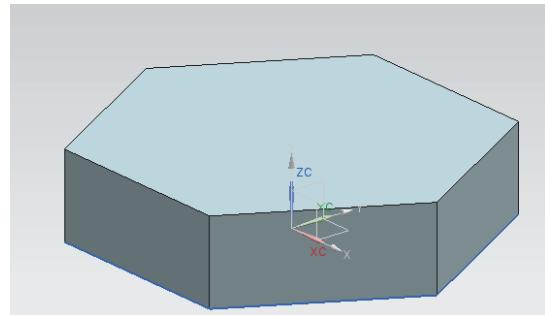


- Select the **Hexagon** to be extruded and enter the **End Distance** as **0.125** inches

The figure of the model is shown.

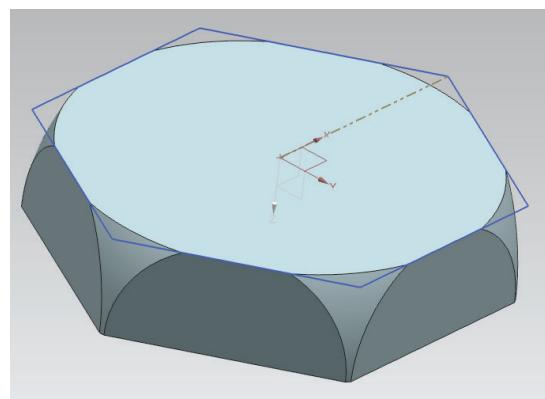
- Choose **Insert → Design Feature → Sphere**
- Enter the **Center Point** location in the **Point Dialog** window as follows

$$XC = 0; YC = 0; ZC = 0.125$$

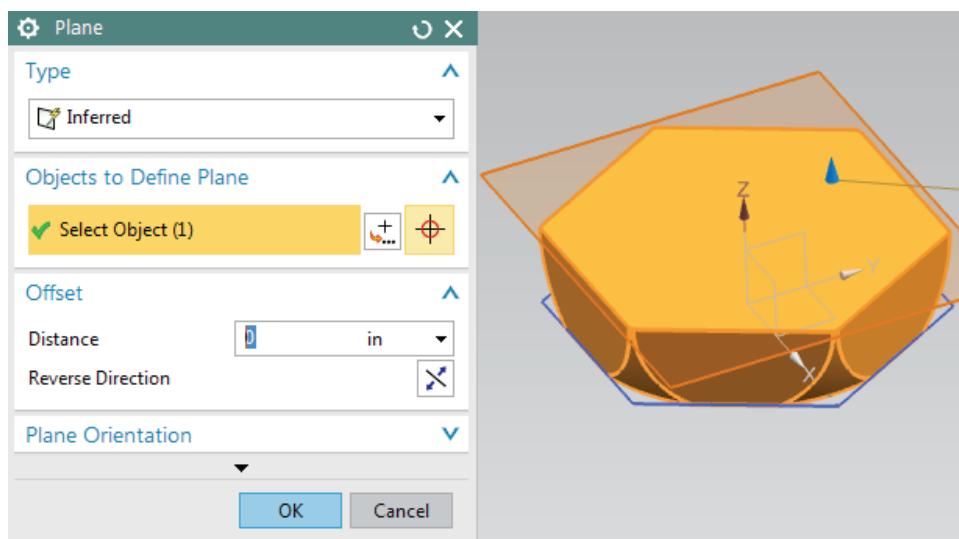


- Enter the **Diameter** value **0.57** inches
- In the **Boolean** operations dialog box select **Intersect** and click **OK**

The model will look like the following. We will now use a *Mirror* command to create the other side of the Nut.



- 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

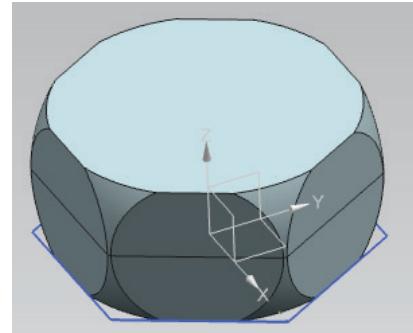


➤ Click on **OK**

➤ Click on **Copy**

➤ Click **Cancel**

You will get the following model.



➤ Choose **Insert → Combine Bodies → Unite**

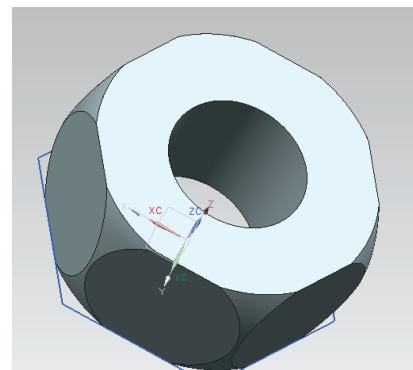
➤ Select the two halves and **Unite** them

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

Diameter = **0.25** inches

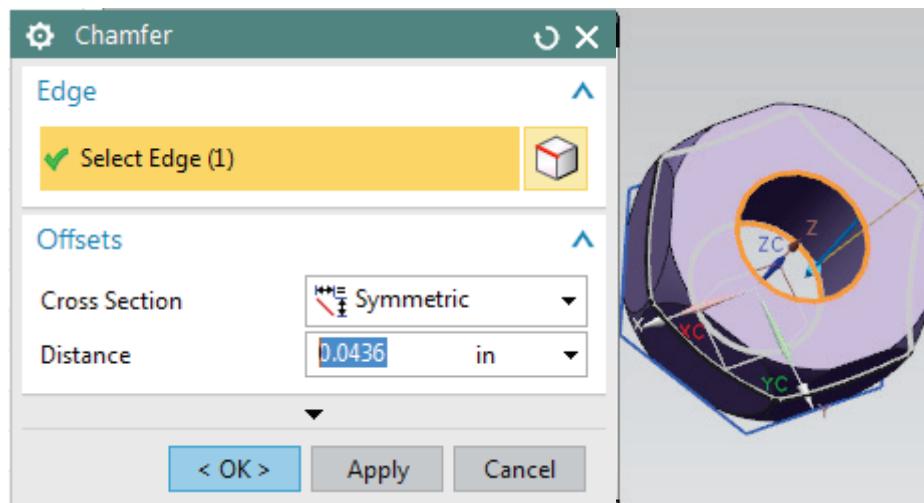
Height = **1** inch

➤ 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 **Distance** as **0.0436** inches and click **OK**

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

4.7.3 L-Bar

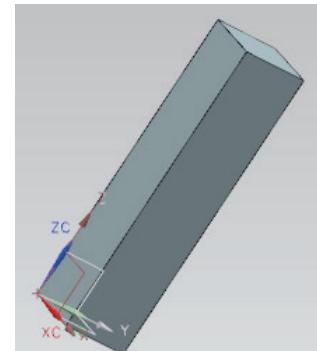
Here, we will make use of some *Primitives* and *Feature Operations* such as *Edge Blend*, *Chamfer*, and *Subtract*. It should be noted that the same model can be more easily created by *2D Sketching* and *Extruding*, but *Primitives* are used here to familiarize the users with these features.

- Create a new file and save it as **Arborpress_L-bar**
- Choose **Insert → Design Feature → Block**
- Create a **Block** with the following dimensions:

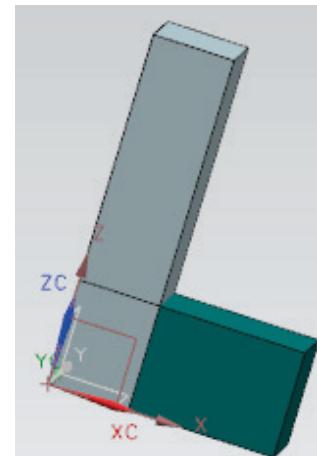
Length = **65** inches

Width = **65** inches

Height = **285** inches

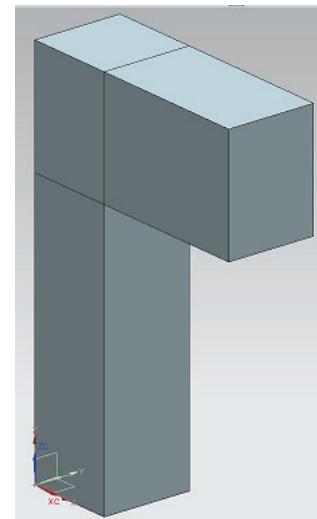


- Create the block at the **Origin**
- Create a second block also placed at the origin with the following dimensions:
 - Length = **182** inches
 - Width = **65** inches
 - Height = **85** inches



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

- Click **Edit → Move Object**
- Select the second block (green) and 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** button is checked and click **OK**
- Click **Move** and then **Cancel** on the next window so that the operation is not repeated



Now we will create a *Hole*. There are several 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 **Specify Vector**, select the **YC Axis** icon
- In the **Specify Point**, enter the following values:

$XC = 130$

$YC = -5$

$ZC = 242$

- 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

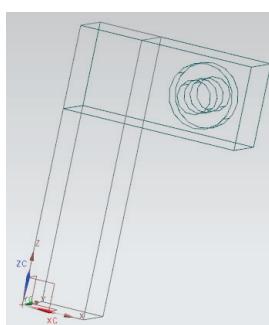
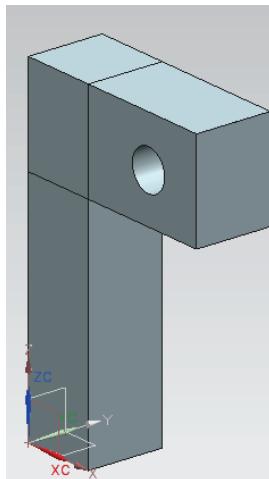
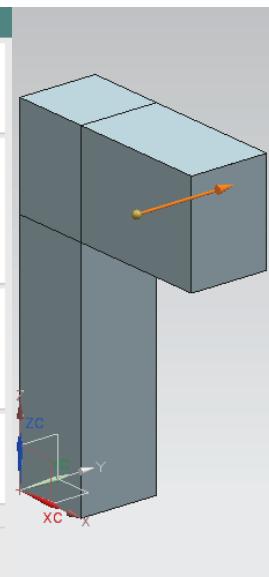
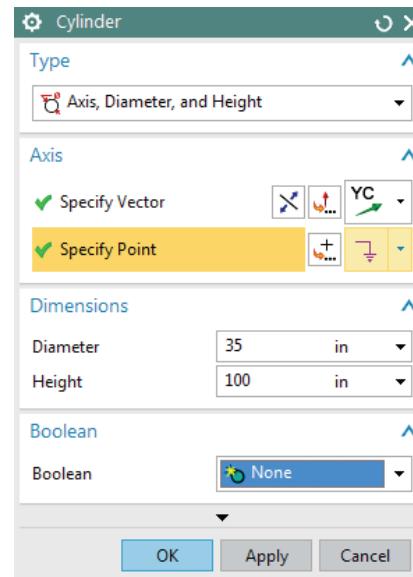
The hole should look like the one in the figure. Now we will create another cylinder and subtract it from the upper block.

The cylinder should be pointing in the positive *Y-direction* set at the following point: $XC = 130$; $YC = 22.5$ and $ZC = 242$ and 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

Now we will create a block.

- Choose **Insert →Design Feature →Block**
- Create a block with the following dimensions:



Length = **25** inches

Width = **20** inches

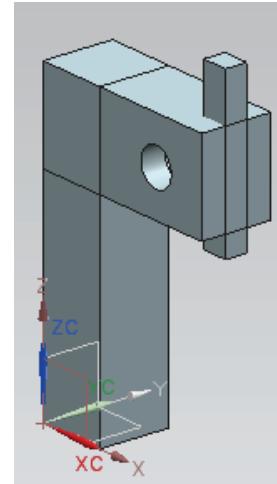
Height = **150** inches

- Click on the **Point Dialog** icon in the **Origin** box and enter the following values:

XC = **157**; YC = **22.5**; ZC = **180**

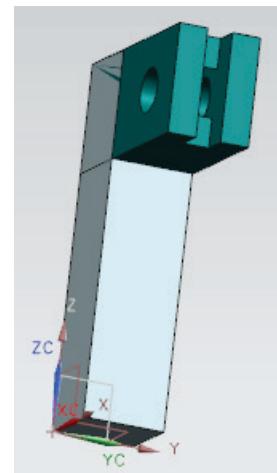
The model will look like the following figure. Now we will subtract this block from the block with the hole.

- Choose **Insert → Combine Bodies → Subtract**
- Click on the block with the two holes (green) as the **Target**
- Select the newly created block as **Tool**
- Click **OK**



The model will be seen as shown. Now we will use the *Blend* function in the *Feature Operations*. We must first unite the two blocks.

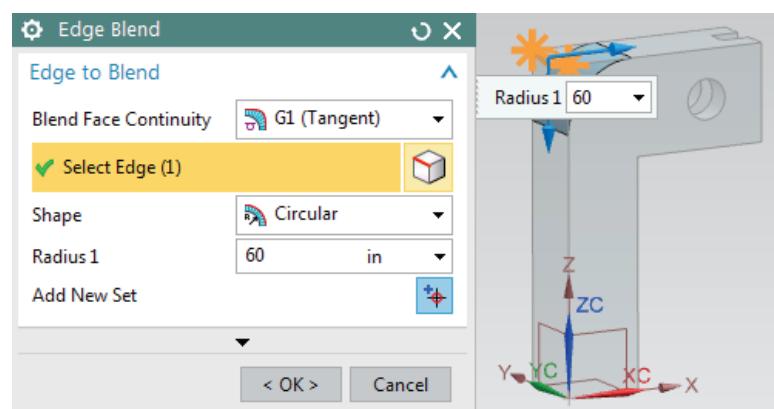
- Choose **Insert → Combine Bodies → Unite**
- Click on the two blocks and click **OK**



The two blocks are now combined into one solid model.

- Choose **Insert → Detail Feature → Edge Blend**
- Change the **Radius** to **60**
- Select the edge at the interface of the two blocks
- Click **OK**

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



We will now make four holes in the model. You can create these holes by using the *Hole* option. 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

- Construct them in the **XC**-direction at the following point coordinates:

Cylinder #1: X = **162**; Y = **11.25**; Z = **210**

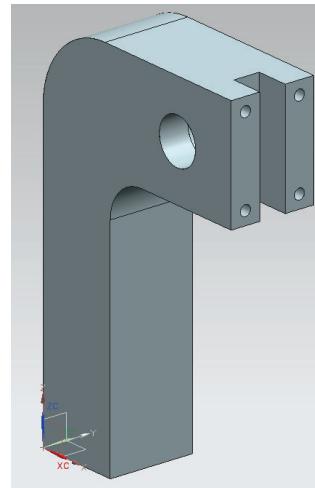
Cylinder #2: X = **162**; Y = **11.25**; Z = **275**

Cylinder #3: X = **162**; Y = **53.75**; Z = **210**

Cylinder #4: X = **162**; Y = **53.75**; Z = **275**

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

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

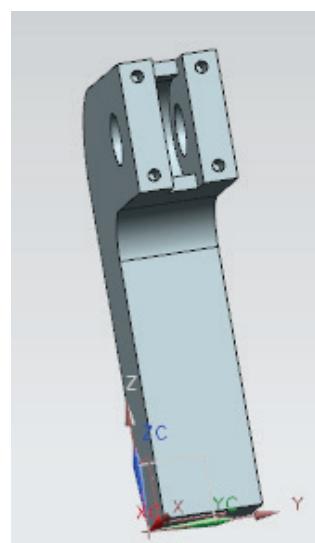
- Enter the following values in the **Point Dialog** as the **Origin** of the **Block**

XC = **130**

YC = **22.5**

ZC = **209.5**

- After creating the block, **subtract** this block from the block at the top



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

4.7.4 Rack

- Create a new part file and save it as **Arborpress_rack.prt**
- Right-click, then choose **Orient View → Isometric**
- Choose **Insert → Curve → Rectangle**

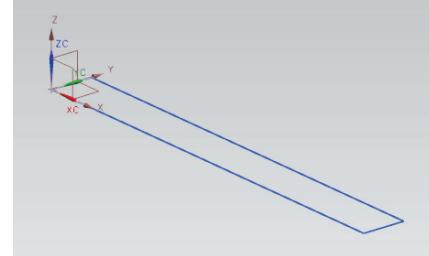
The *Point* window will open. Note the *Cue Line* instructions. The *Cue Line* provides the step that needs to be taken next. You need to define the corner points for the *Rectangle*.

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 **Cancel**
- Right-click on the screen and choose **FIT**



Note: We have three options for creating a rectangle:

- Two point
- Three points
- By center

The default option is *By 2 Points*.

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

OR

- Click on the **Extrude** icon on the **Form Feature** toolbar.

The *Extrude* dialog box will pop up.

- Click on the **Rectangle**.
- Choose the default **Positive ZC**-direction as the **Direction**
- In the **Limits** window, type in the following values:

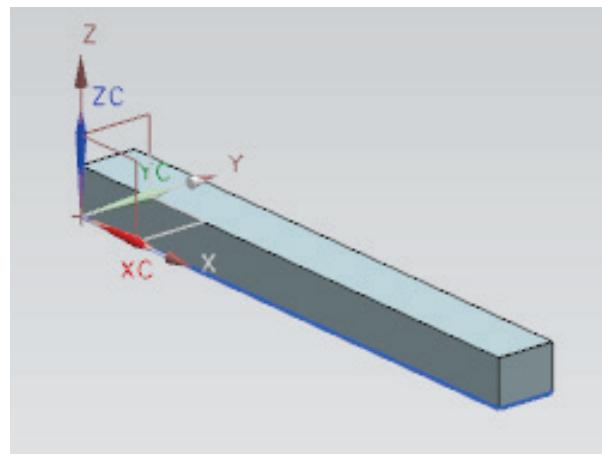
Start = **0**

End = **20**

- Click **OK**

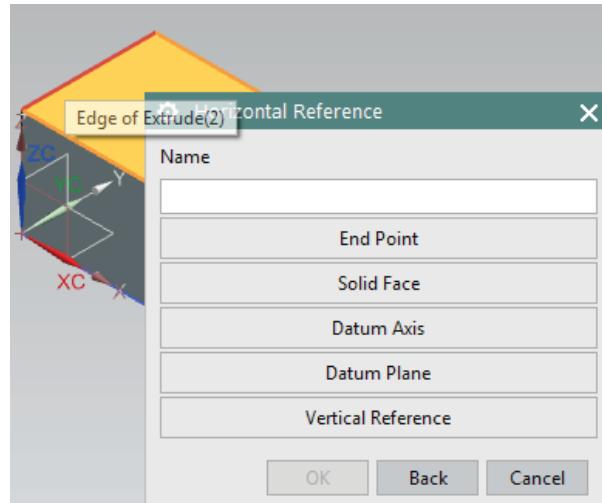
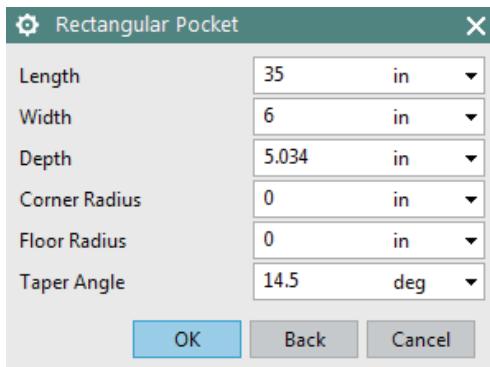
The extruded body will appear as shown below.

- Choose **Insert →Design Feature → Pocket**
- Choose **Rectangular** in the pop up window
- Click on the top surface of the rack
- Click on the edge as shown in the figure for the **Horizontal Reference**

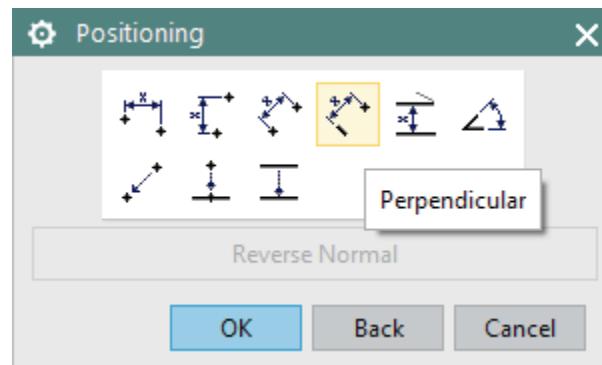


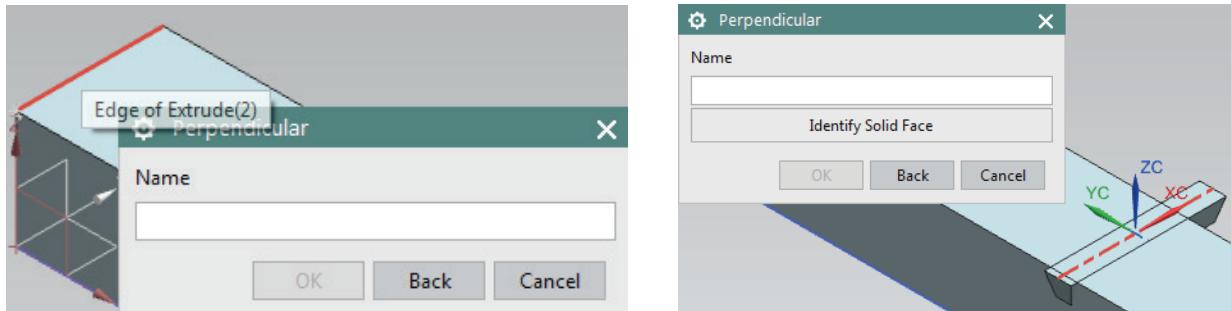
This will pop up the parameters window.

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



- When the **Positioning** window pops up, choose the **PERPENDICULAR** option
- Click on the edge of the solid and then click on the blue dotted line as shown below



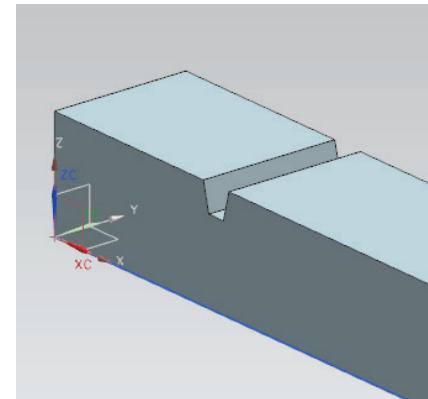
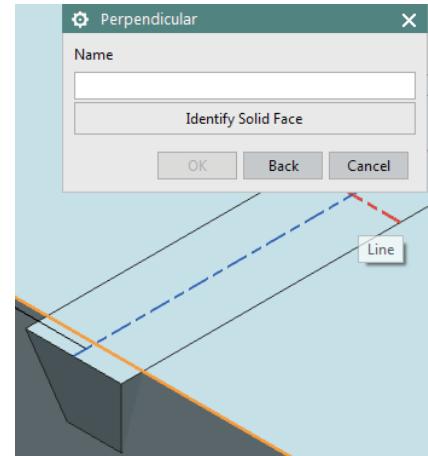


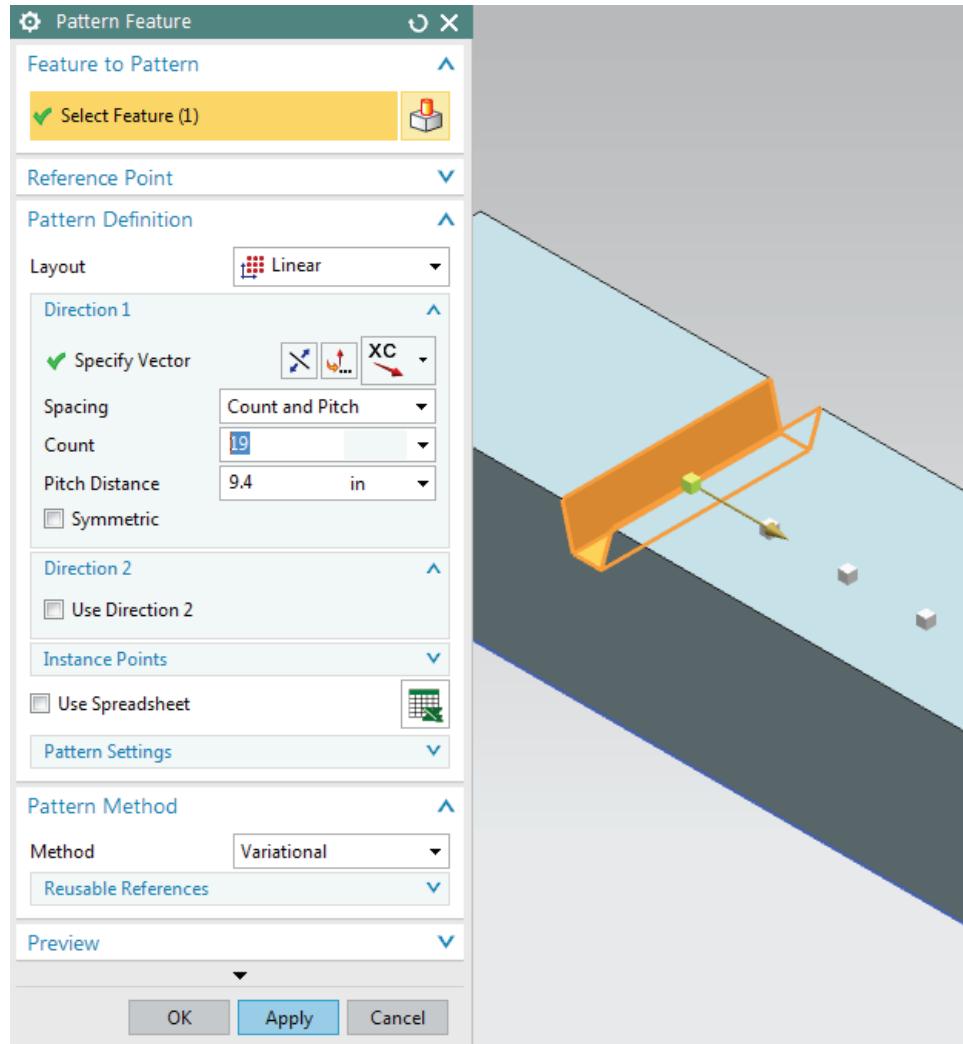
- Enter the **Expression** value as **37.8** and Choose **OK**
- Once again pick the **Perpendicular** option and then choose the other set of the edges along the Y-Axis, as shown on the right (the one perpendicular to the last blue line selected)
- Enter the expression value as **10** and click **OK**
- Click **OK** and then **Cancel**

The model will now look as follows.

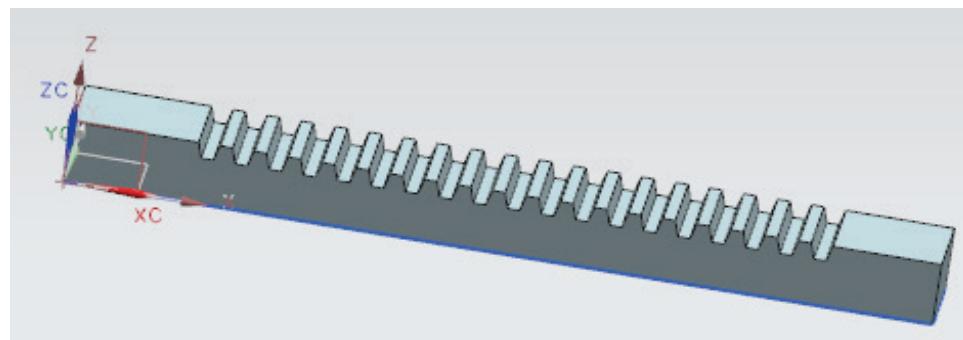
Let us create the instances of the slot as the teeth of the *Rack* to be meshed with *Pinion*.

- Click on **Pattern Feature** icon in the **Feature Group**
- Click on the 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**
- Click **OK**





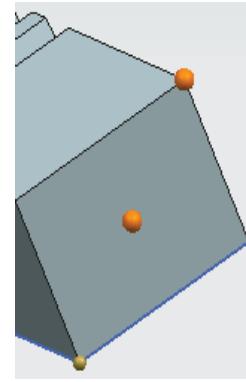
The model of the *Rack* will look as the one shown in the figure.



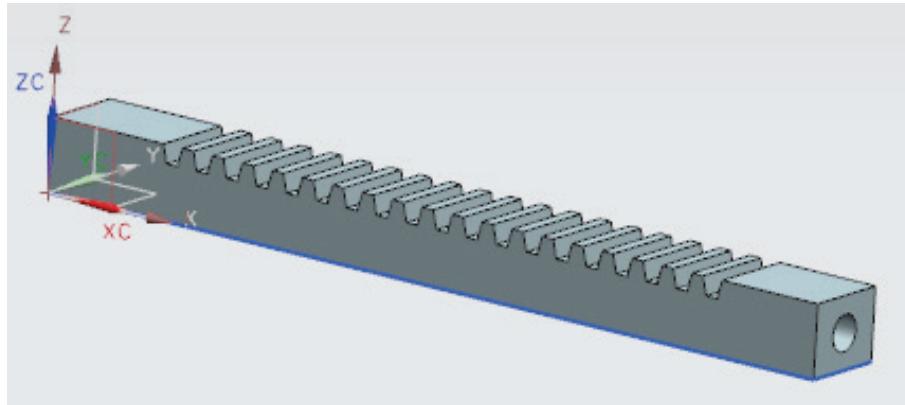
We will now create a *Hole* at the center of the rectangular cross section. To determine the center of the cross-section of the rectangular rack, we make use of the *Snap Points*

- Choose **Insert → Design Feature → Cylinder**

- Choose **-XC**-Direction in the **Specify Vector** dialog box
- Click on the **Point Dialog**
- In the **Points** dialog box select **Between Two Points** option and select the points as shown in the figure on the right (diagonally opposite points). The option selects the midpoint of the face for us
- Click **OK**
- Enter the following values in the **Dimension** dialog box
 - Diameter = **10** inches
 - Height = **20** inches
- Choose **Subtract** in the **Boolean** dialog box

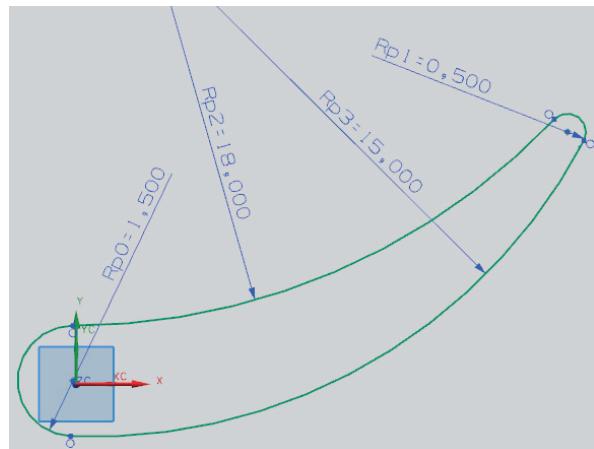


The final model is shown below. Save and close the model.



4.7.5 Impeller

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



Now let us model a cone.

- Choose **Insert → Design Feature → Cone**
- Select **Diameters and Height**
- Select the **-XC-Direction** in the **Specify Vector** dialog box
- In the **Point Dialog**, enter the coordinates **(14, 0, 0)**.
- Enter the following dimensions:
 - Base Diameter = **15** inches
 - Top Diameter = **8** inches
 - Height = **16.25** inches

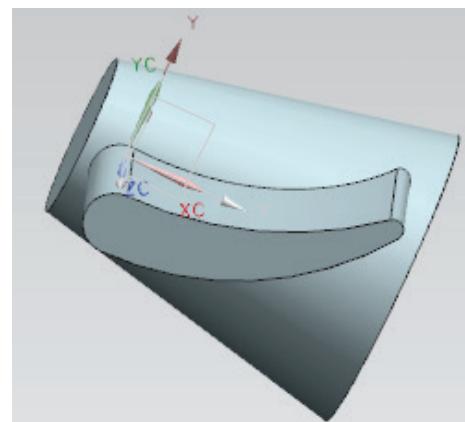
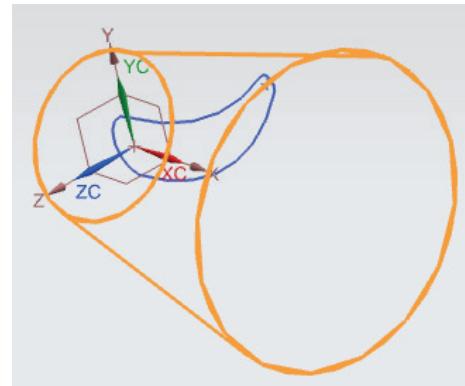
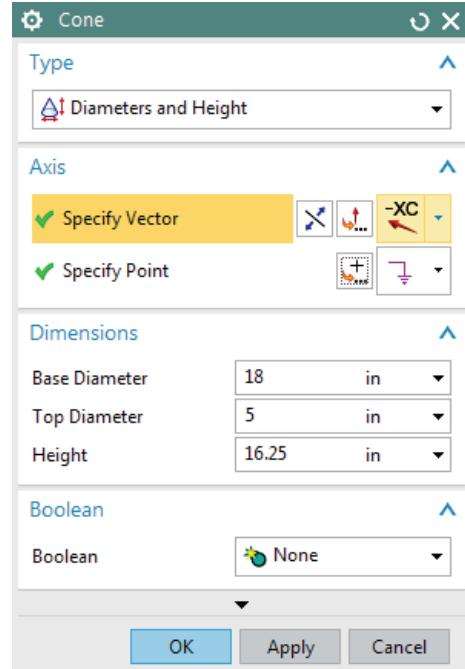
The cone will be seen as shown below if you choose *Static Wireframe View*.

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

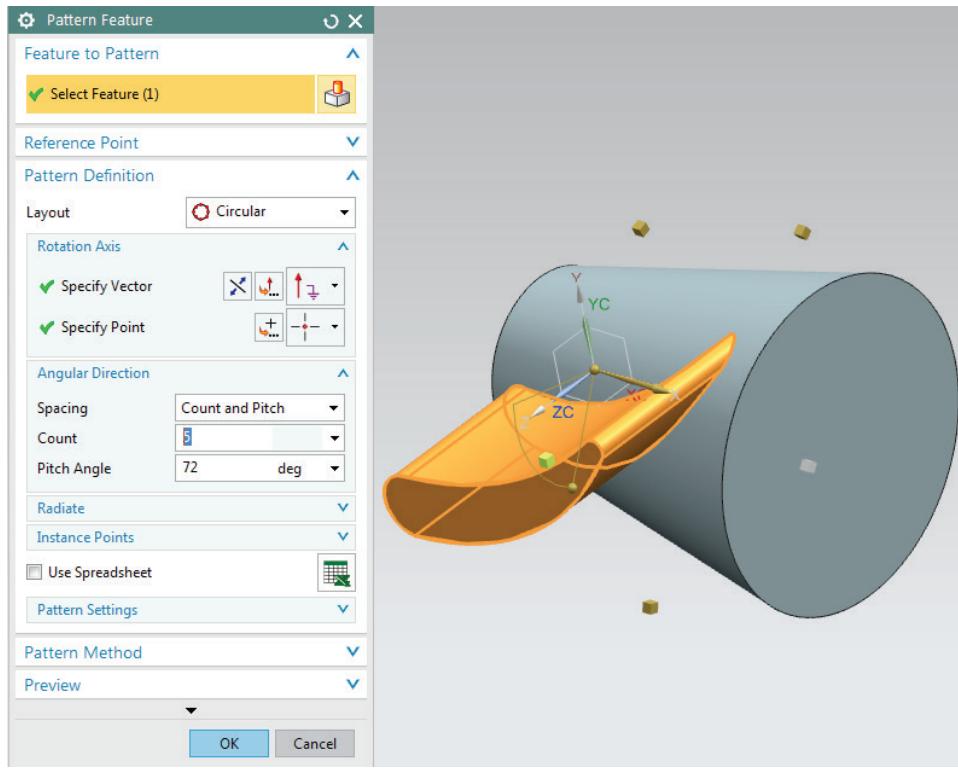
The model will be as follows.

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

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



➤ Click **OK**

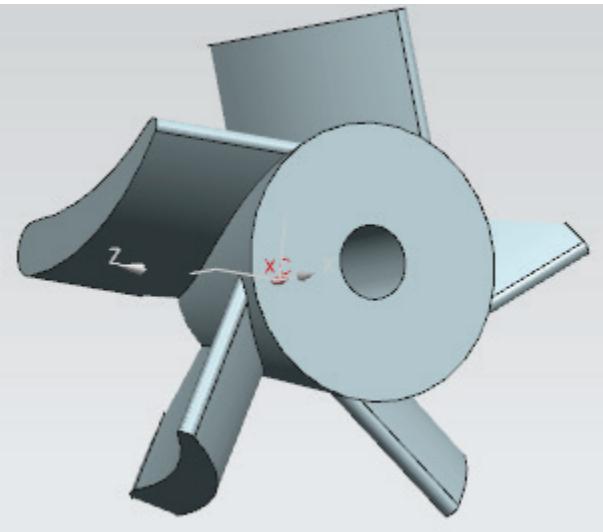


Now, let us create two holes in the cone for the shaft and the locking pin. Note that these holes can also be created by *Hole* menu option.

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

The final model will look like the following.

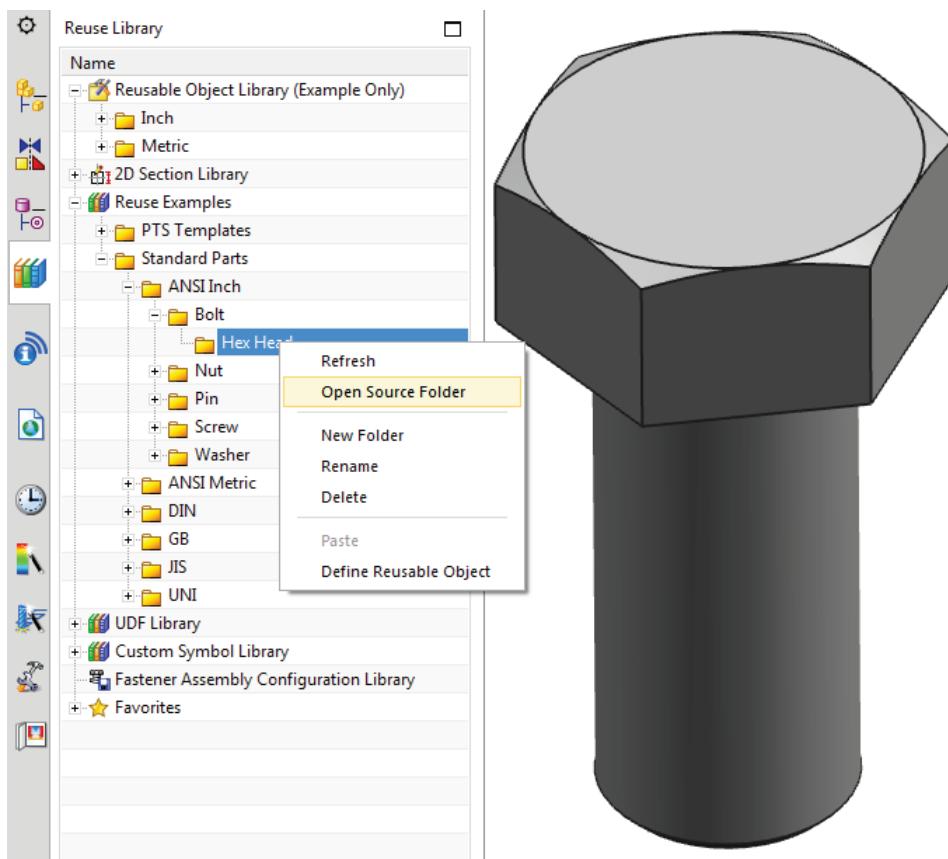
Save and close your work.



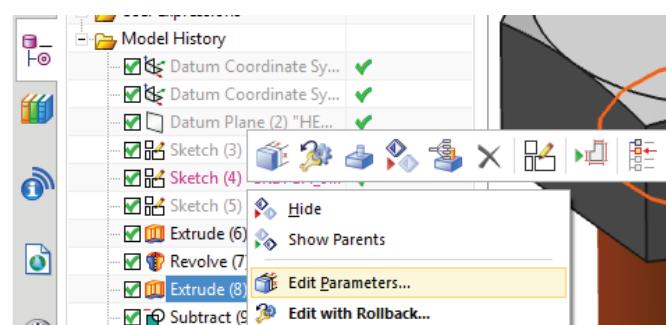
4.8 STANDARD PARTS LIBRARY

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

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



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



4.9 SYNCHRONOUS TECHNOLOGY

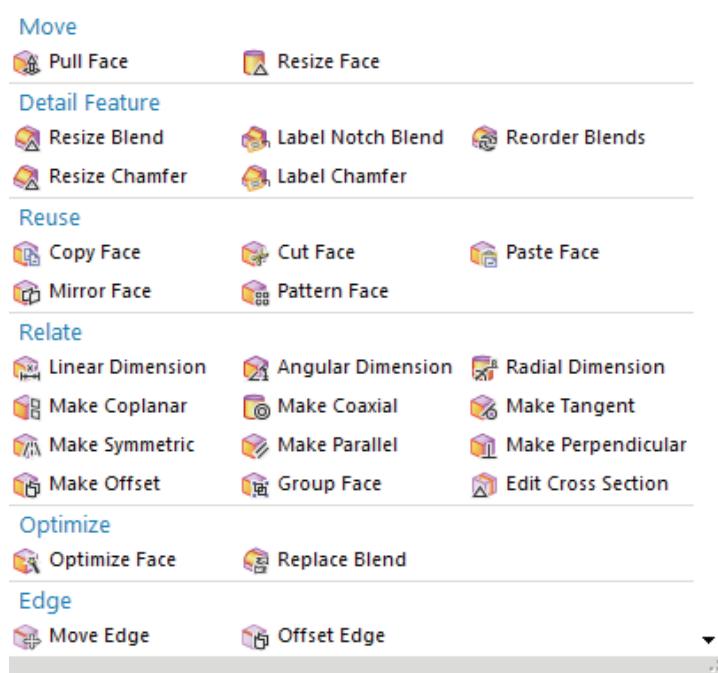
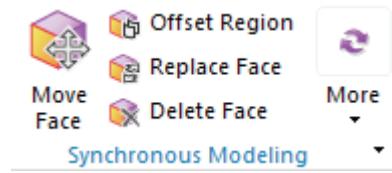
One of the important and unique features which NX offers apart from *Design Features* and *Freeform Modeling* is *Synchronous Technology*. With the options available in *Synchronous Modeling* group in the ribbon bar in the *Modeling Application* tab, the user can modify complex 3D models without the model history tree and without knowing the feature relationships and dependencies. The “push-and-pull” options can be used to modify the 3D model using faces, edges and cross-sections. NX 10 supports the *Synchronous Modeling* to work with 3D models from CATIA, Pro/ENGINEER®, SolidWorks®, and Autodesk Inventor®, apart from the standard formats including IGES, ISO/STEP and JT.

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

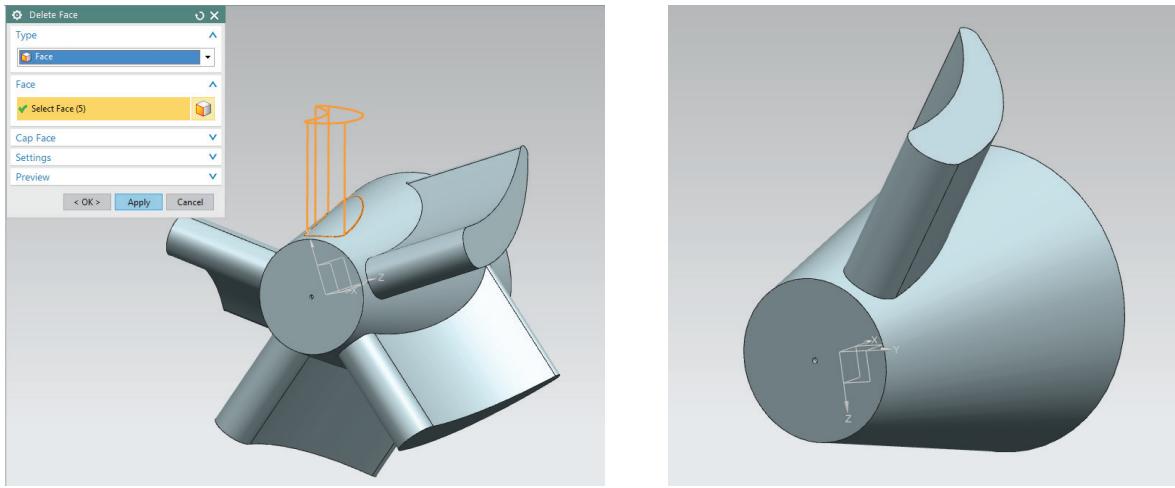
- Open a new file in NX
- Choose **File → Import → Impeller_impeller.stp**

Observe here that the .stp file would not have any model history.

We will explore some of the options available in the *Synchronous Modeling* group in the ribbon bar. Click *More* to view a comprehensive list of options available in synchronous modeling.

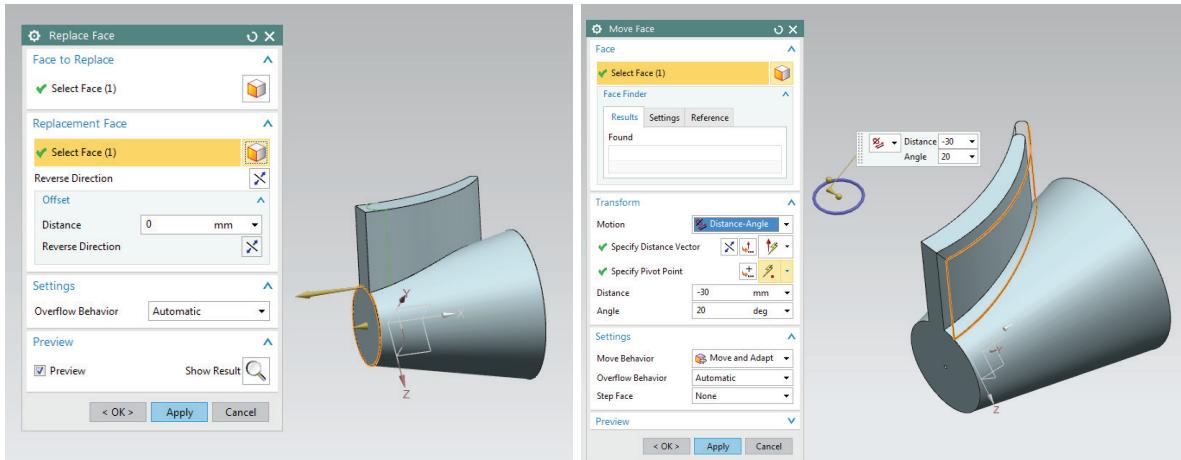


- Click **Delete Face** and select the faces of the blade to delete the blade
- Repeat the process and delete all except one blade. The part should look as shown below.

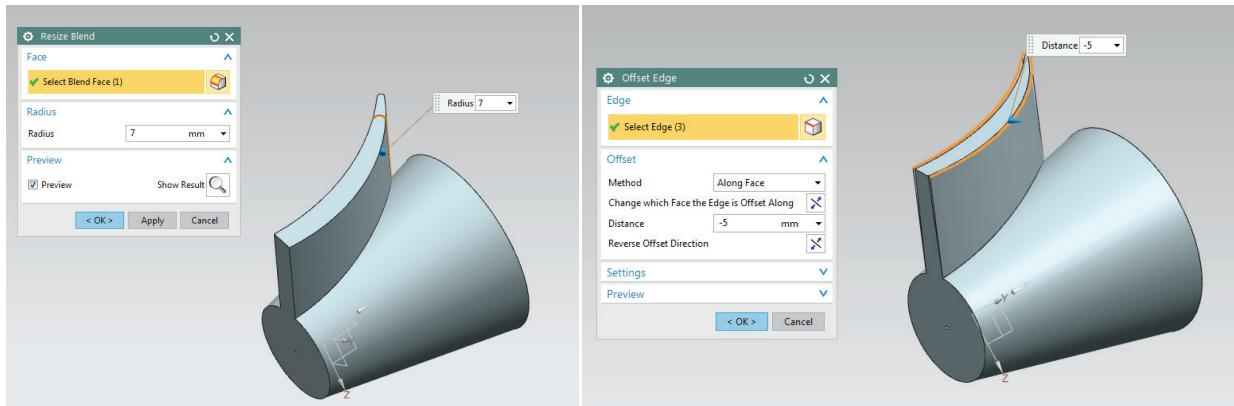


- Click **Replace Face** and select the end face of the blade with large blend radius as **Face to Replace** and select the flat surface of the cone with smaller diameter as the **Replacement Face** to delete the blade.

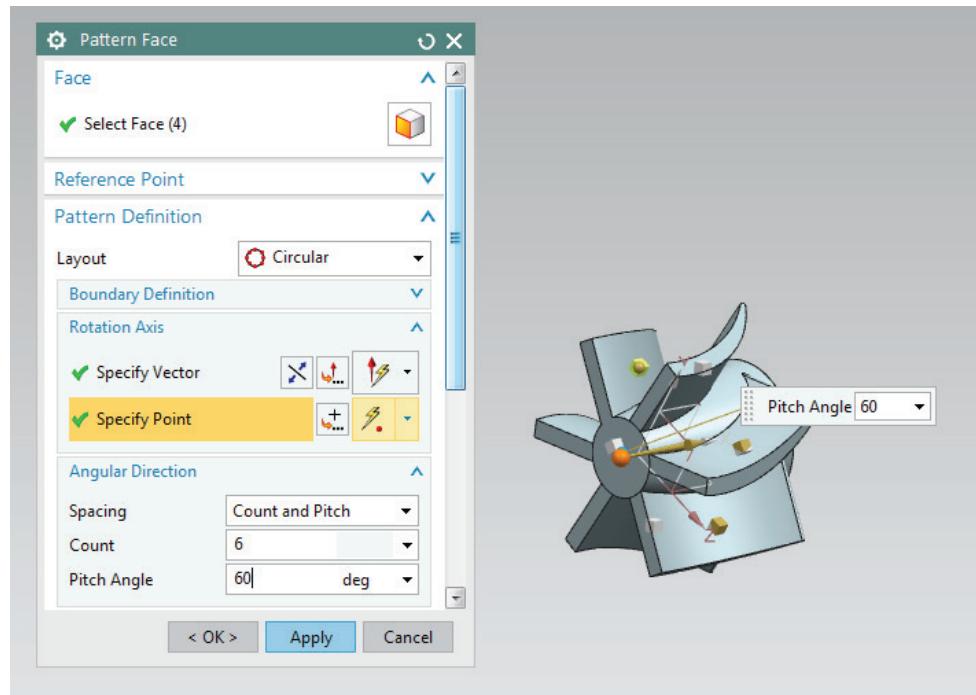
The part should look as shown below.



- Click **Move Face** and select one side of the blade and enter distance **-30** and angle **20** in the transform section
- Click **Resize Blend** and select the blended surface of the blade and enter radius as **7 mm** to sharpen the end



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



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

4.10 EXERCISES

4.10.1 Circular Base

Model a circle base as shown below using the following dimensions:

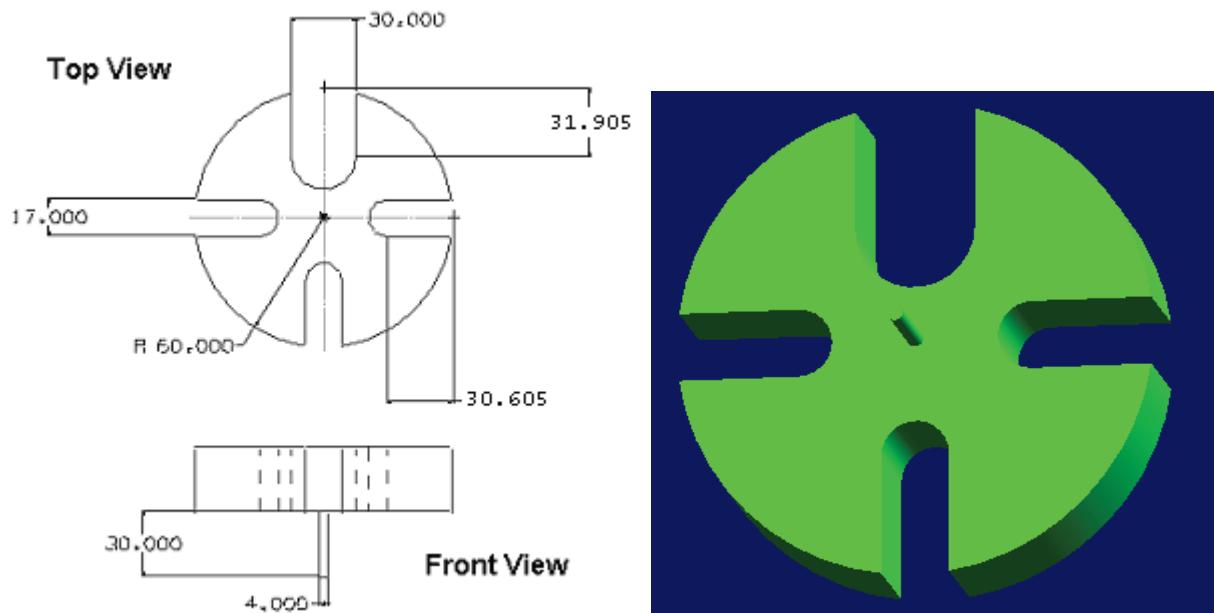
Outer diameter = 120 inches

Distance of 3 small slots = 17 inches

Distance of the large slot = 30 inches

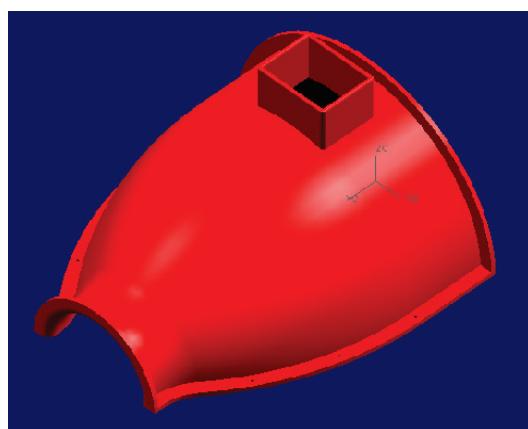
Diameter of the central rod = 4 inches and length = 30 inches

Length of slots may vary.



4.10.2 Impeller Upper Casing

Model the upper casing of the *Impeller* as shown below.



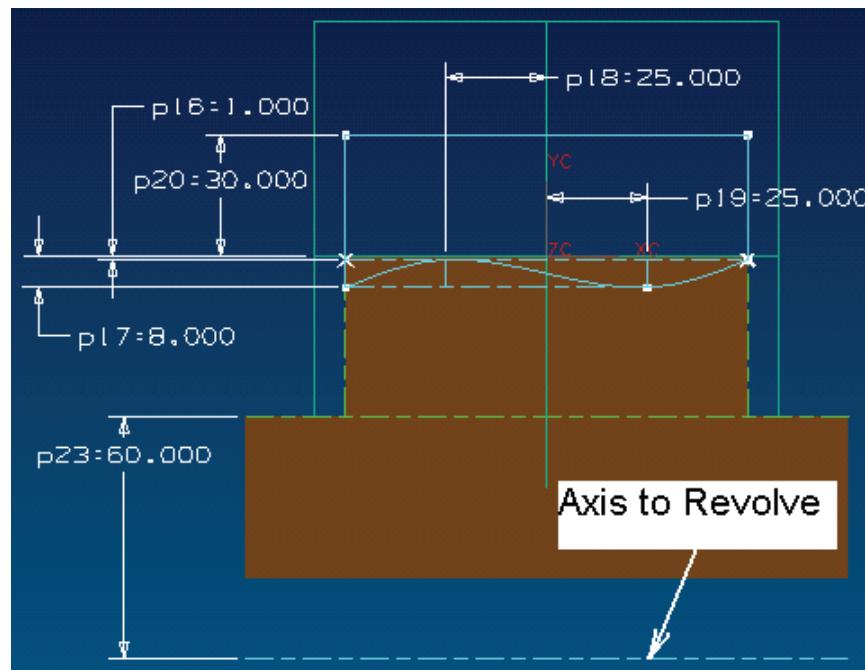
The dimensions of the upper casing are the same as for the lower casing, which is described in the previous exercise in detail. The dimensions for the manhole should be such that impeller blades can be seen and a hand can fit inside to clean the impeller.

4.10.3 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.

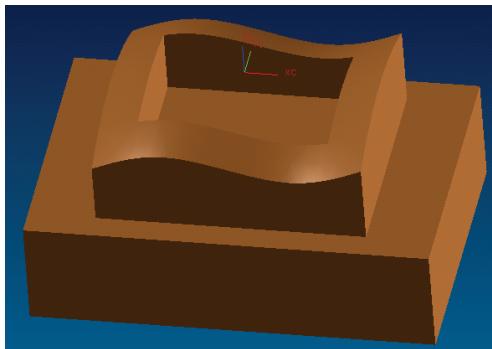
Create and *Unite* another block over the first one with 100, 80 and 40 along X, Y and Z and centrally located to the previous block.

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.

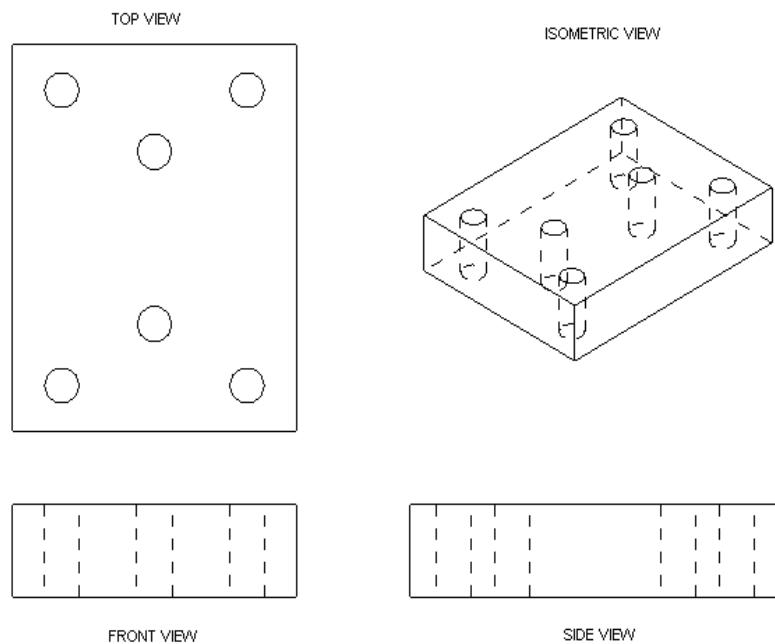


CHAPTER 5 – DRAFTING

The NX 10 *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 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.



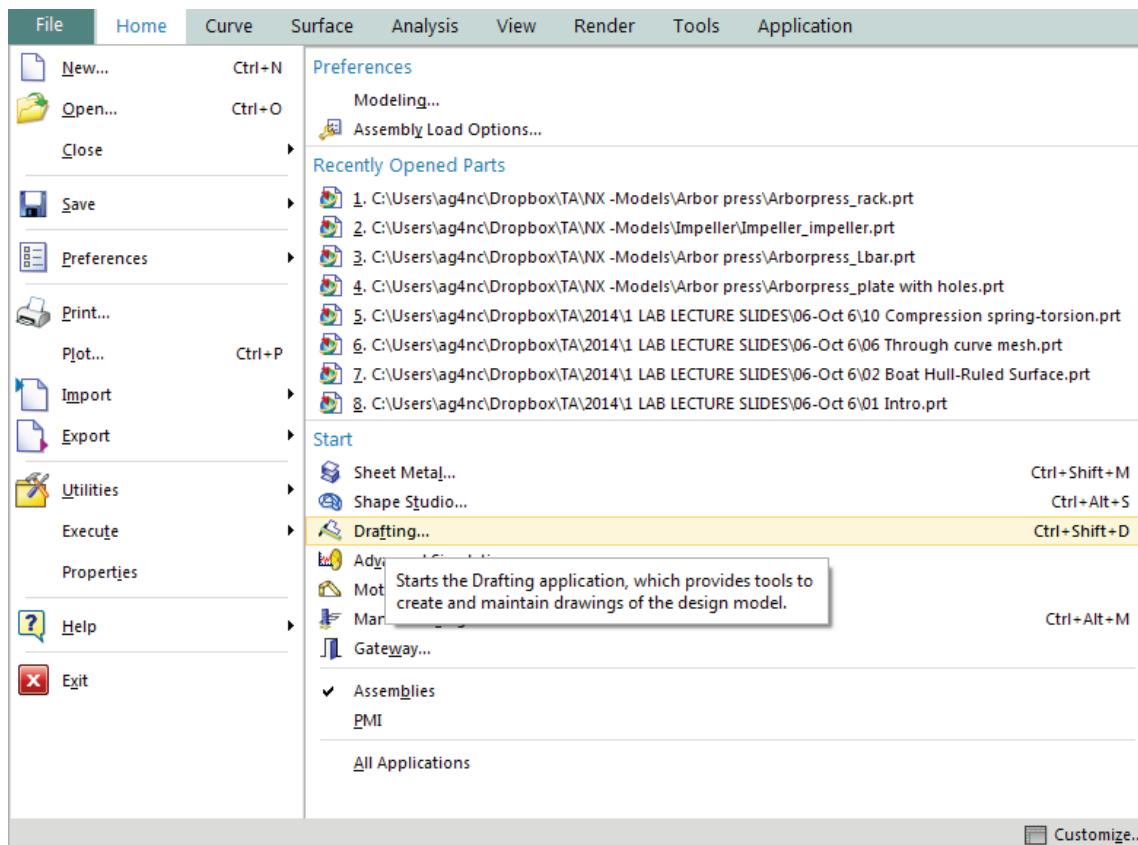
Some of the useful features of the *Drafting Application* are:

- 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 CREATING A DRAFTING

- Open the file **Arborpress_rack.prt**
- From the NX 10 Interface, choose **File → Drafting** as shown or choose **Application** tab and select **Drafting**



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 do not fit into a standard sized sheet.

Preview

This shows the overall design of the *Template*.

Units

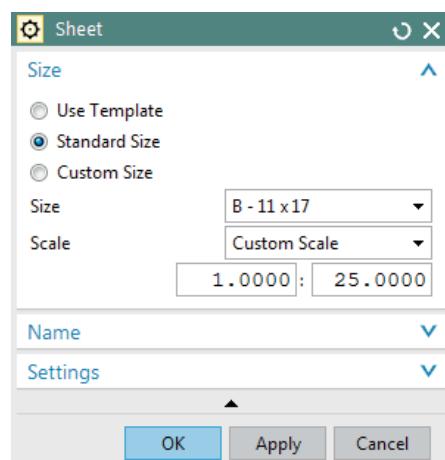
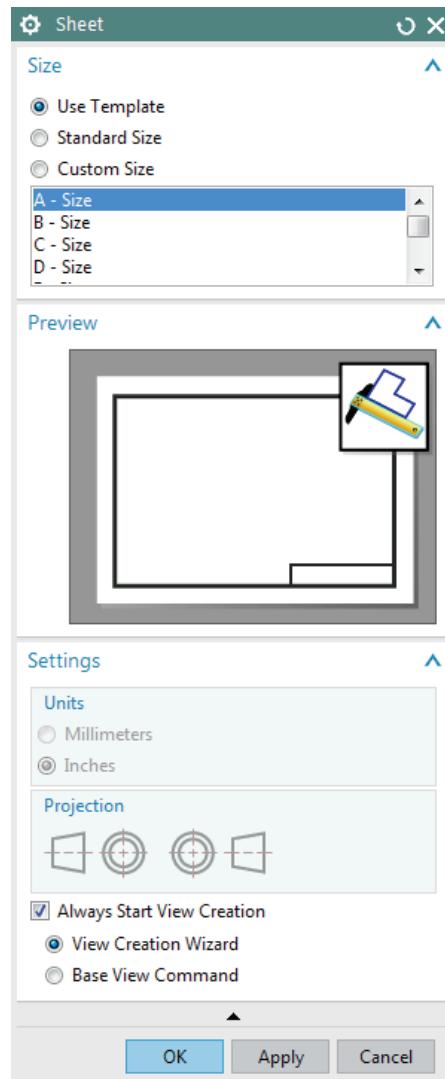
Units follow the default units of the parent 3-D model. In case you are starting from the *Drafting Application* you need to choose the units here.

Projection

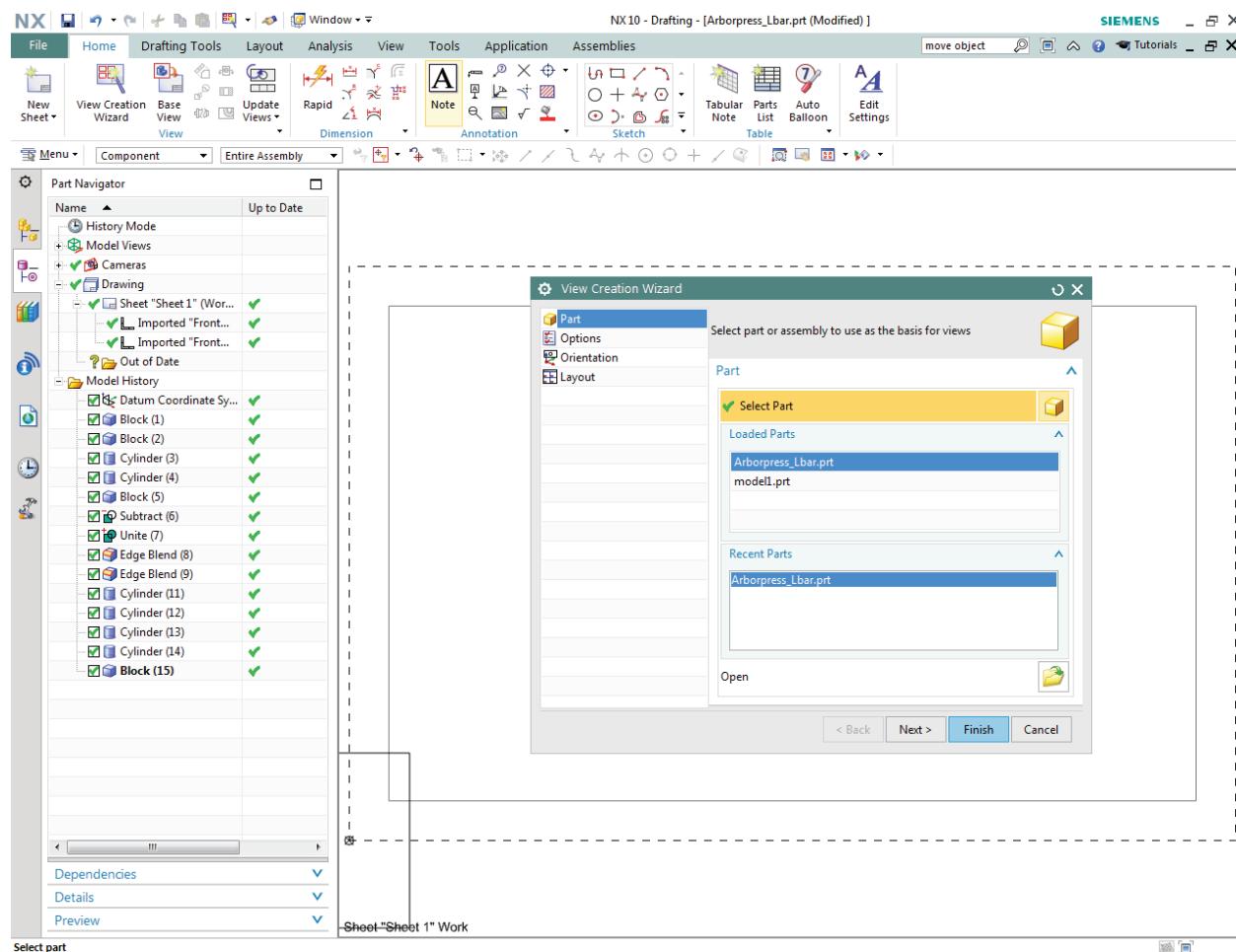
You can choose the *Projection Method* either *First Angle* or *Third Angle* method.

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**
- Change the **Scale** to **1:25** by using the drop-down menu and choosing the **Custom Scale** under the **Scale**
- Click **OK**

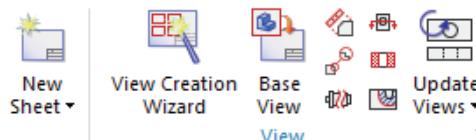


This will open the *Drafting Application* and the following screen will be seen as below. Let us first look at the *Drafting Application Interface*.



You will see a dialog box pops-up which will help you choose the parts, views and other options.

- Change the options and views and click **Finish**
- Choose **Insert → View → Base** or click on **Base View** in the **View Group**



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

- Choose the **View** to be **Front**

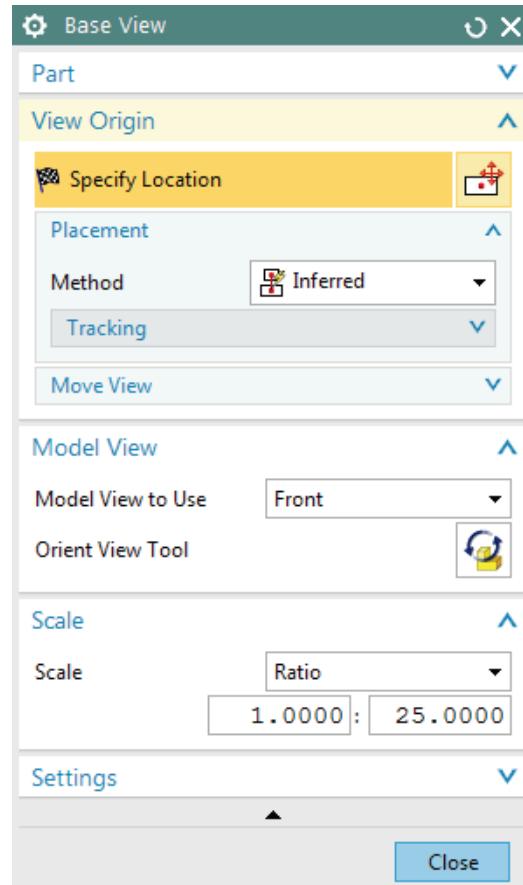
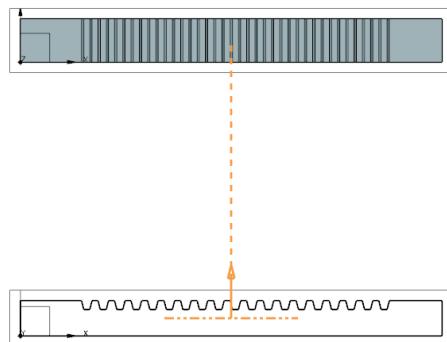
You can find the *Front View* projection on the screen. You can move the mouse cursor on the screen and click on the place where you want the view.

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*.

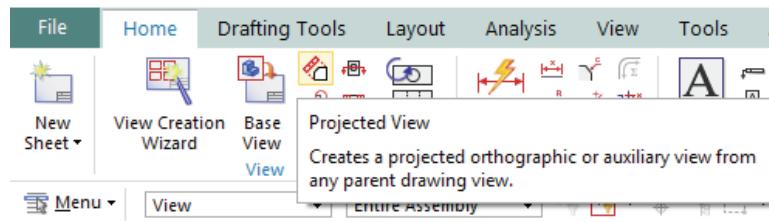
You can find different views by moving the cursor around the first view. If you want to add any orthographic views after closing this file or changing to other command modes

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

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



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



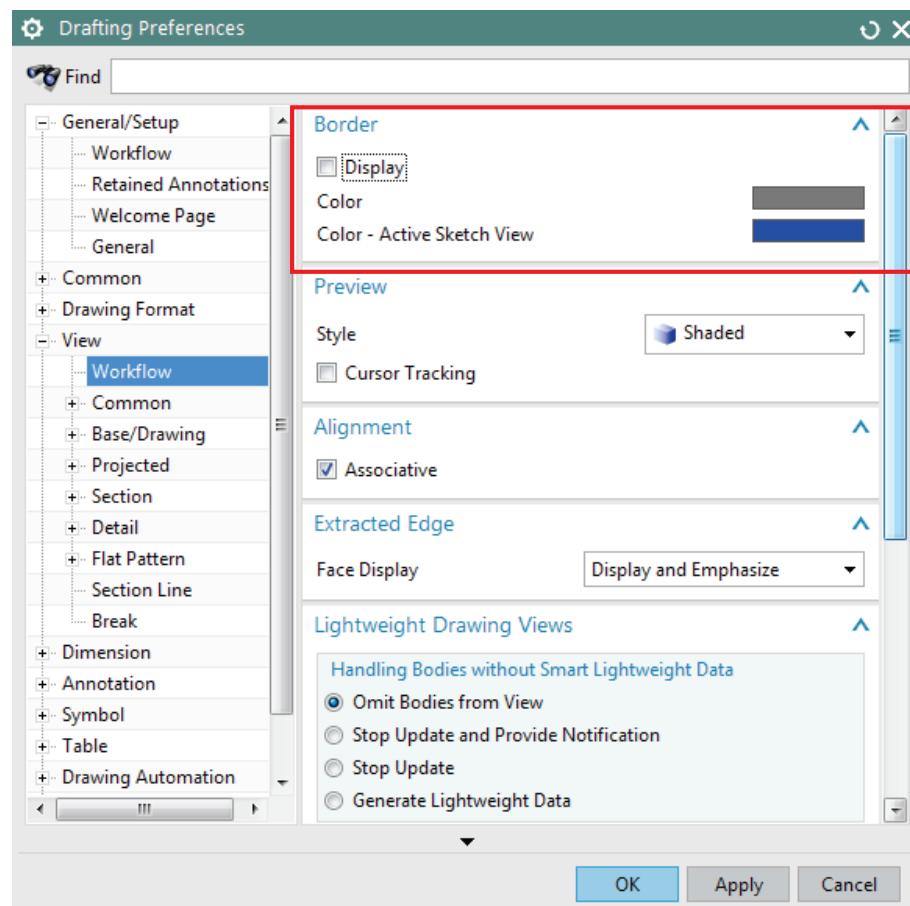
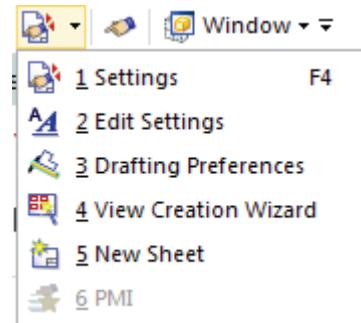
- Move the cursor and click to get the other views
- Click **Close** on the **Projected View** dialog box or press <Esc> key on the keyboard to close the window

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

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

The *Drafting Preferences* window will pop up.

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



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

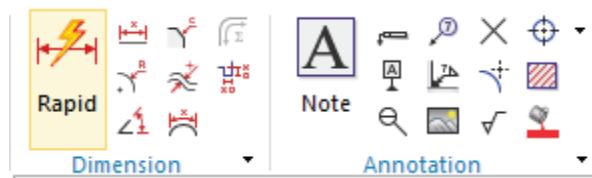
5.3 DIMENSIONING

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

- Choose **Menu → Insert → Dimension**

OR

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

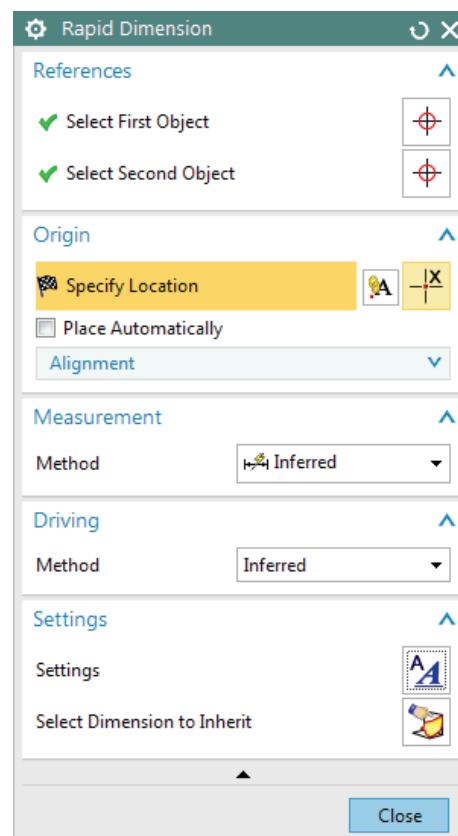
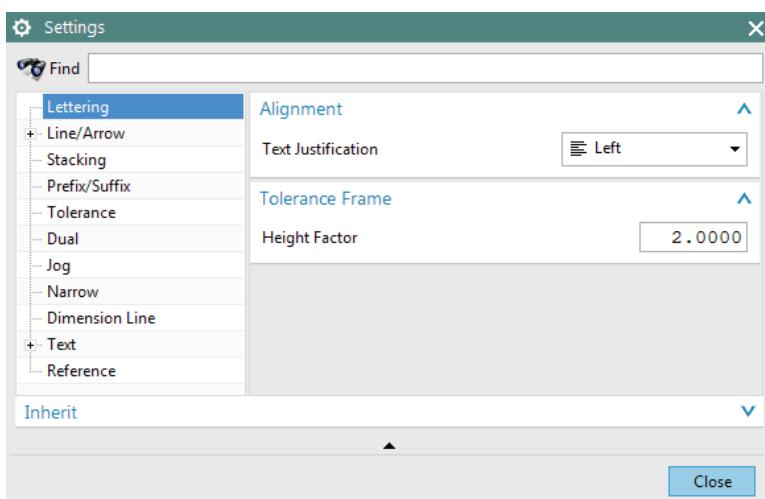


- Click on **Points** and **Edges**, move the mouse and click on the appropriate location to draw dimensions

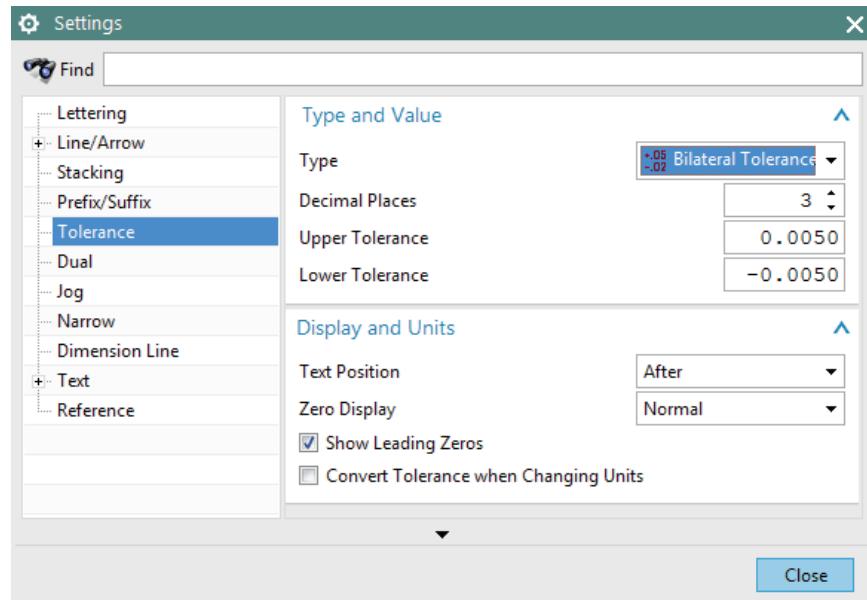
The icons in this window are helpful for changing the properties of the dimensions.

- Click on the **Settings Button**

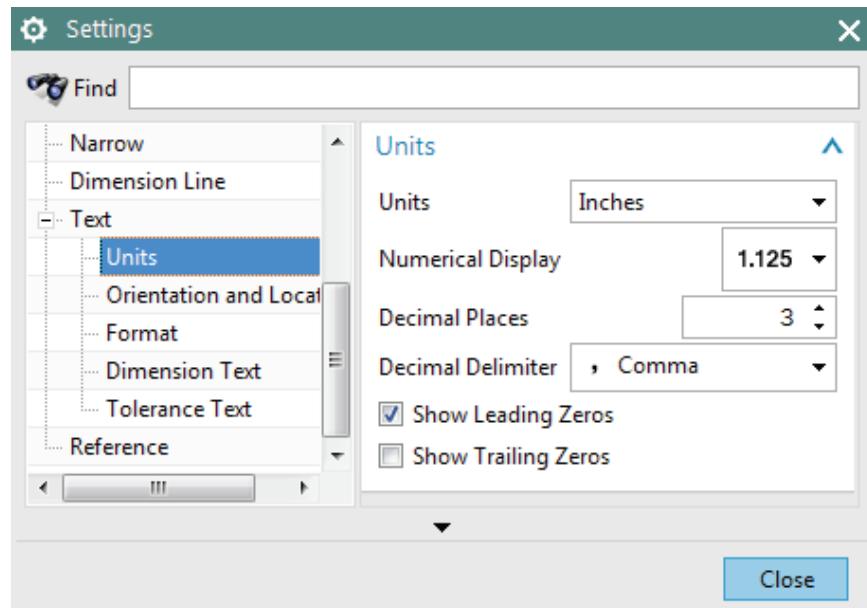
Here you will be able to modify the settings for dimensioning. A dialog appears as shown below.



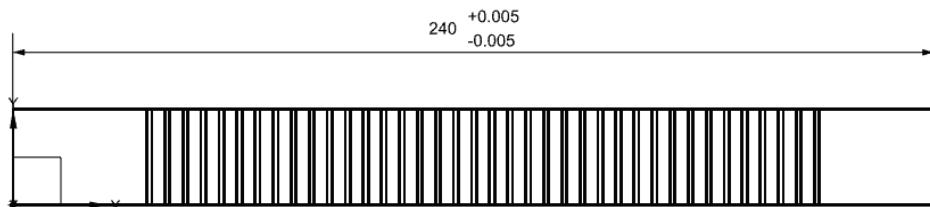
The first list is for *Lettering*. This allows the user to justify and select the frame size. In the *Line/Arrow* section, you can vary the thickness of the arrow line, arrow head, angle format etc. The most important section is the *Tolerance* list. Here you can vary the tolerance to the designed value.



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



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

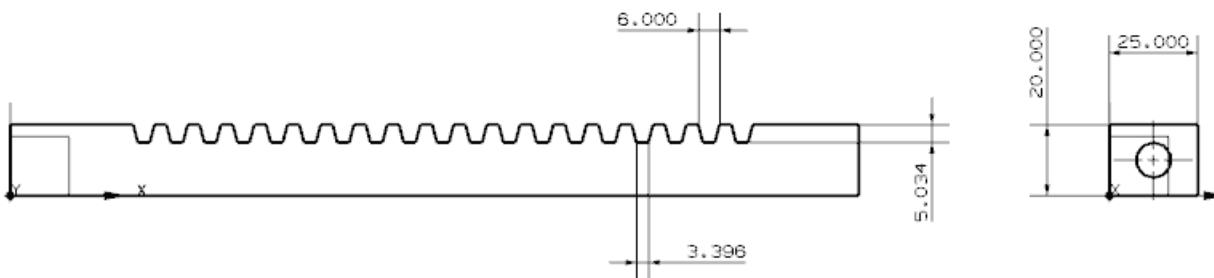


The dimension that represents the distance between these points will appear. You can put the location of the dimension by moving the mouse on the screen. Whenever you place your views in the *Sheet* take into consideration that you will be placing the dimensions around it.

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

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

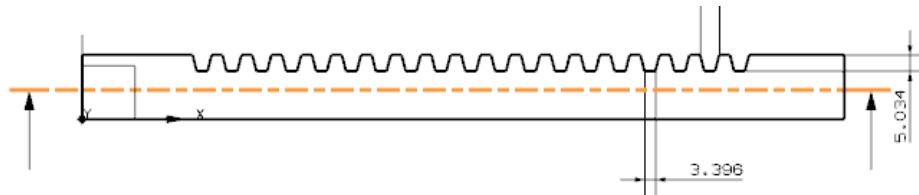
- Right-click on the dimension you just created and choose **Settings** or **Edit Display**
- You can modify font, color, style and other finer details here
- Give dimensions to all other views as shown in the following figure



5.4 SECTIONAL VIEW

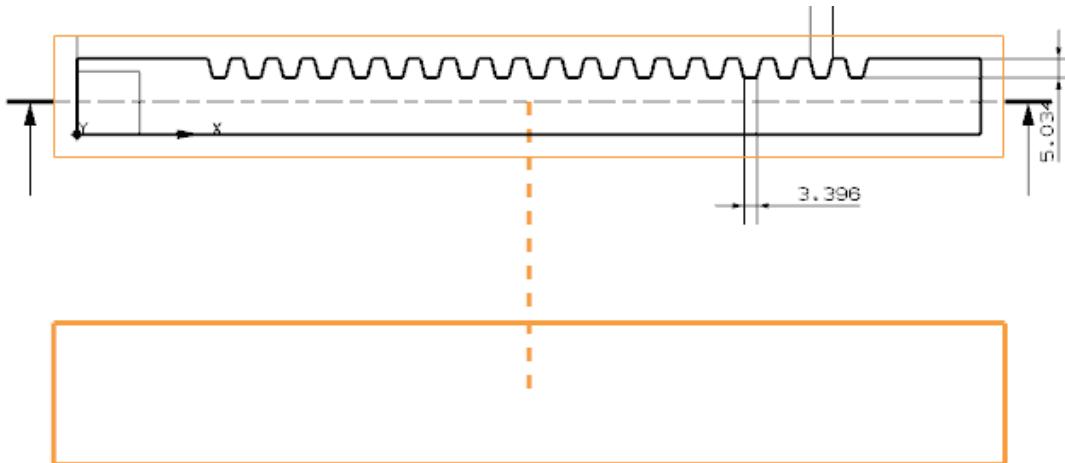
Let us create a *Sectional View* for the same part to show the depth and profile of the hole.

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

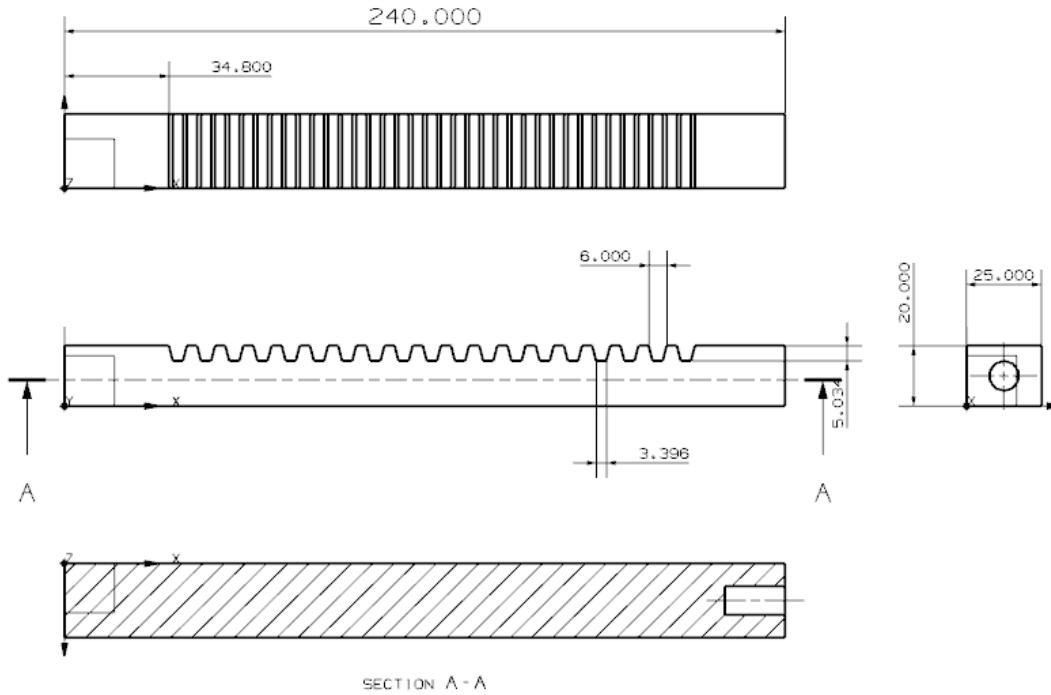


- Click on the middle of the **View** as shown. This will fix the position of the sectional line (Section Plane)

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



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



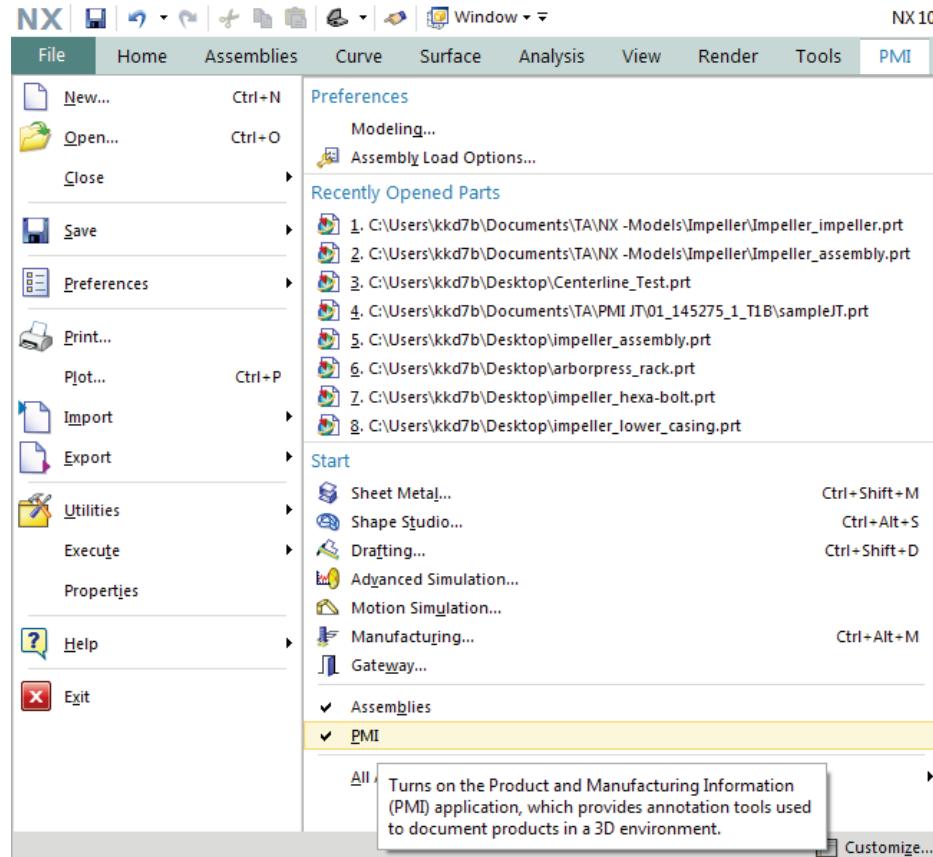
Save and close your model.

5.5 PRODUCT AND MANUFACTURING INFORMATION

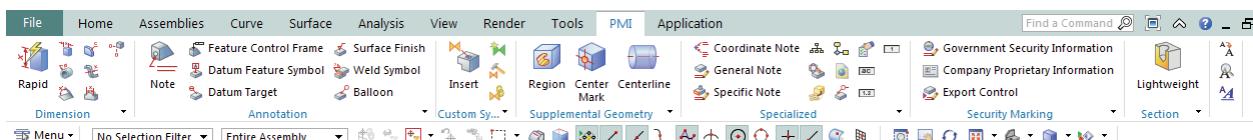
Product and Manufacturing Information (PMI) is one of the important applications in NX which provides annotation tools used to document products in a 3D environment. PMI application includes a comprehensive 3D annotation environment that allows design teams to share details such as Geometric Dimensioning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

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

- Open the file **Impeller_impeller.prt**
- From the NX 10 interface, choose **File → PMI** (turn on the check mark)



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



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



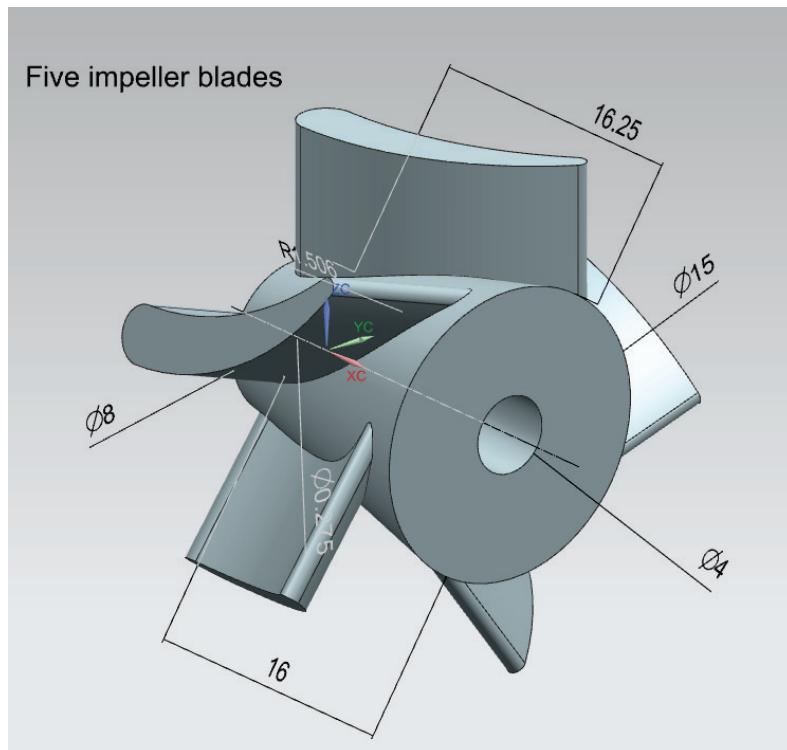
- Click **Rapid** icon

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

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

- Click the **Centerline** icon  in the **Supplemental Geometry** group and select the inner surface of the impeller to insert the centerline for the part
- Click the **Note** icon in **Annotation** group to provide any comments or **Surface Finish** icon, select the object, location of text and leader line to insert the specific surface finish details, if required

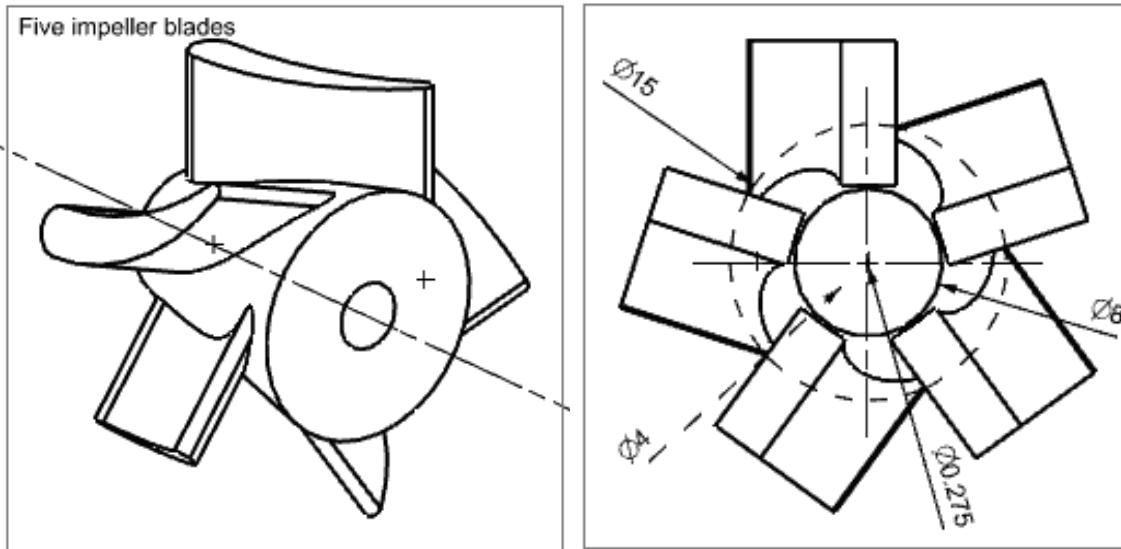
The *Trimetric* view of the impeller after PMI dimensioning would look as shown below.



- Save the file, select **Application** tab and click on **Drafting** icon in the ribbon bar
- Follow the similar procedures explained in the previous section to create the **Drawing** sheet for the 3D part

During the creation of the sheet, in the *View Creation Wizard*, select the *Inherit PMI* option, and select the *Aligned to Drawing (Entire Part)* and check the *Inherit PMI onto Drawing* option. This would inherit the dimensions of the 3D model and show on the drawing sheet including the

comments as shown below. The user has to select the appropriate views to reflect the dimensions on the drawing sheet.



5.6 EXAMPLE

- Open the model **Impeller_hexa-bolt.prt**
- Choose **File → Drafting** or select **Drafting** in **Application** tab
- On the **Sheet** window, select sheet **E-34 X 44** and change the **Scale** value to **8.0 : 1.0**
- Click **OK**



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



- Add the **Front** view by repeating the same procedure explained in the last example

➤

- Add the **Orthographic Views** including the **Right** view and **Top** view



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

The screen will have the following three views.



To see the hidden lines,

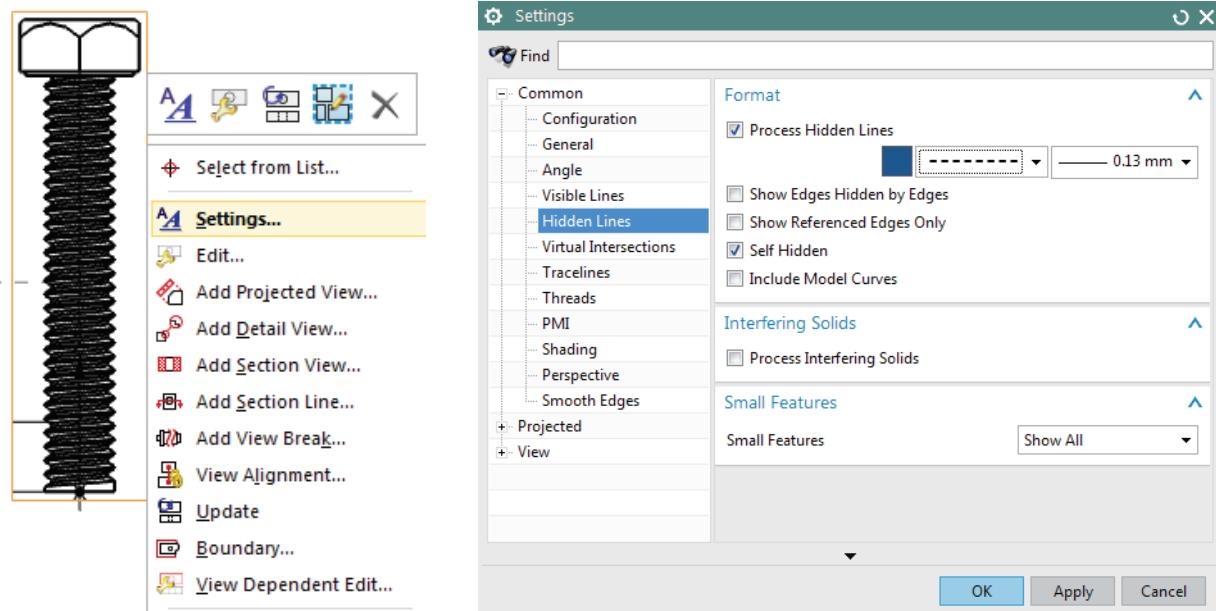
- Choose **Preferences** → **Drafting** → **View**

OR

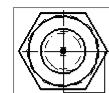
- Select the views, right-click and choose **Settings** as shown below

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**

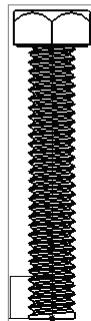
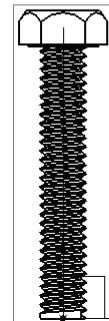


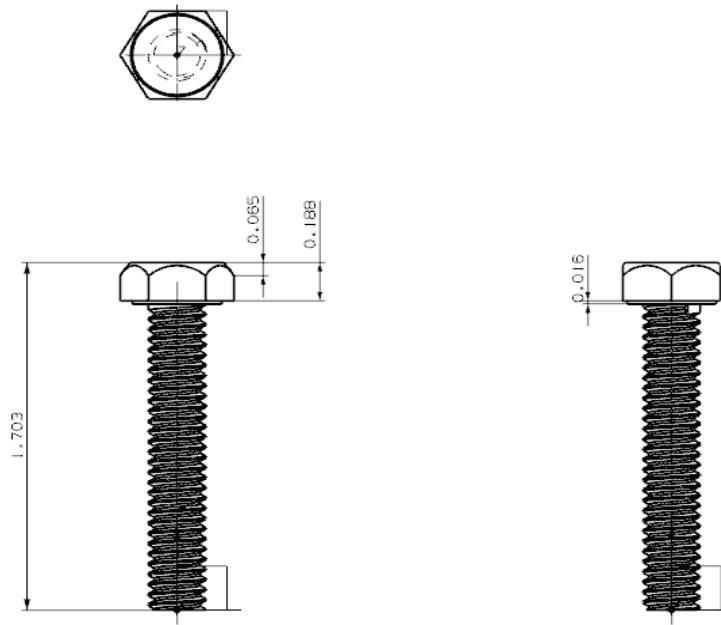
You can see the hidden lines as shown in the picture on the right.



Now we will proceed to dimensioning.

- Choose **Insert** → **Dimensions** → **Linear** or click the **Linear Dimension** icon  in the **Dimension** group
- Give vertical dimensions to all the distances shown below





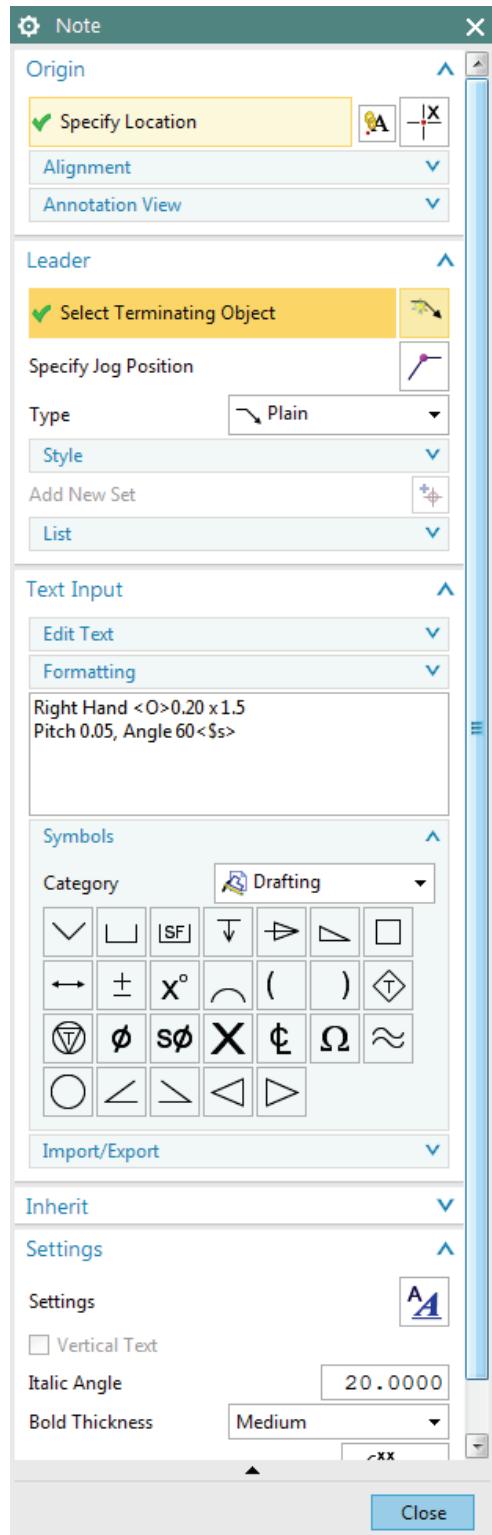
For the threading, we will use a leader line.

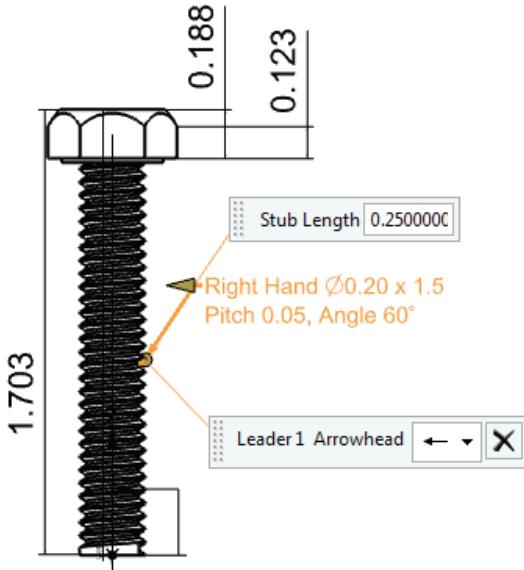
- Click on the **Note** icon shown in the Toolbar
- In the **Note** window that opens, enter the following text. You can find \varnothing and the degree symbol on the **Symbols** tab

Right Hand $\varnothing 0.20 \times 1.5$

Pitch 0.05, Angle 60°

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





- Close the **Annotation Editor**

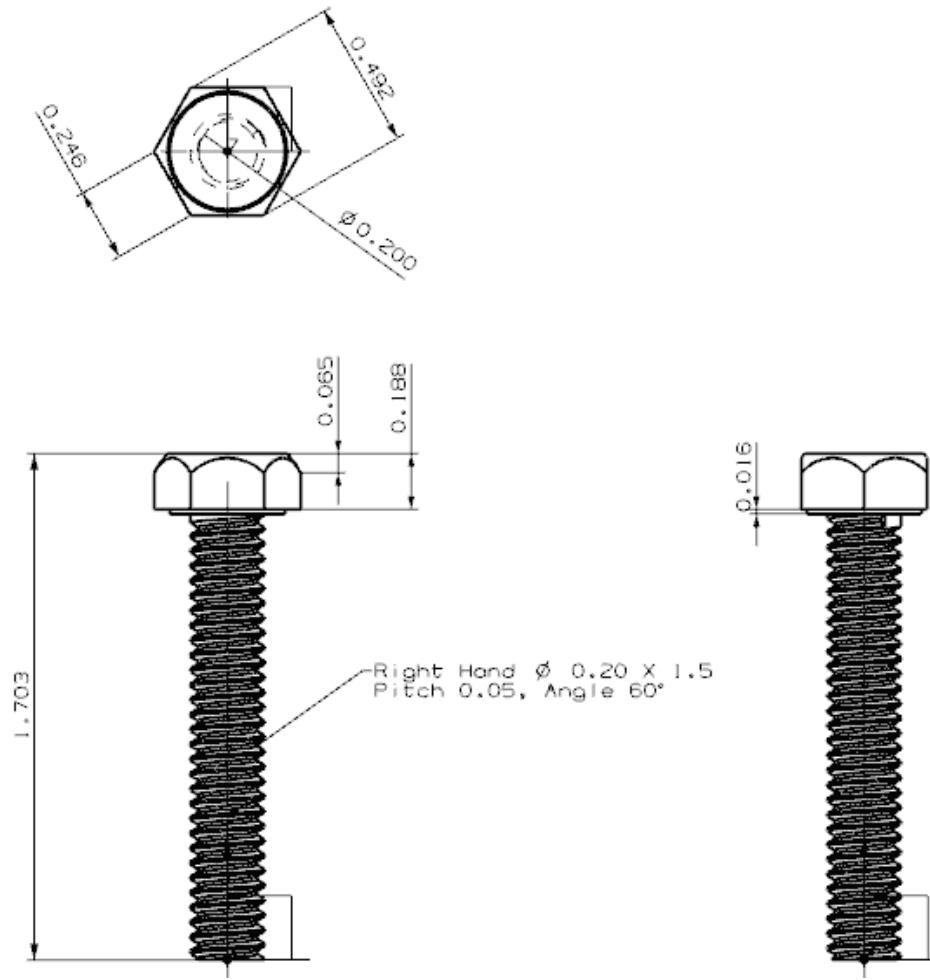
Since the height of the *Lettering* is small, we will enlarge the character size as well as the arrow size.

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

Now we will add additional dimensions and views.

- Choose **Insert → Dimensions → Radial** or click the **Radial Dimension** icon  in the **Dimension** group
- Click the circle of the bolt in the top view to give the diameter dimension
- Click **Insert → View → Base View** or click the **Base View** icon 
- Select the **Isometric** view and place the view somewhere on the screen

The final drawing is shown below. Remember to save.



5.7 EXERCISE

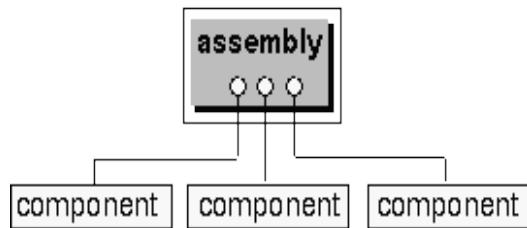
Perform *Drafting* and give dimensions to the circle base that you modeled in Exercise 4.8.1.

CHAPTER 6 – ASSEMBLY MODELING

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

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

NX 10 *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.



6.1 TERMINOLOGY

Assembly

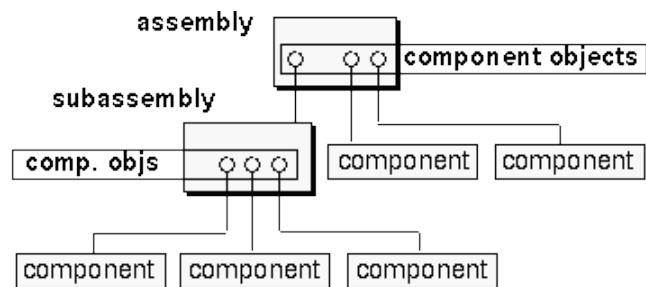
An assembly is a collection of pointers to piece parts and/or subassemblies. An assembly is a part file, which contains component objects.

Component Object

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

Component Part

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



Component Occurrences

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

Reference Set

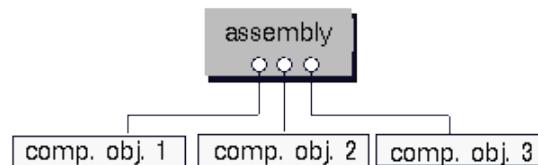
A *reference set* is a named collection of objects in a component part or subassembly that you can use to simplify the representation of the component part in higher level assemblies.

6.2 ASSEMBLING APPROACHES

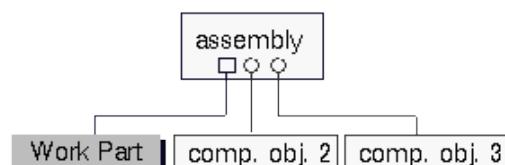
There are two basic ways of creating any assembly model.

- Top-Down Approach
- Bottom-Up Approach

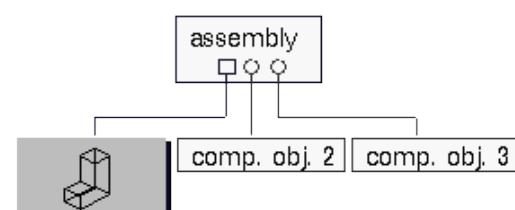
- Create component objects first.



- Make a component the Work Part.



- Create geometry in the component part.



6.2.1 Top-Down Approach

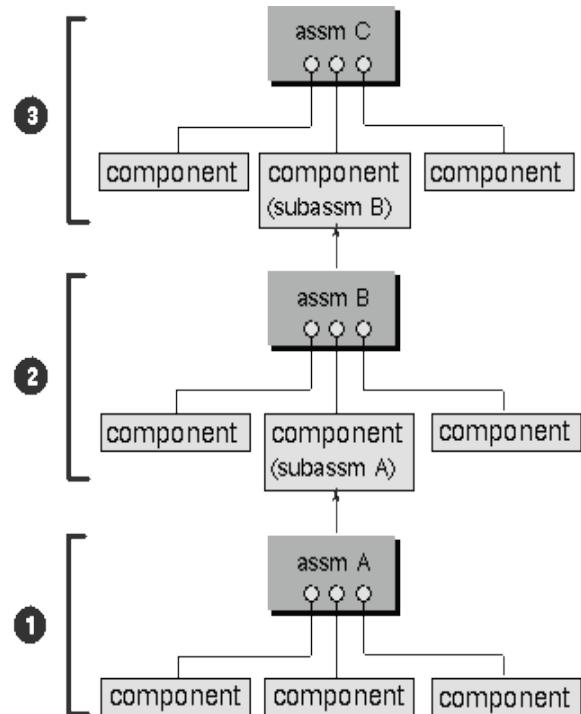
In this approach, the assembly part file is created first and components are created in that file. Then individual parts are modeled. This type of modeling is useful in a new design.

6.2.2 Bottom-Up Approach

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

6.2.3 Mixing and Matching

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



6.3 ASSEMBLY 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.

Descriptive Part Name	Info	Read-o...	M	Position	Count	Reference Set
Impeller_assembly (Order: Ch...)					11	
Constraints				✓	19	
Impeller_upper-casing				●		Model ("MODEL")
Impeller_lower_casing				●		Model ("MODEL")
Impeller_impeller				●		Model ("MODEL")
Impeller_Shift				●		Model ("MODEL")
Impeller_hexa-bolt x 2				●		Model ("MODEL")
impeller_washer x 2				●		Model ("MODEL")
Impeller_hexa_nut x 2				●		Model ("MODEL")

- 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

6.4 MATING CONSTRAINTS

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



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



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



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



Parallel: Objects selected will be parallel to each other.



Perpendicular: Objects selected will be perpendicular to each other.



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



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

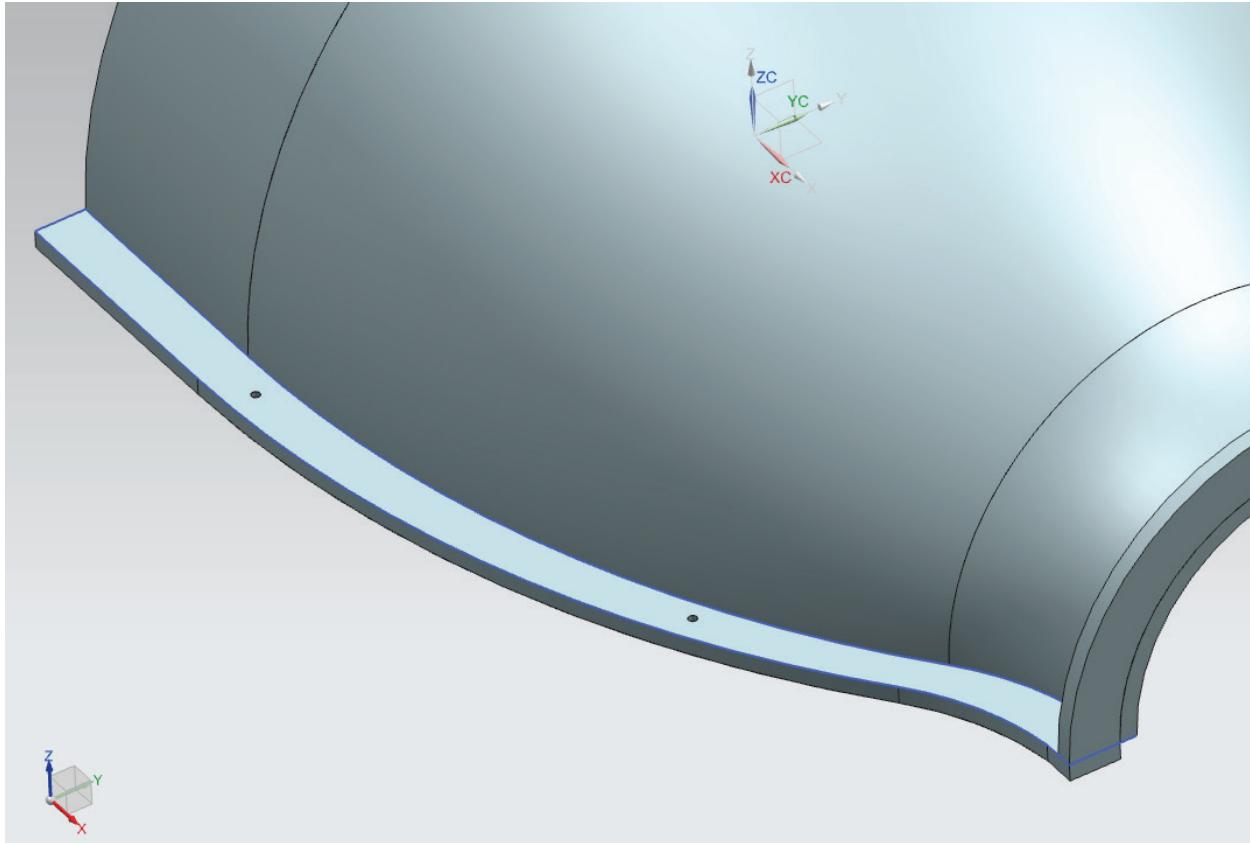


Angle: This fixes a constant angle between the two object entities chosen on the components to be assembled.

6.5 EXAMPLE

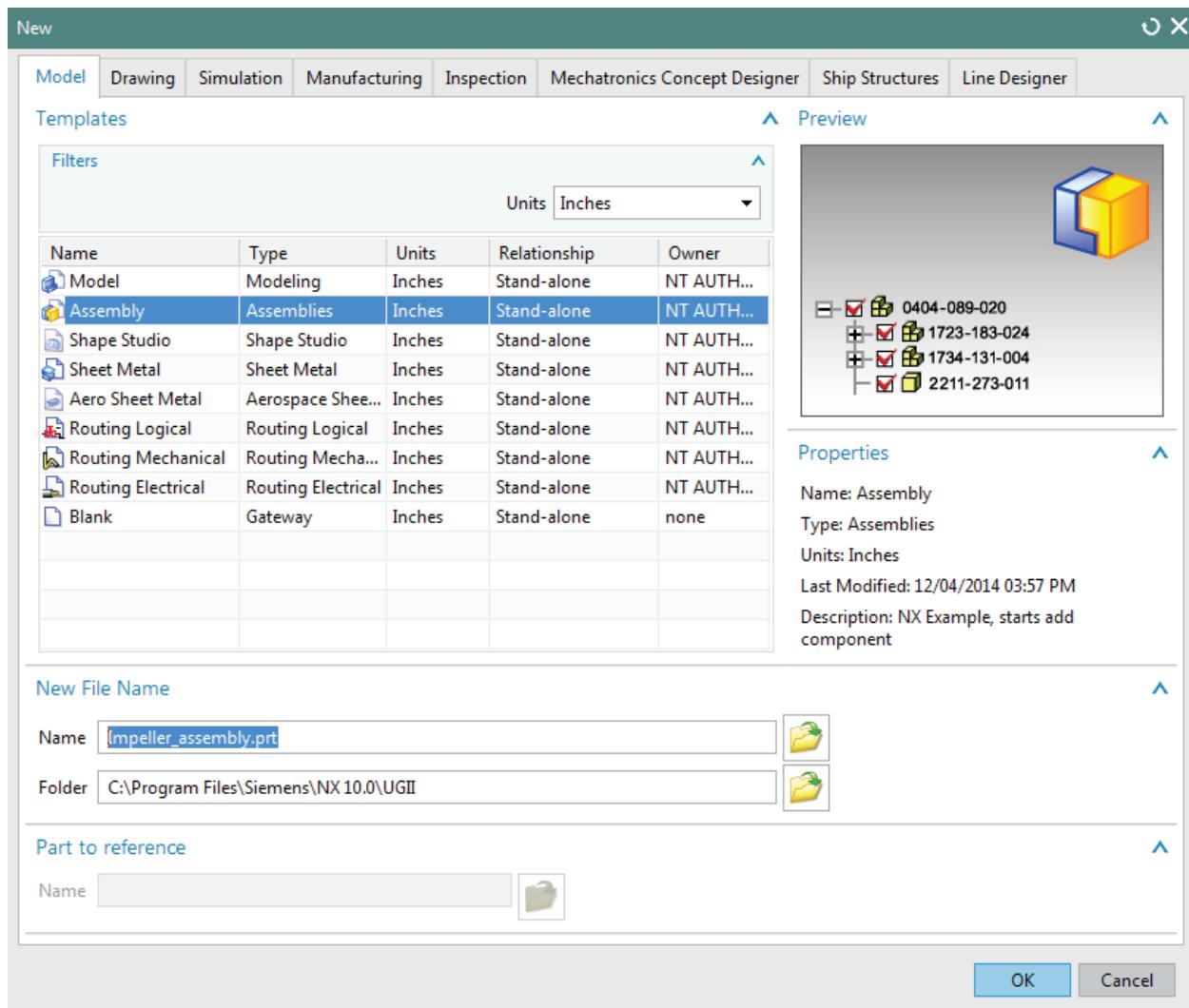
We will assemble the impeller component objects. You have modeled all the components in previous chapters. Now we have to insert them into the assembly environment and apply constraints to locate them relative to each other. Once the assembling is completed, we can create an exploded view and prepare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

Before starting the assembly modeling, make two through-holes on each side of the *Impeller-lower-casing* and *Impeller-upper-casing* (a total number of 4 holes for each casing) for the *Hexabolt*. Diameter of the holes should be 0.25 and their location should be similar to the figure below. Make sure to create the holes in the same places for lower and upper casing to that when they are assembled they match.



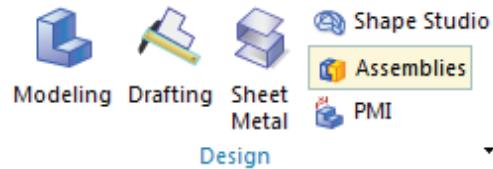
6.5.1 Starting an Assembly

- Create a new file
- Choose **Assembly** under the **Model** tab
- Name it as **Impeller_assembly.prt**



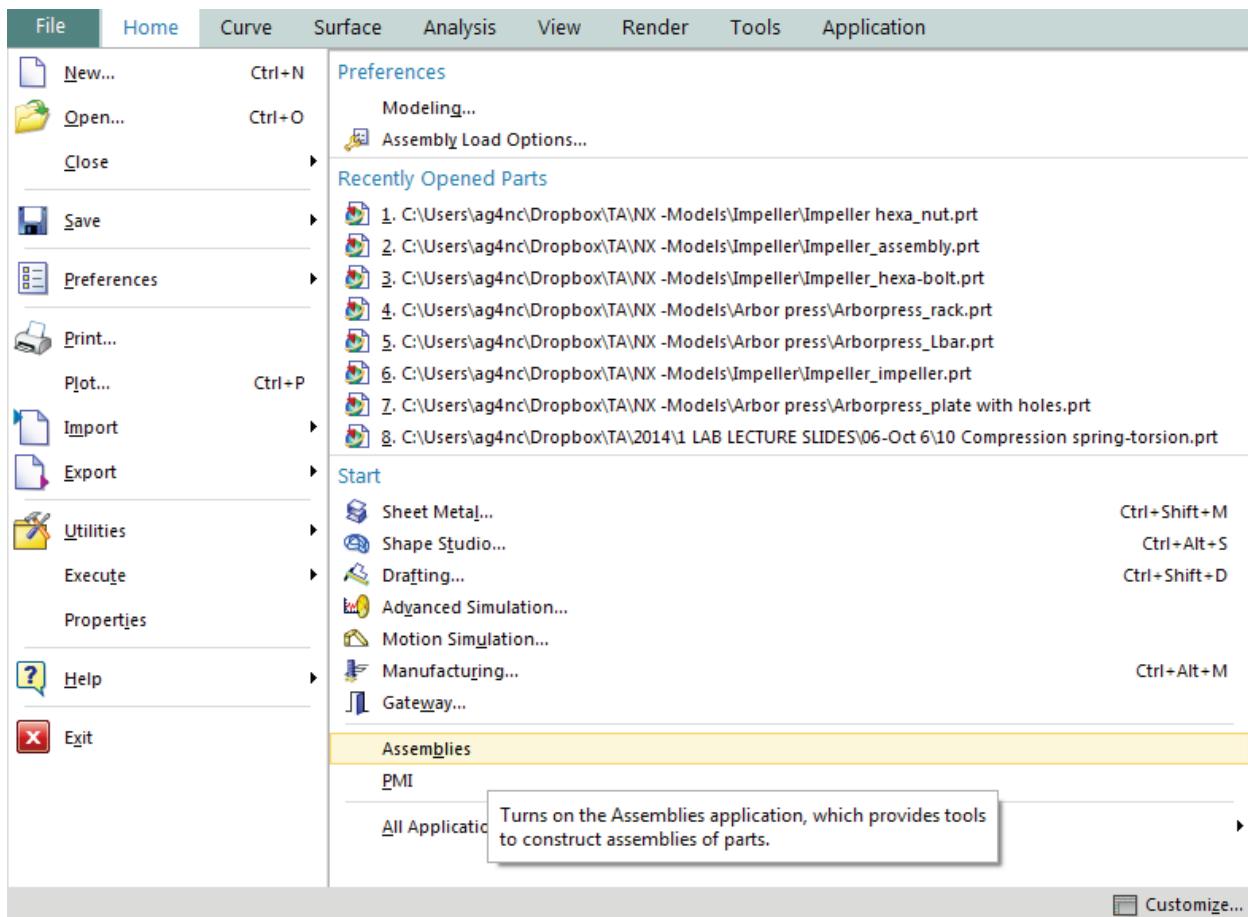
OR, if you are in the *Modeling Application* and want to start assembling,

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

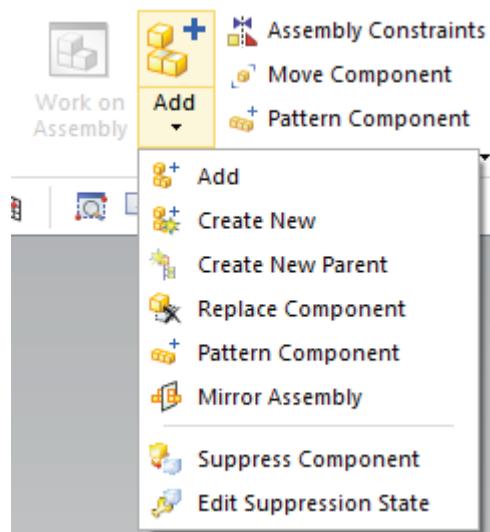


OR

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



- The **Home** menu bar will now display tools for assembly



In the **Components** option,

- *Add* option adds new component objects whose part files are already present.

- *Create New* lets you create new component geometries inside the assembly file in case you are using *Top-Down* approach of assembly.

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

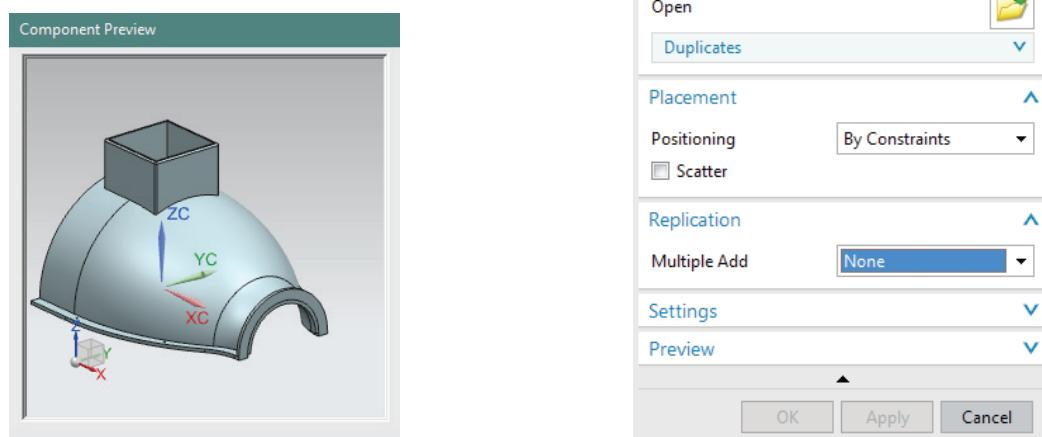
6.5.2 Adding Components and Constraints

- Choose **Add**

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

- Click on the **Open** icon and select the file **Impeller_upper-casing.prt**
- Click **OK** in the part name dialog box

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 a certain location. This can be done by changing the *Positioning* option in the *Placement* dialog box to *Absolute Origin*.

- Click **OK**

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 **Positioning** dialog box change the option to **By Constraints**
- Choose **Apply**

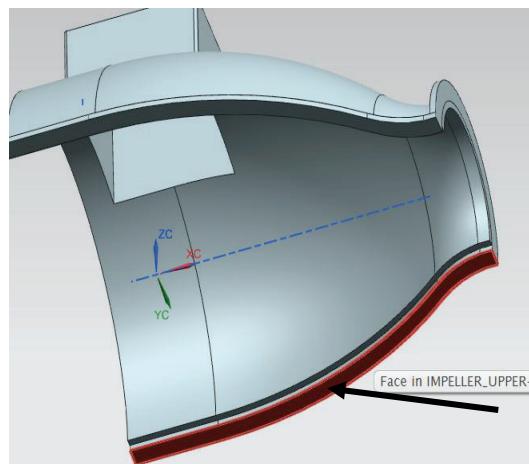
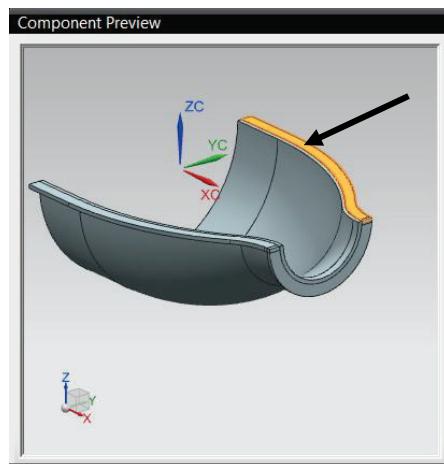
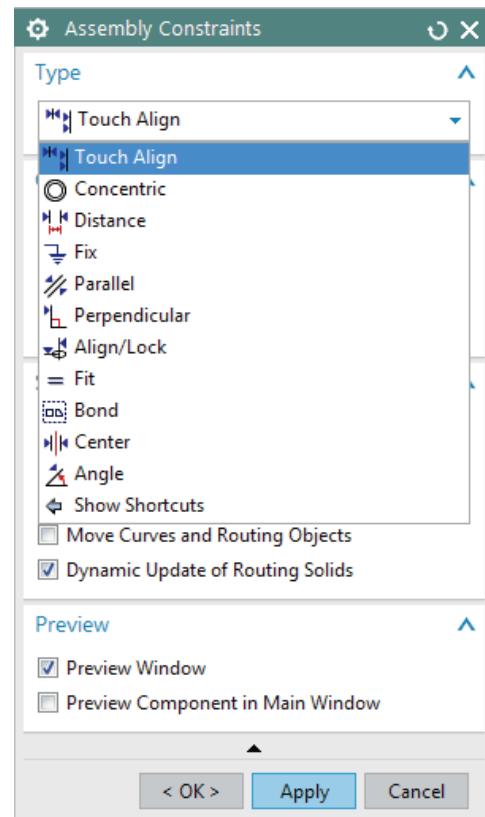
This will show you the added component in a *Component Preview* window as before.

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.

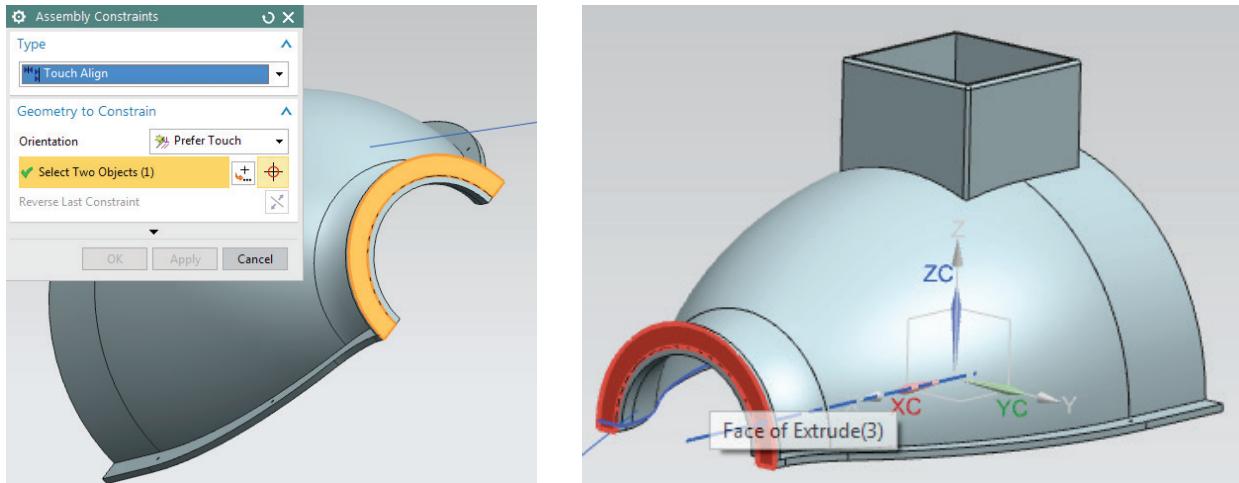
Here you can see the different Mating Types, which were explained above in the previous section.

- Make sure the **Touch Align** icon  is selected in the **Type** dialog box
- First, select the face that the arrow is pointing to in the **Component Preview** window as shown below in the figure on the left.
- 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.

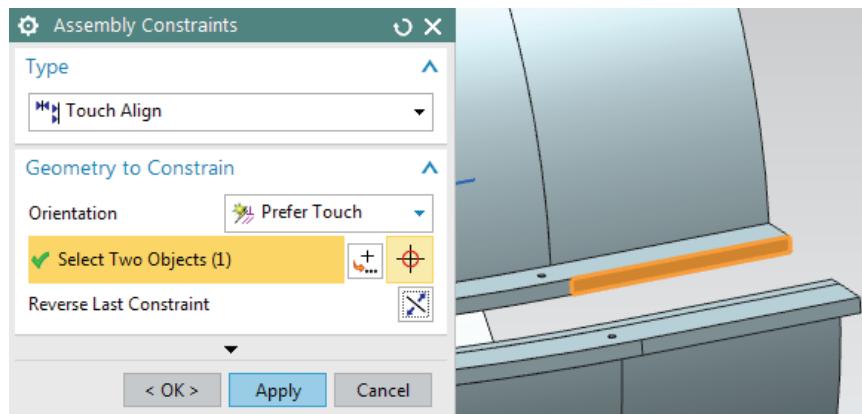


- Click on the **Assembly Constraints**
- Choose the **Touch Align** as the **Type**
- Click on the **Flange** of the lower casing
- Click on the **Flange** of the upper casing, you may need to inverse the direction of constraint by click on the **Inverse** icon 

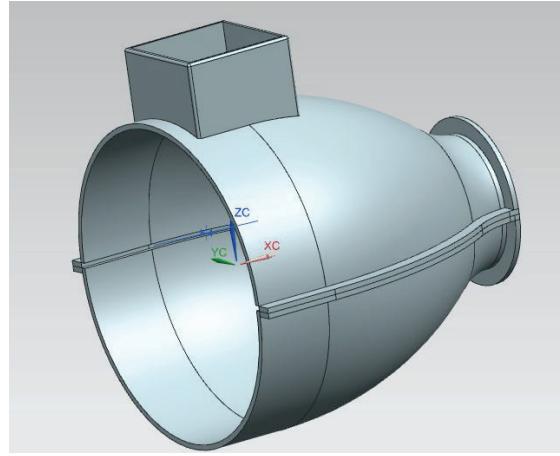


Note: if it is difficult for you to select the faces because of the position of the parts, you can move them by clicking on the *Move Component* in the same *Assemblies* group.

- Select the **Tough Align** again
- Click on the flat face of the lower casing as shown and then the same face on the upper casing



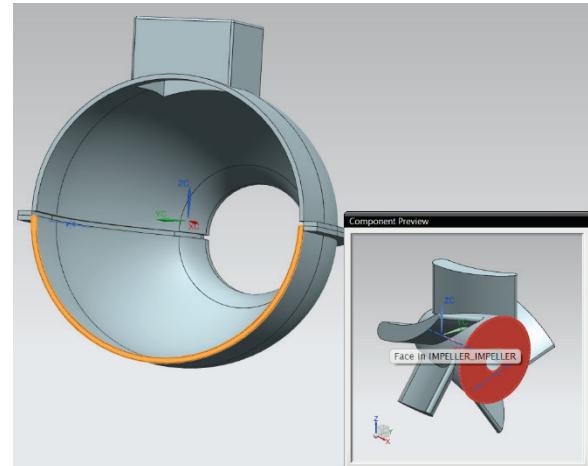
The two assembled components will be seen as shown in the figure below.

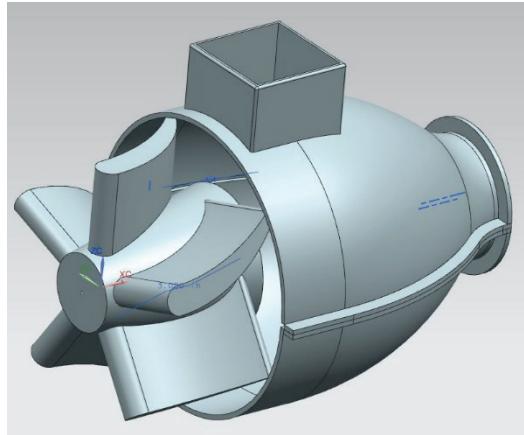


The lower casing is constrained with respect to the upper casing. Now let us add the impeller.

- Choose **Assemblies** → **Components** → **Add Component** as an alternative way to add a component to the assembly
- Open the file **Impeller_impeller.prt**
- Click **OK** on the dialog box
- 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, uncheck the **Preview Window** option

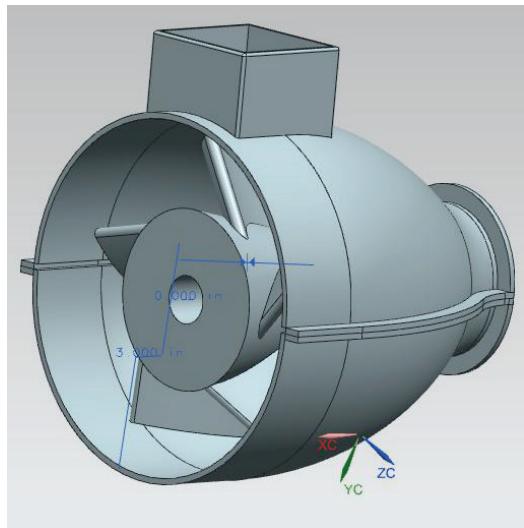
The preview may show the impeller oriented in the direction opposite to the one we want.





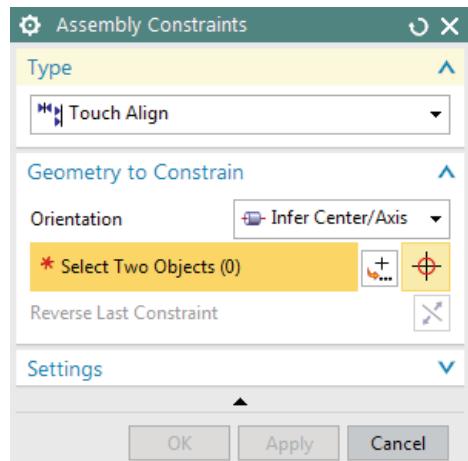
- On the **Assembly Constraints** window, click on the **Reverse the Last Constraint** option in the **Geometry to Constrain**

Now the impeller will be oriented in the right direction.

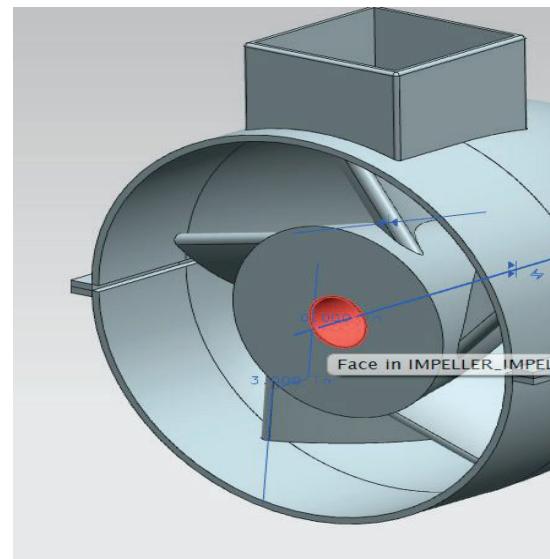
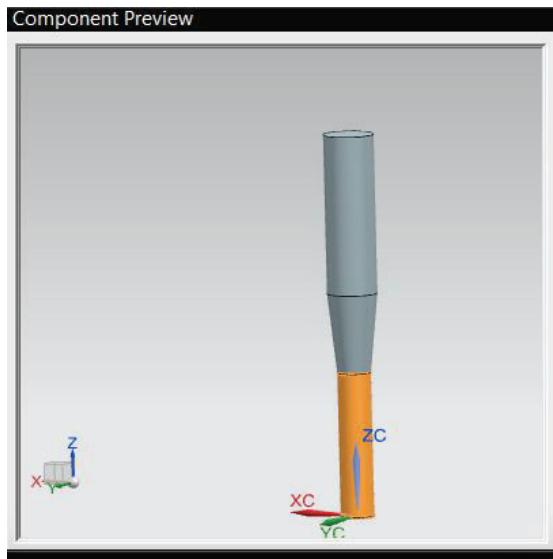


We will now add the shaft using the *Center* constraint.

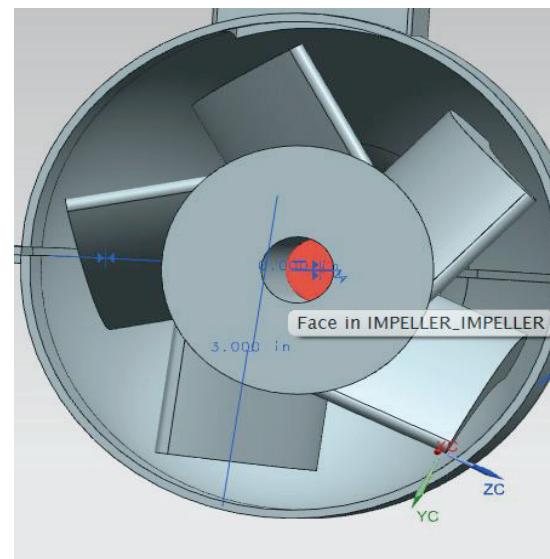
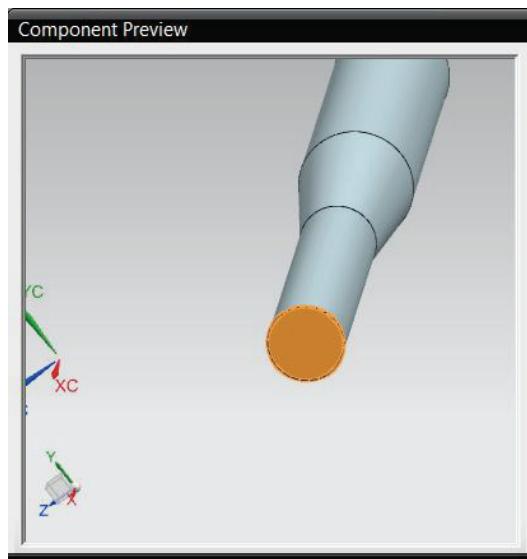
- Click on **Assemblies** → **Components** → **Add Component**
- Open the file **Impeller_shaft.prt**
- Click **OK** on the dialog box
- Choose the **Touch Align** icon



- Choose the **Infer Center/Axis** option in the **Geometry to Constrain** dialog
- 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

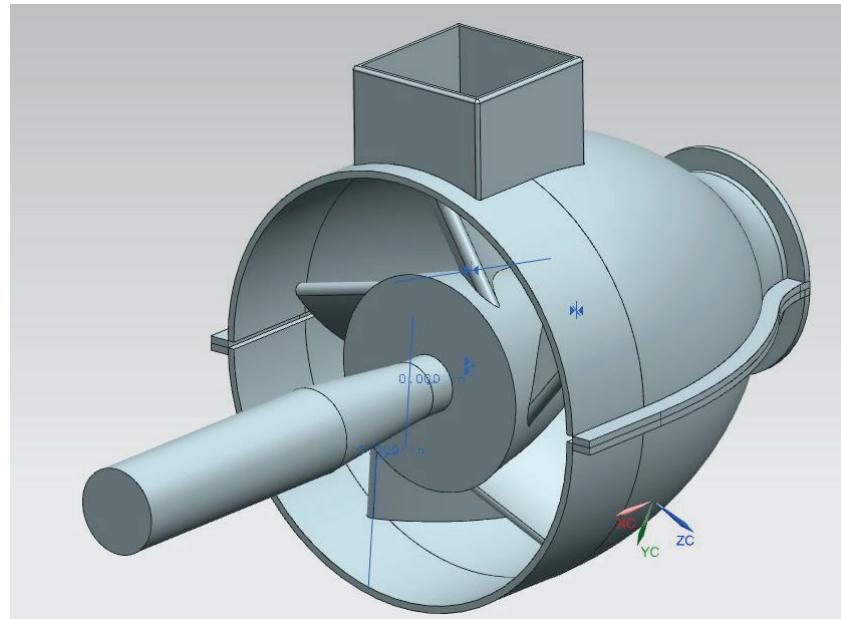


- Choose the **Touch Align** constraint
- First, select the face on the shaft and then select the bottom face of the hole in the impeller as shown

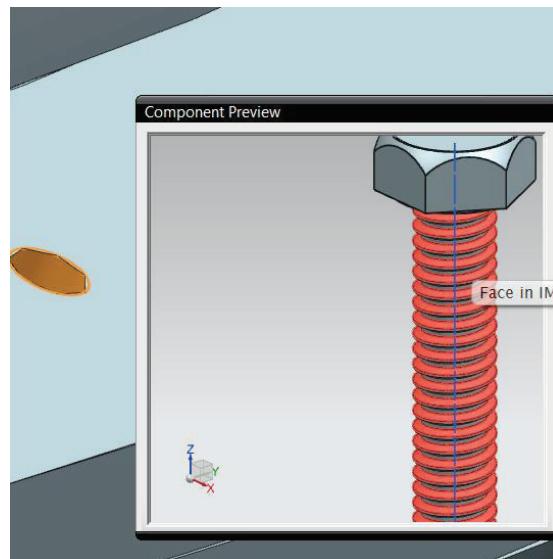


- Choose **Apply** and then click **OK**

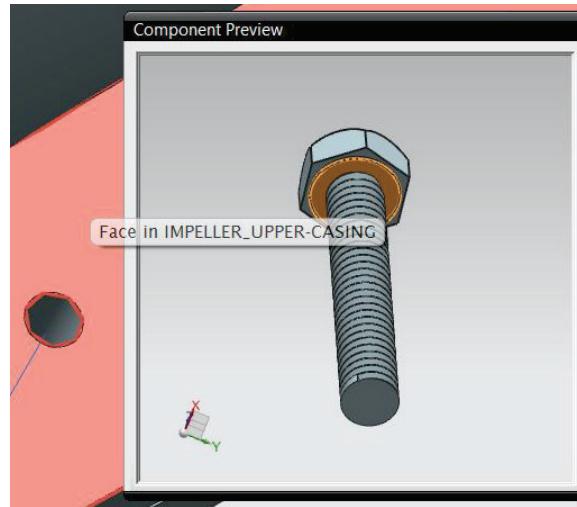
The assembly will now look like the figure below.



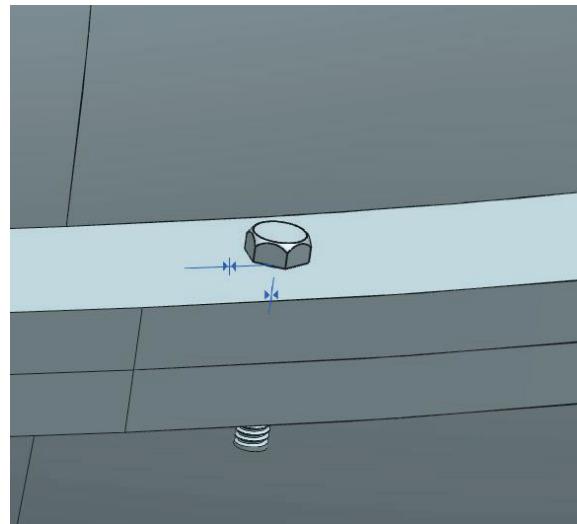
- Click on **Assemblies** → **Components** → **Add Component**
- Open the file **Impeller_hexa-bolt.prt**
- Choose the **Touch Align** constraint. Use the **Infer Center/Axis** option in the **Geometry to Constrain** dialog box
- First, select the outer cylindrical threading on the bolt and then select 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.

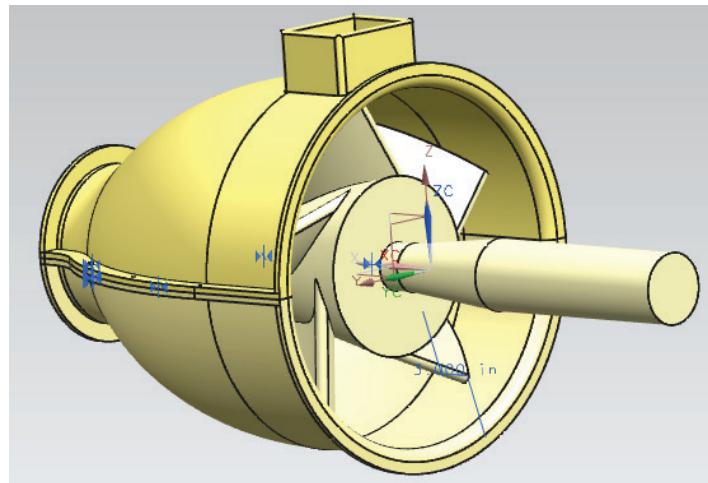


- Repeat the same procedure to add bolts and nuts to all the holes in the casing.

This completes the assembly of the impeller.

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

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

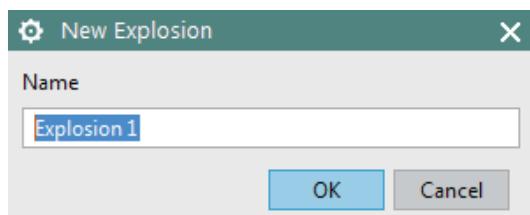


6.5.3 Exploded View

In this section, we are going to create an *Exploded View* of the assembly to show a separated part-by-part picture of the components that make the assembly. In today's industrial practice, these 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**

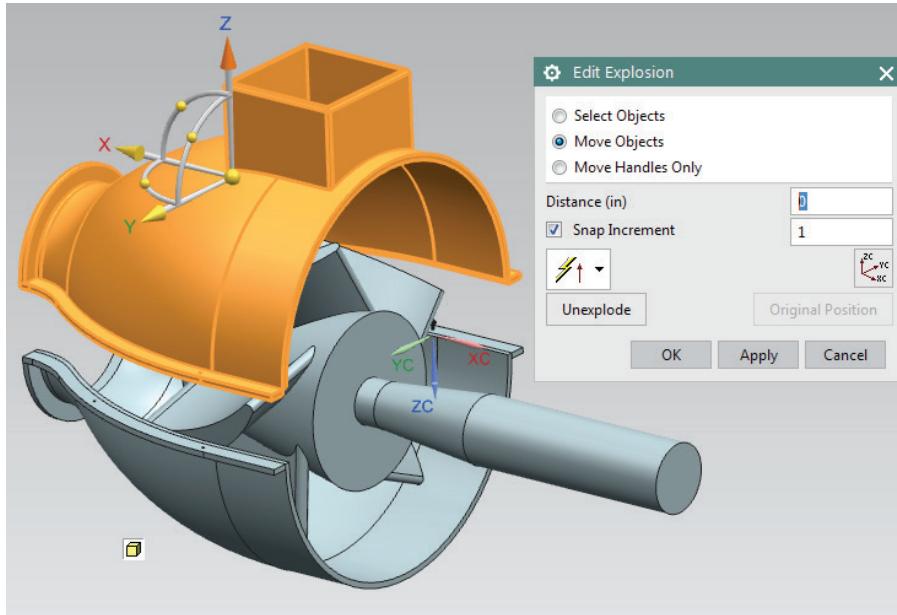
This will pop a dialog box asking for the name of the *Exploded View* to be created. You can leave name as the default name and choose *OK*



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

- Right click on the upper casing and choose **Edit Explosion**

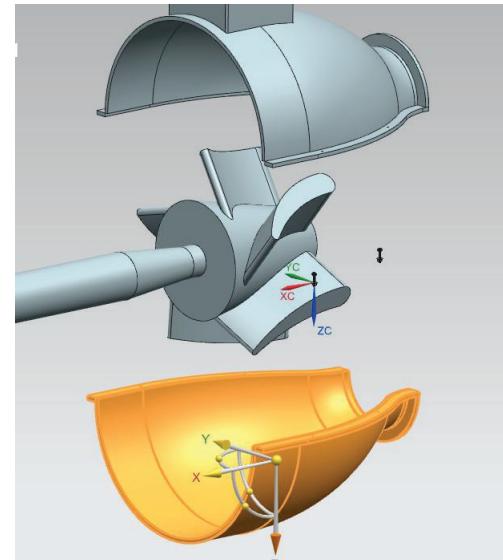
The *Edit Explosion* window will pop up along with a coordinate system on the component.



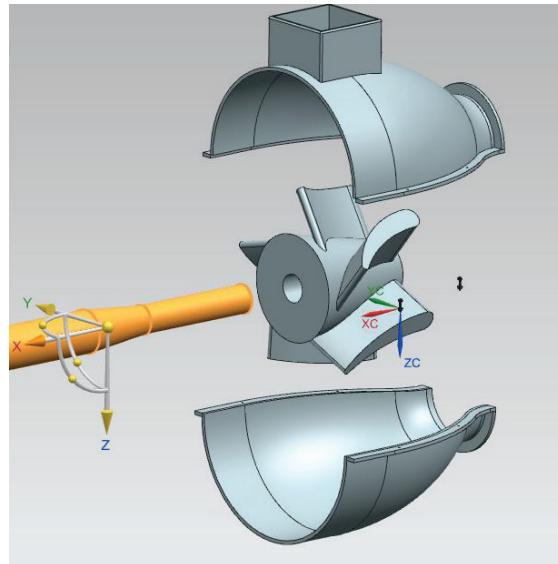
- Click on the **Z** axis; hold the mouse and drag upwards until the reading in the **Distance** shows **-20** (**substitute +20 if you have designed in opposite direction**)
- Click **OK**
- Right click on the lower casing and choose **Edit Explosion**

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

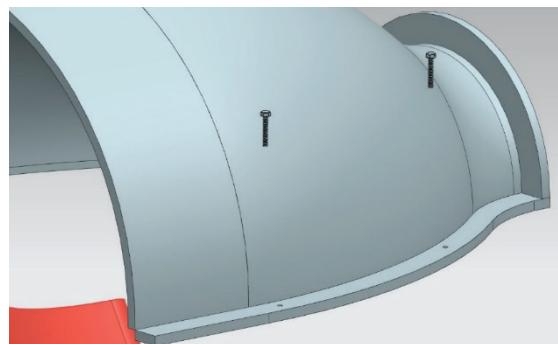
- Click on the **Z-axis**; hold the mouse and drag downwards until the reading in the Distance shows **20** as shown in the following figure.
- Right click on the shaft and choose **Edit Explosion**
- This time click on the **X-axis**; hold the button and drag to the right side until the reading in the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



- Choose **OK**

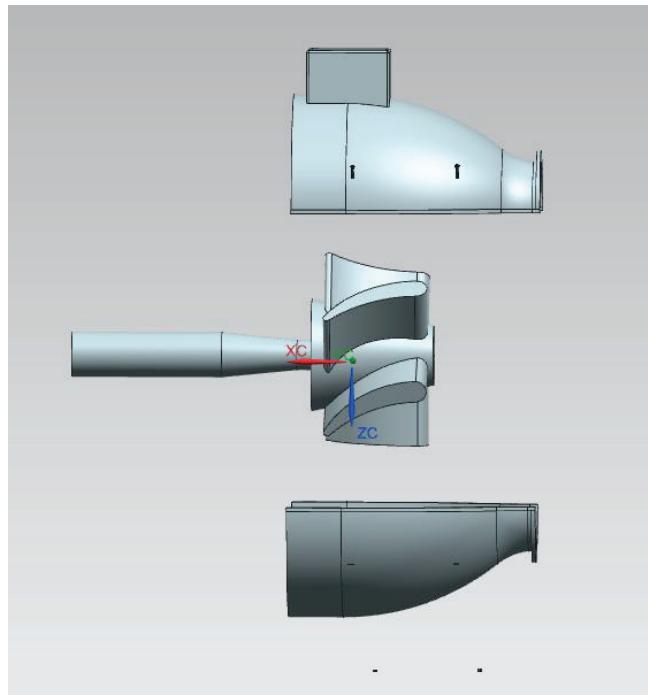


- Select all the four hexagonal bolts in the assembly by clicking on them
- Right click on one of them and choose **Edit Explosion**
- This time click on the **Z-axis**; hold the button and drag upwards until the reading in the Distance shows **25** as shown in the following figure. This will move all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

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



If you want to retain the original assembly view you can *Unexplode* any 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**

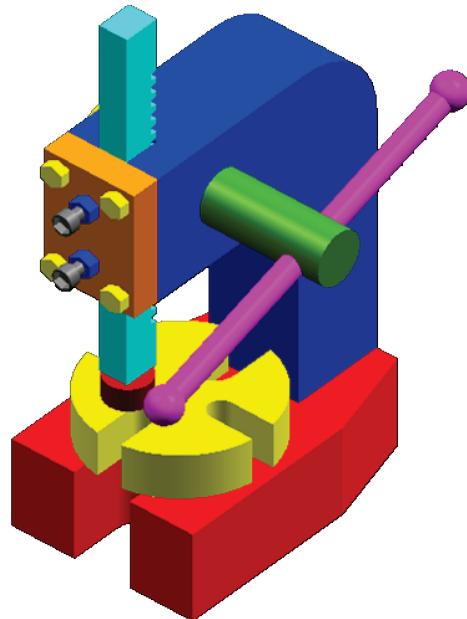
6.6 EXERCISE

In previous sections of 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

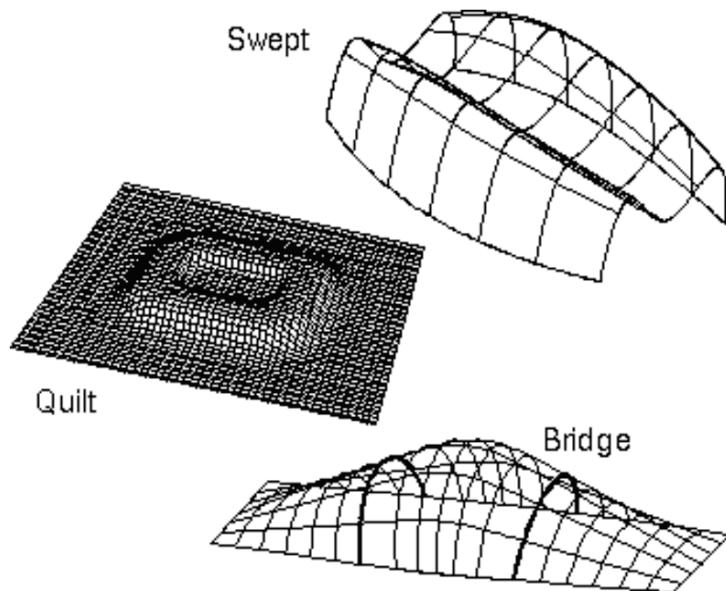
- Handle
- Hexagonal Bolt
- L-bar
- Pin
- Pinion
- Pinion handle
- Plate
- Rack
- Sleeve

All these parts are provided in a folder that can be accessed along with this tutorial in the same internet address (<https://web.mst.edu/~mleu/>).



CHAPTER 7 – FREEFORMING

In this chapter, you will learn how to create freeform models in NX 10. Up to this point, you have learned different ways to create models by using *Form Features* or by *Sketching*. *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 first need a set of points, curves, edges of sheets or solids, faces of sheets or solids, or other objects. The following sections cover some of the methods that you can use to create solids using some of the freeform features.

7.1 OVERVIEW

The *Freeform Features* in NX 10 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. These options are located at various places like *Menu → Insert → Surface/Mesh Surface/Sweep/Flange Surface* and *Menu → Edit → Surface* for more advanced options. A few of the menus that are more useful are discussed below.

7.1.1 Creating Freeform Features from Points

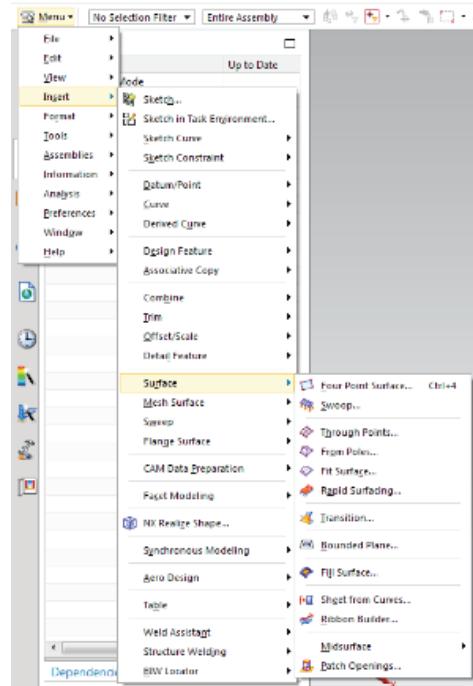
In the case where the geometry you are constructing or pre-existing data includes only points, you may be able to use one of these three options to build the feature from the given points.

- Click on **Insert → Surface**

 **Four point surface:** if you have four corner points.

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

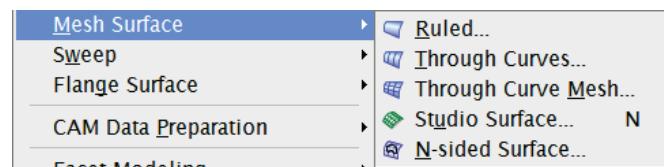
 **From Poles:** if defined points form a rectangular array tangential to the lines passing through them.



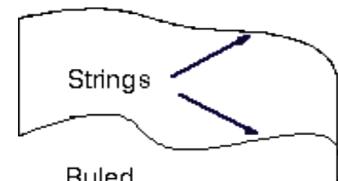
7.1.2 Creating Freeform Features from Section Strings

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.

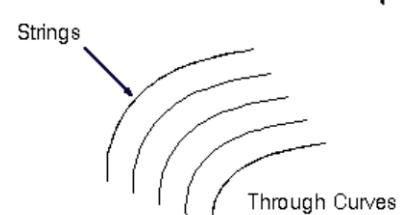
- Click on **Insert → Mesh Surface**



 **Ruled:** Used if two strings are roughly parallel.



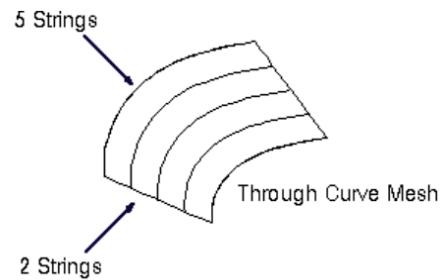
 **Through Curves:** Used if the three or more strings are roughly parallel.



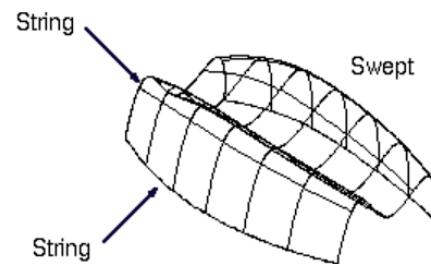
If construction geometry contains two or more strings (curves, faces, edges) that are roughly parallel to each other, and one or more section strings that are roughly perpendicular to the first set of curves (guides), you may be able to use one of these following options to build the feature.



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



Swept: Used if at least two section strings are roughly perpendicular (choose **Insert → Sweep**).



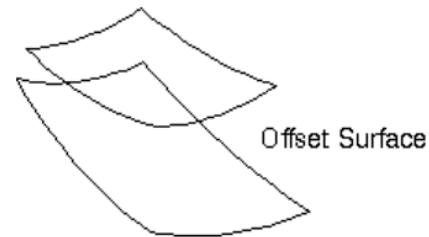
7.1.3 Creating Freeform Features from Faces

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

- Click on **Insert → Offset/Scale**



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



- Click on the **Insert → Flange Surface → Extension**



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

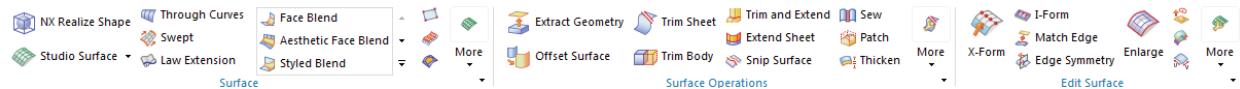


7.2 FREEFORM FEATURE MODELING

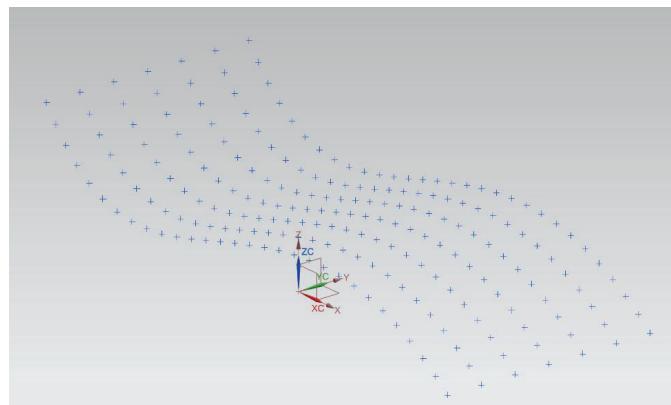
Let us do some freeform modeling on structured points, a point cloud, curves and faces. Structured points are a set of point's defined rows and columns. A point cloud has a set of scattered points that form a cloud.

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.



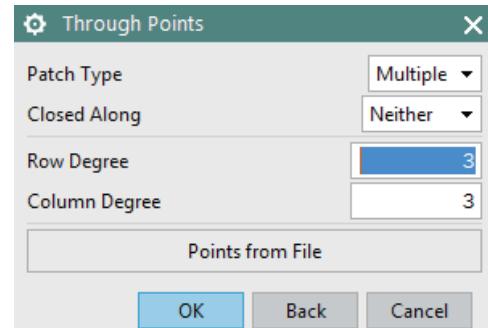
- Choose **Insert → Surface → Through Points**

OR

- Click on the Icon  in the Toolbar

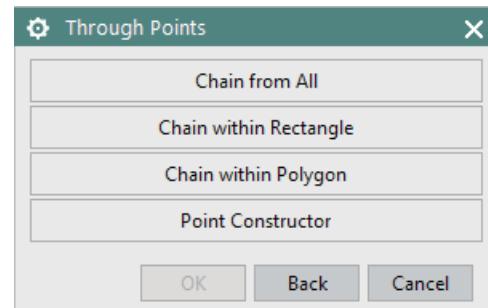
The dialogue box will pop up as shown in the right.

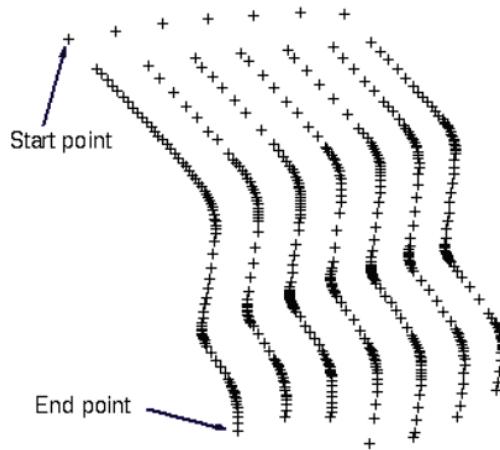
- For **Patch Type**, select **Multiple**
- For **Closed Along**, select **Neither**
- For **Row Degree** and **Column Degree**, enter **3**.
- Click **OK**



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

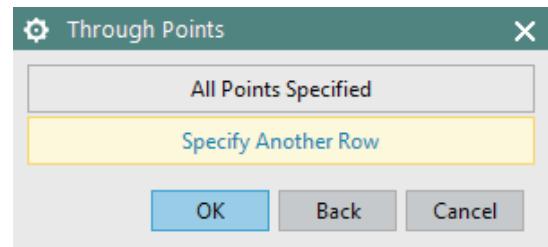




The first row of points will be highlighted.

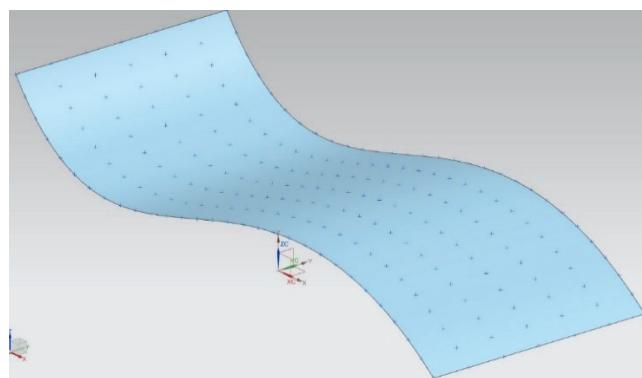
- Repeat the same procedure to select the first four strings of points.

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

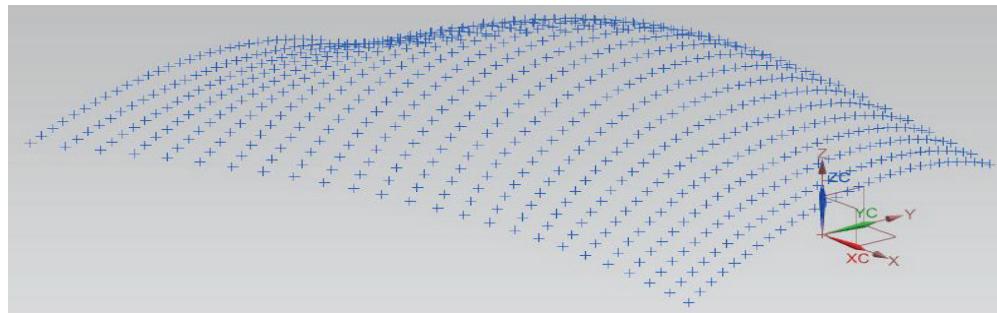
You will see the surface as shown below.



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**

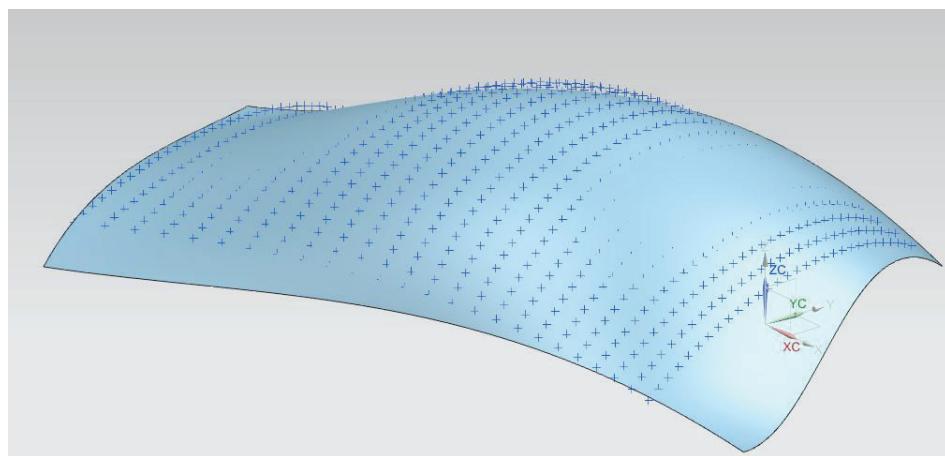
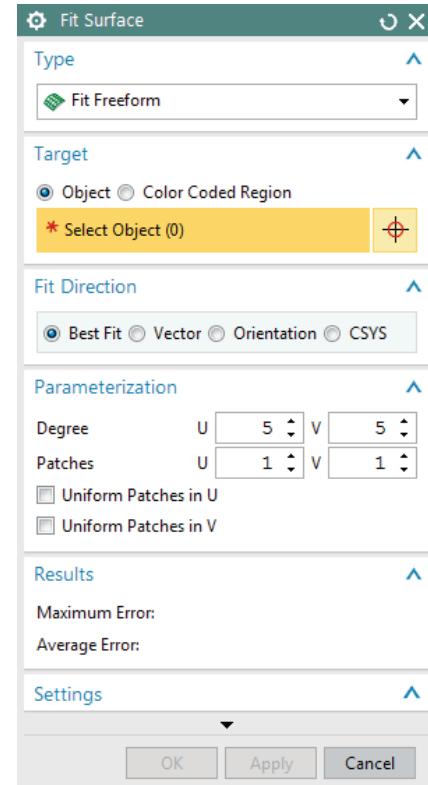
OR

- 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.
- In the **Fit Direction** drop-down menu, choose **Best Fit** for. This matches the point cloud coordinate system with original system
- Change the default values for **U** and **V** degrees to 3
- Click **OK**

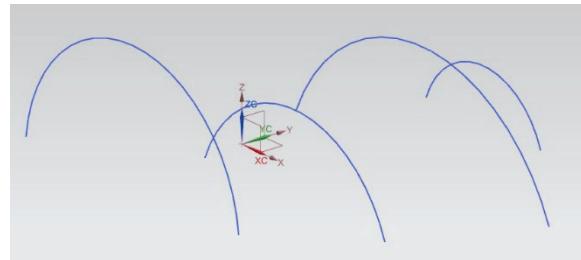
The final sheet will look like the following.



7.2.3 Modeling with Curves

- Open the file named **freeform_thrcurves_parameter.prt**

The curves will be seen as in the figure below.

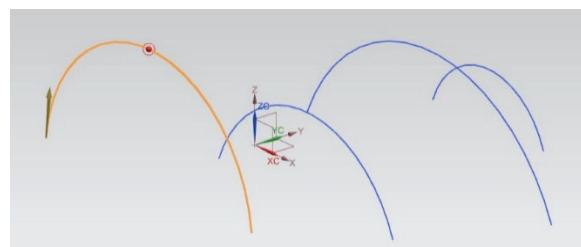


- Choose **Insert → Mesh Surface → Through Curves**

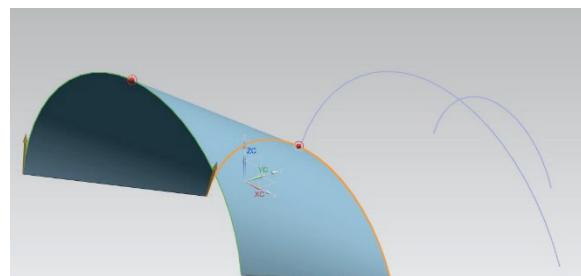
OR

- Click on this Icon  on the Toolbar
- Select the first **section string** as shown below. Be sure to select somewhere on the left side of the arc.

A direction vector displays at the end of the string.



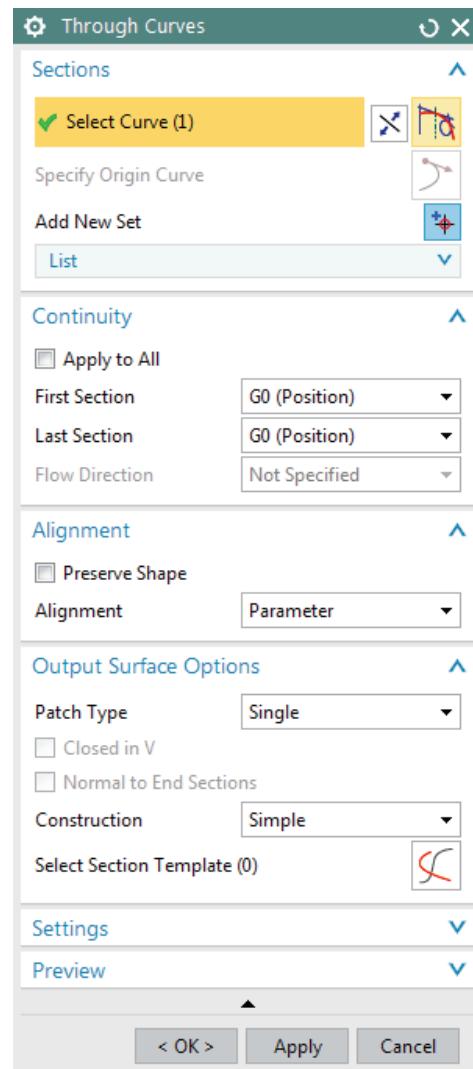
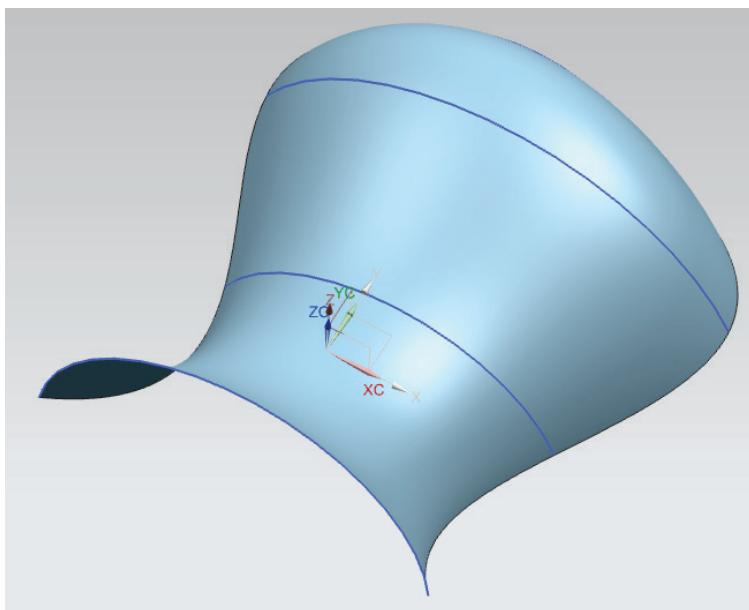
- Click the middle mouse button **MB2**
- Click on the next curve similar to first one and click the middle mouse button **MB2**. You can see a surface generated between the two curves as shown in the figure



- Repeat the same procedure to select the remaining strings. Remember to click **MB2** after selecting each curve.
- For **Alignment**, choose **Parameter**
- For **Patch Type**, choose **Single**
- 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**

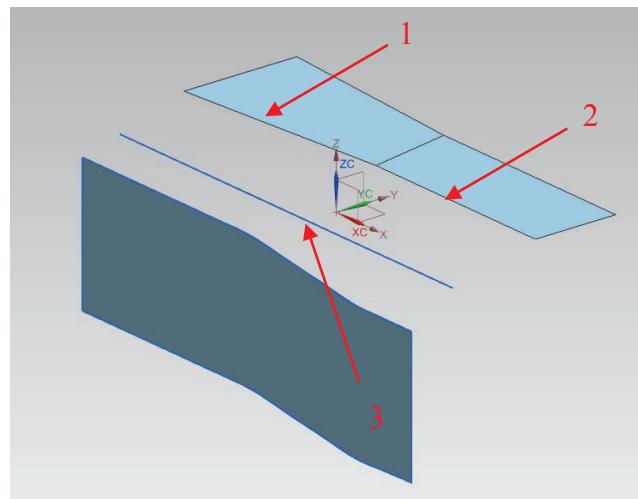


7.2.4 Modeling with Curves and Faces

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

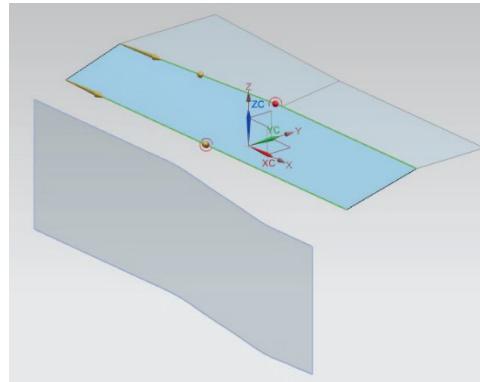
The curve and faces will be seen as follows.

- Choose **Insert → Mesh Surface → Through Curves**
- Select the left edge of the top plane



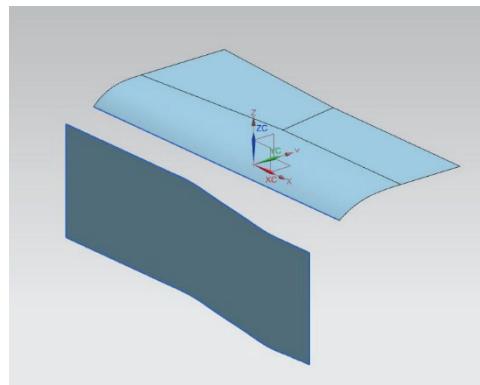
- Select the middle edge and click **MB2**
- Select the line
- In the Dialog box, under the **Alignment** section, uncheck the **Preserve Shape** check box

You would get the following shape displayed on screen.



Make sure that all the arrows are pointing in the same direction.

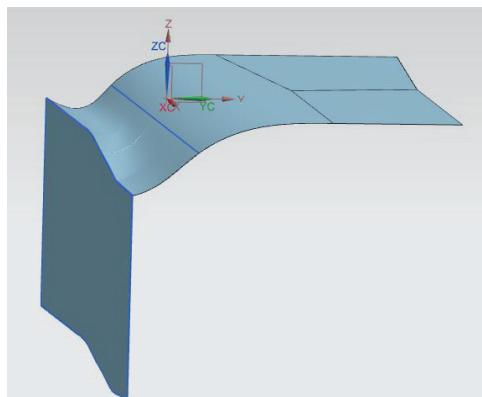
- In the **Alignment** dialog box choose **Parameter**
- In the **Continuity** dialog box select **G2 (Curvature)** option and select the two faces of the top plane as shown
- Click **APPLY**



- 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.



7.3 EXERCISE

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 – FINITE ELEMENT ANALYSIS

Finite Element Analysis (FEA) is a method 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 NX 10 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.

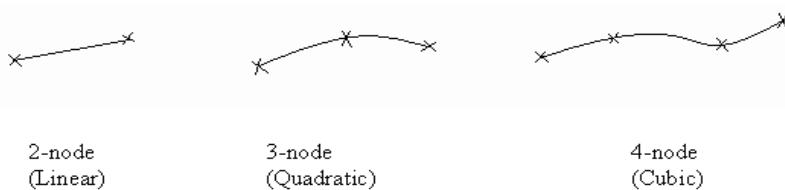
Some of the applications of *FEA* are *Structural Analysis*, *Thermal Analysis*, *Fluid Flow Dynamics*, and *Electromagnetic Compatibility*. Of these, *FEA* is most commonly used in structural and solid mechanics applications for calculating stresses and displacements. These are often critical to the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

8.1 OVERVIEW

8.1.1 Element Shapes and Nodes

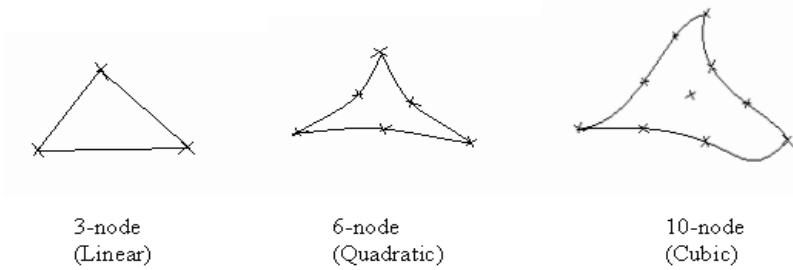
The elements can be classified into different types based on the number of dimensions and the number of nodes in the element. The following are some of the types of elements used for discretization.

One-dimensional elements



Two-dimensional elements

Triangular:

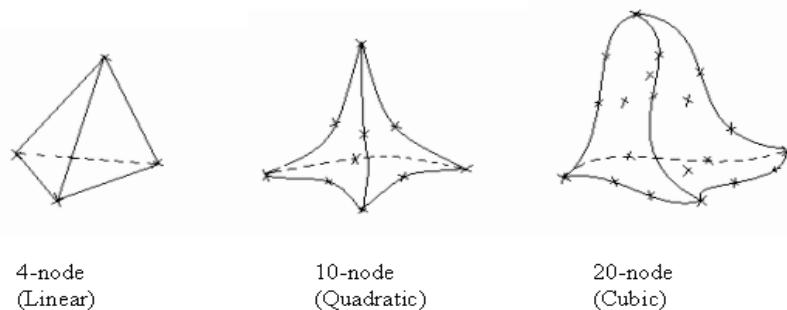


Quadrilateral:

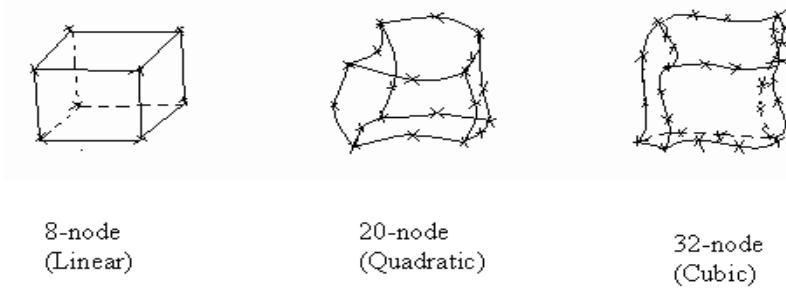


Three-dimensional elements

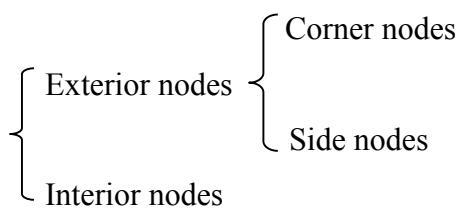
Tetrahedral (a solid with 4 triangular faces):



Hexahedral (a solid with 6 quadrilateral faces):



Types of nodes



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

8.1.2 Solution Steps

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

Choosing the Material Properties: This allows you to change the 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.

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

Applying the 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.

Meshing the Bodies: This is 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.

Solution and Results: 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.

8.1.3 Simulation Navigator

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

To display the *Simulation Navigator*,

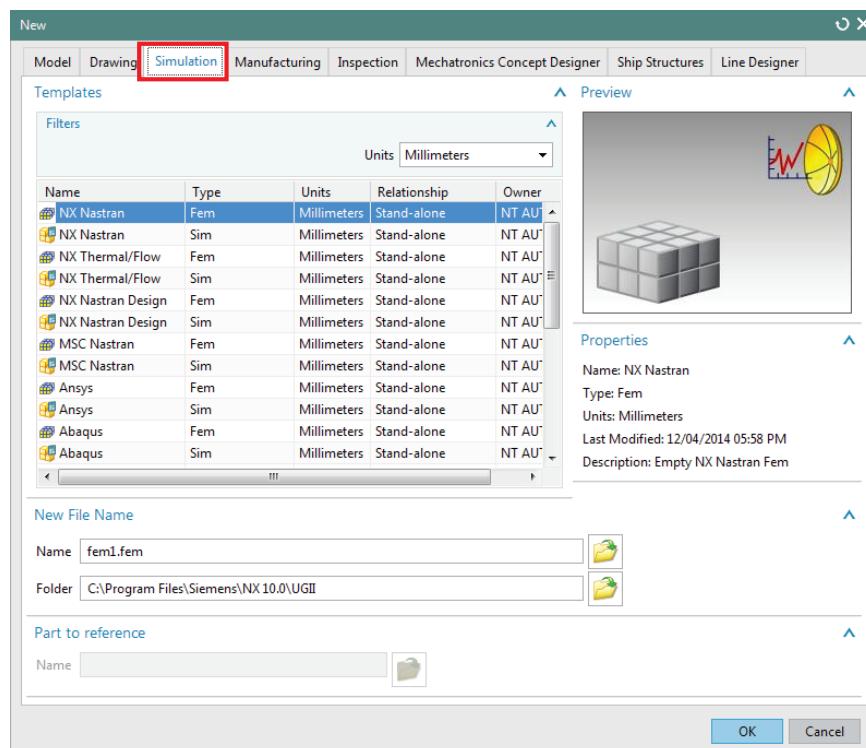
- Click the **Simulation Navigator** tab in the **Resource bar** as shown in the figure

It shows 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.

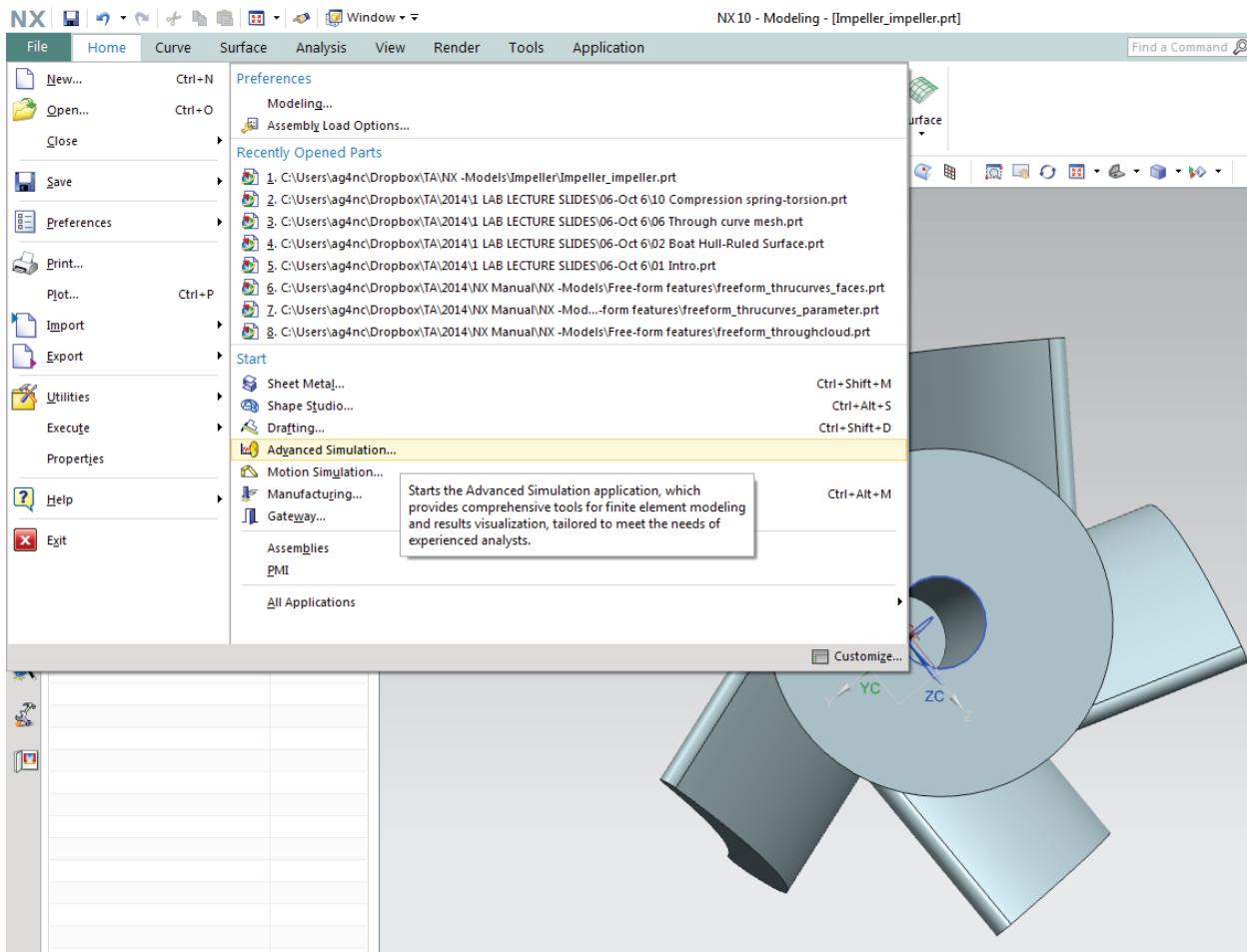
Name	Status
Impellar_impellar_sim1...	
Impellar_impellar...	
Impellar_impellar...	
Polygon Geo...	
Regions	
Simulation Objec...	
Load Container	
Constraint Conta...	
Solution 1	Active
Simulation O...	
Constraints	
Subcase - Static ...	
Loads	

8.2 SCENARIO CREATION

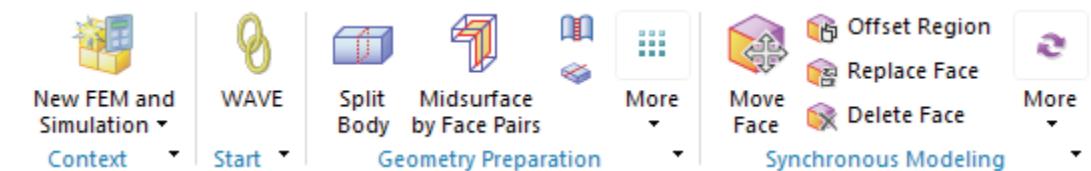
- Copy and paste the file **Impeller_impeller.prt** into a new folder to avoid changes being made to the assembly
- Click on **New → Simulations** if the part is NOT already opened in the NX window



- Open this newly copied file
- If part is already opened in NX, then click on **File → Advanced Simulations**



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



The *Design Simulation* module is different from when the first scenario is created. NX 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 *Solution*, 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 interpart 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.

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

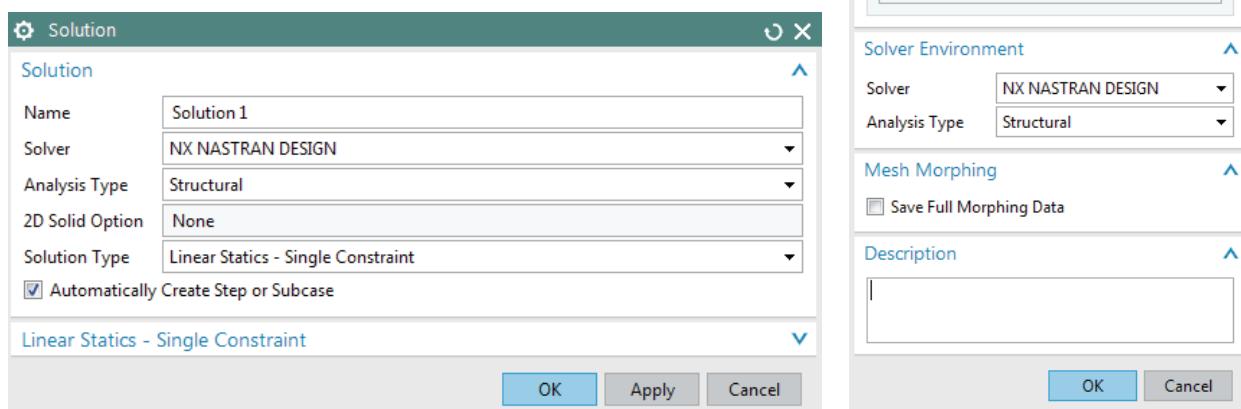
- Click on the *New FEM and Simulation* icon on the



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.

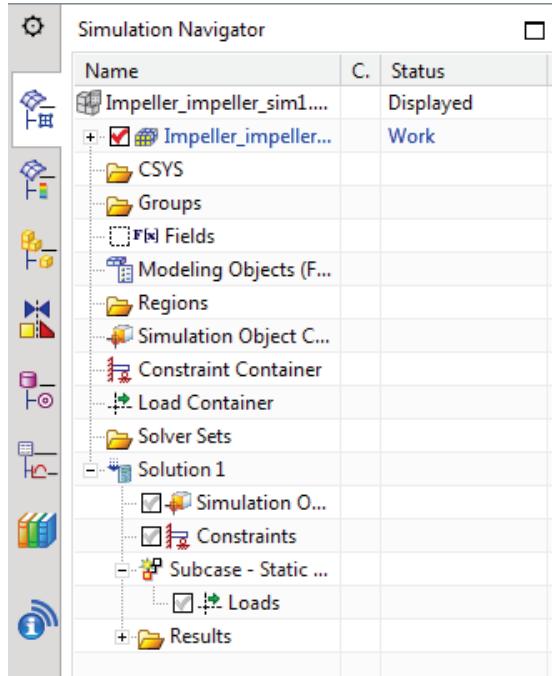


In the *Solution* window, you can select the *Solver* and the *Analysis Type*.

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

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

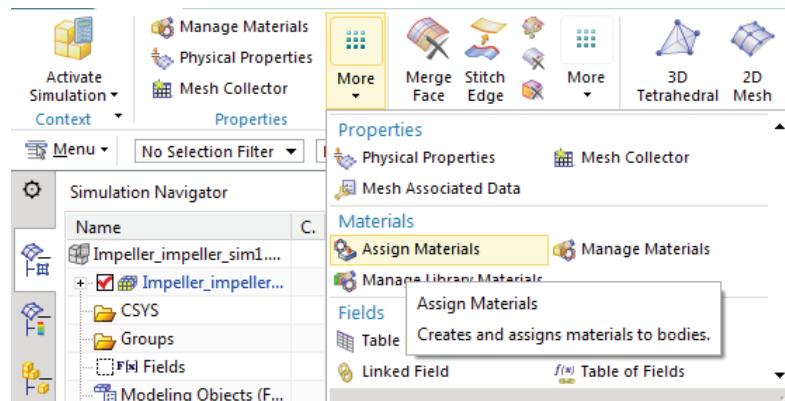
The Simulation Navigator will now look like the following figure.



8.3 MATERIAL PROPERTIES

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

- Click on the **More** icon in **Properties** group on the Toolbar
- Choose **Assign Materials**



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

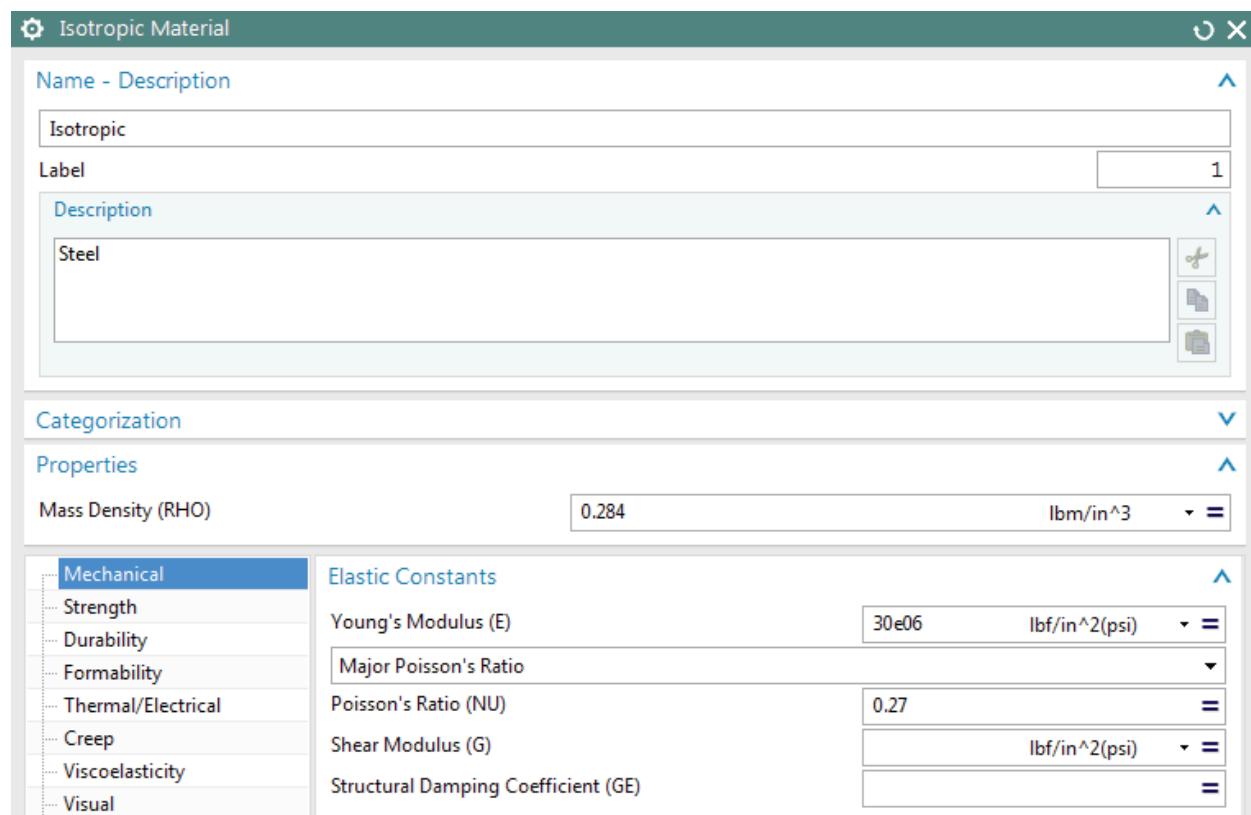
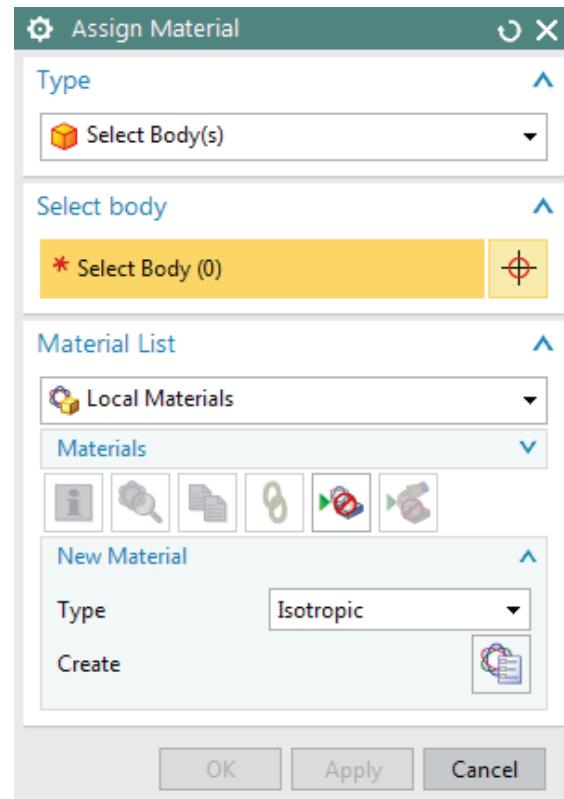
- Select the **Impeller**
- Choose **Local Materials**
- Click on the **Create** icon

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

(Note that 30e6 represents 30×10^6)

- Choose **OK** to exit the **Isotropic Material** window

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



8.4 MESHING

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

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

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

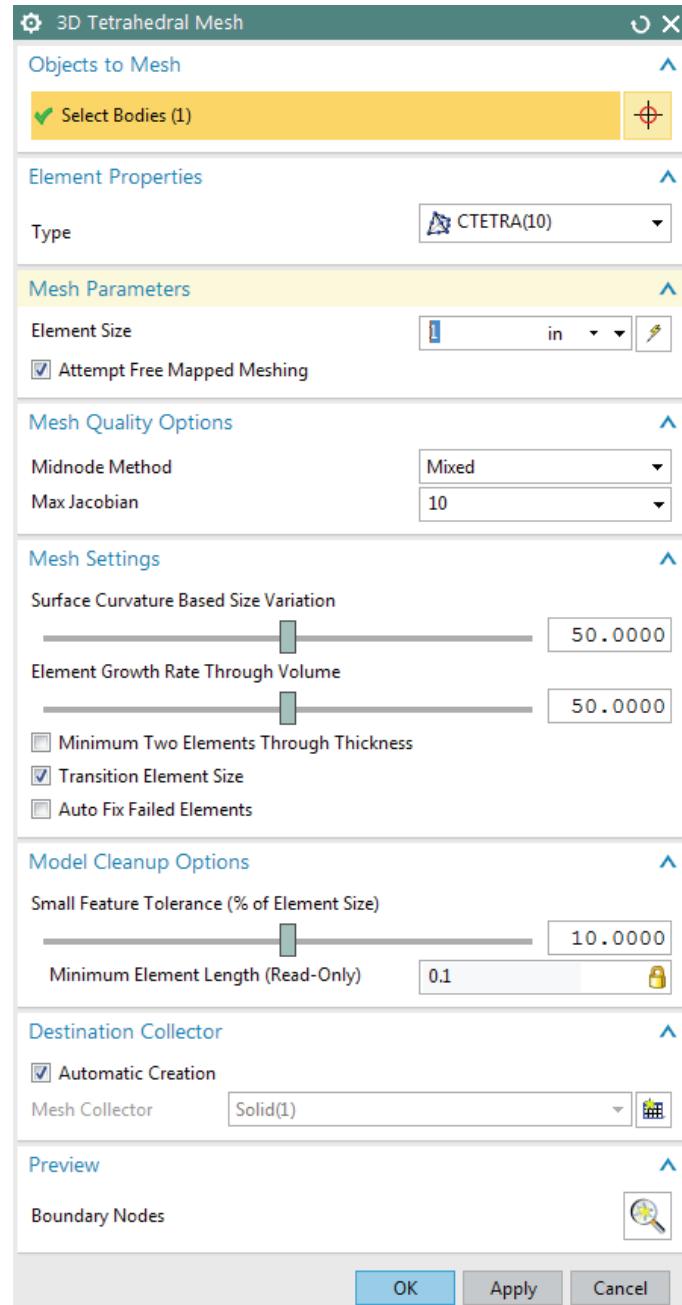
- Click on the solid object model on the graphic screen.

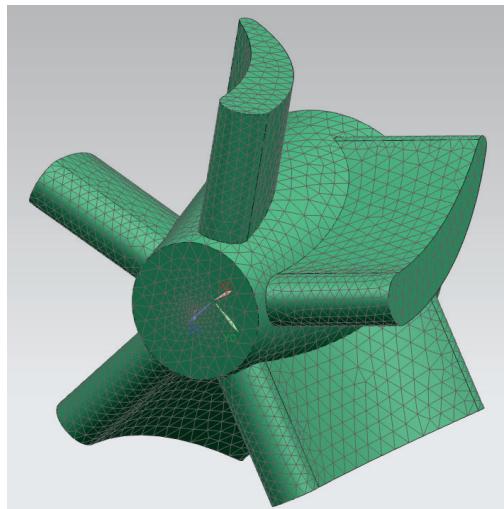
There are two types of *Tetrahedral Elements* available in NX 10. One is 4-nodes and the other is 10-node.

- Choose the **Type** to be **TETRA10**
- Enter the **Overall Element Size** as **1.0**
- Choose **OK**

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

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

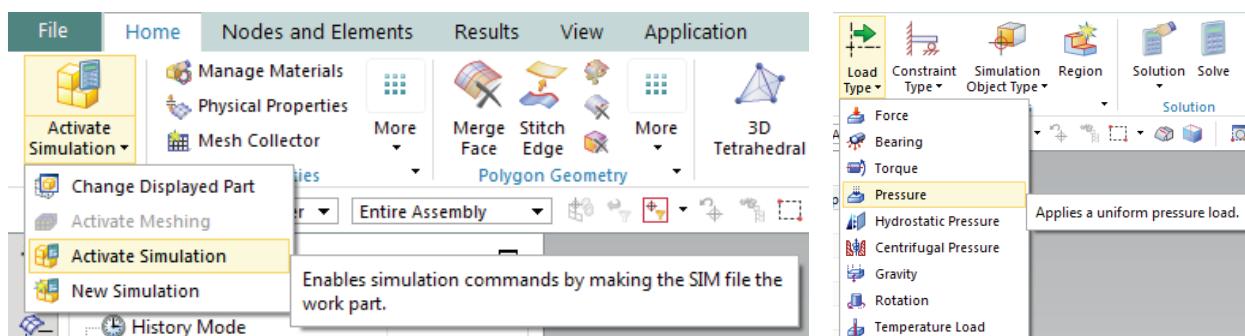




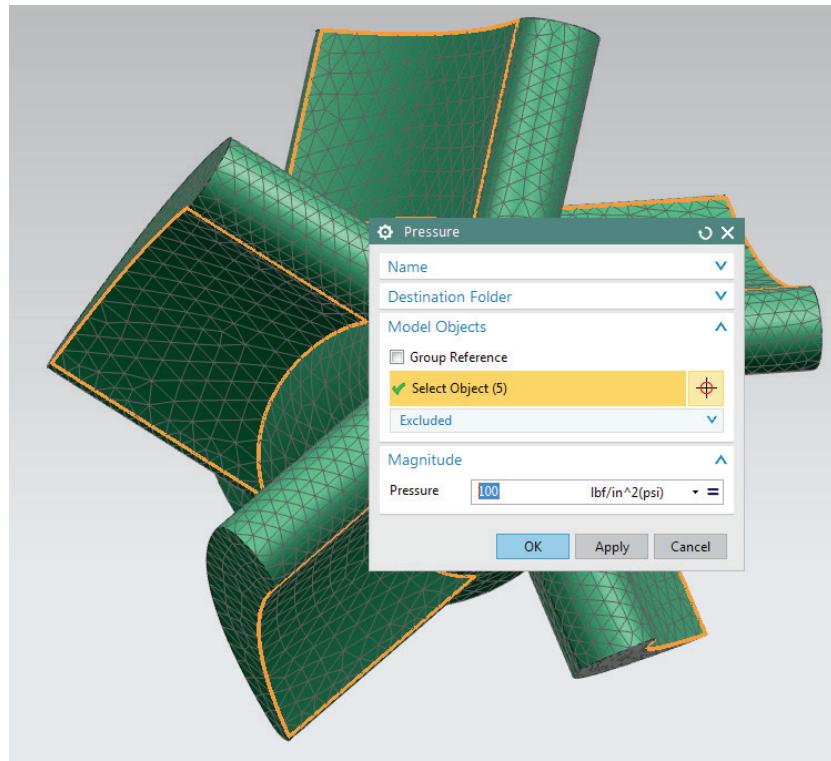
8.5 LOADS

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 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.

- Click on the **Activate Simulation** to apply loads as shown below
- Click on **Load Type** and choose **Pressure**



- Click on the five concave surfaces of the blades as shown in the following figure
- Enter the value for **Pressure** as **100** and keep the units as lb-f/in² (psi)



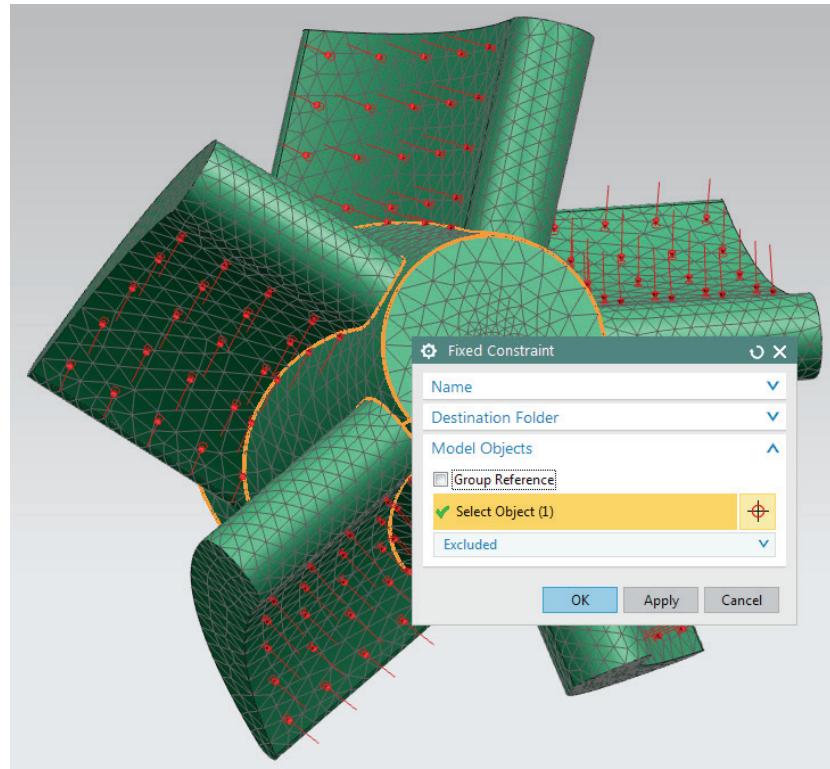
8.6 BOUNDARY CONDITIONS

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

- Click on the **Constraint Type** icon
- Select the **Fixed Constraint**

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

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



8.7 RESULT AND SIMULATION

8.7.1 Solving the Scenario

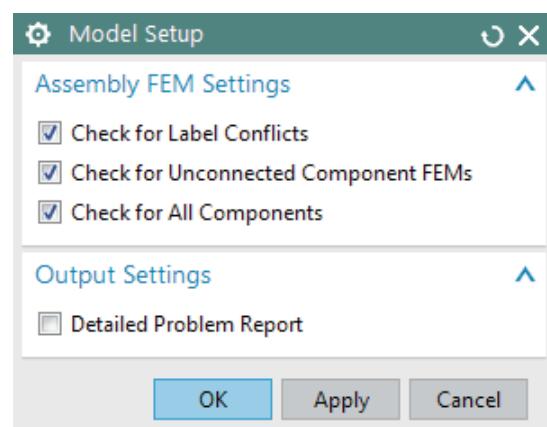
The Finite Element Model is now ready for solving and analysis. It is a good practice to first check for model completion before we get into solving the model. To check the model

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

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. In case you get errors or warnings go back to the previous steps and complete the required things. If you do not get errors or warnings you are ready to solve the FEA problem.



```

File Edit
=====
Information listing created by : ag4nc
Date : 15/10/10 21:24:00
Current work part : C:\Users\ag4nc\Dropbox\TA\NX -Models\Impeller\Impeller_sim1.sim
Node name : r15mleu
=====

Mesh-Based Errors Summary
-----
Total: 0 errors and 0 warnings

Material-Based Errors Summary
-----
Total: 0 errors and 0 warnings

Solution-Based Errors Summary
-----
Iterative Solver Option
More than 80 percent of the elements in this model are 3D elements.
It is therefore recommended that you turn ON the Element Iterative Solver in the "Edit Solution" dialog.

Total: 0 errors and 0 warnings

Load/BC-Based Errors Summary
-----
Total: 0 errors and 0 warnings

NASTRAN Model Setup Check completed

```

- Click on the **Solve** icon 

This will open the *Solve* window.

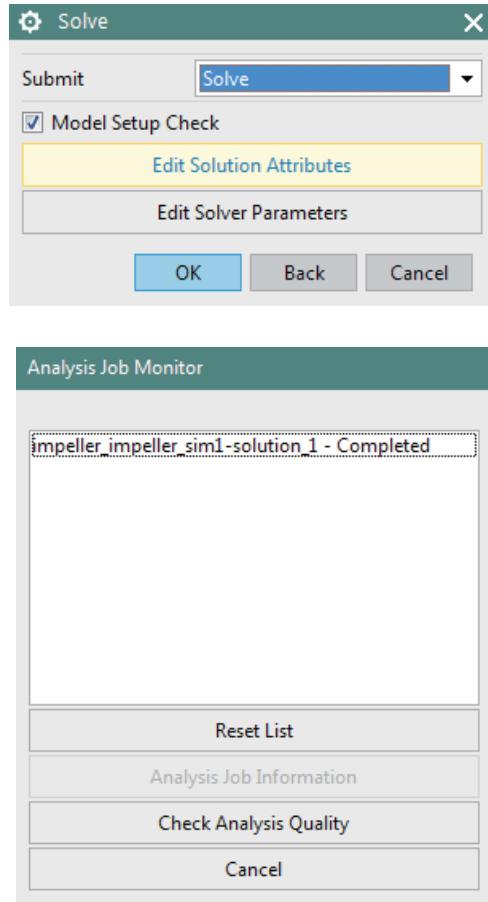
- Click **OK** without making any changes

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*

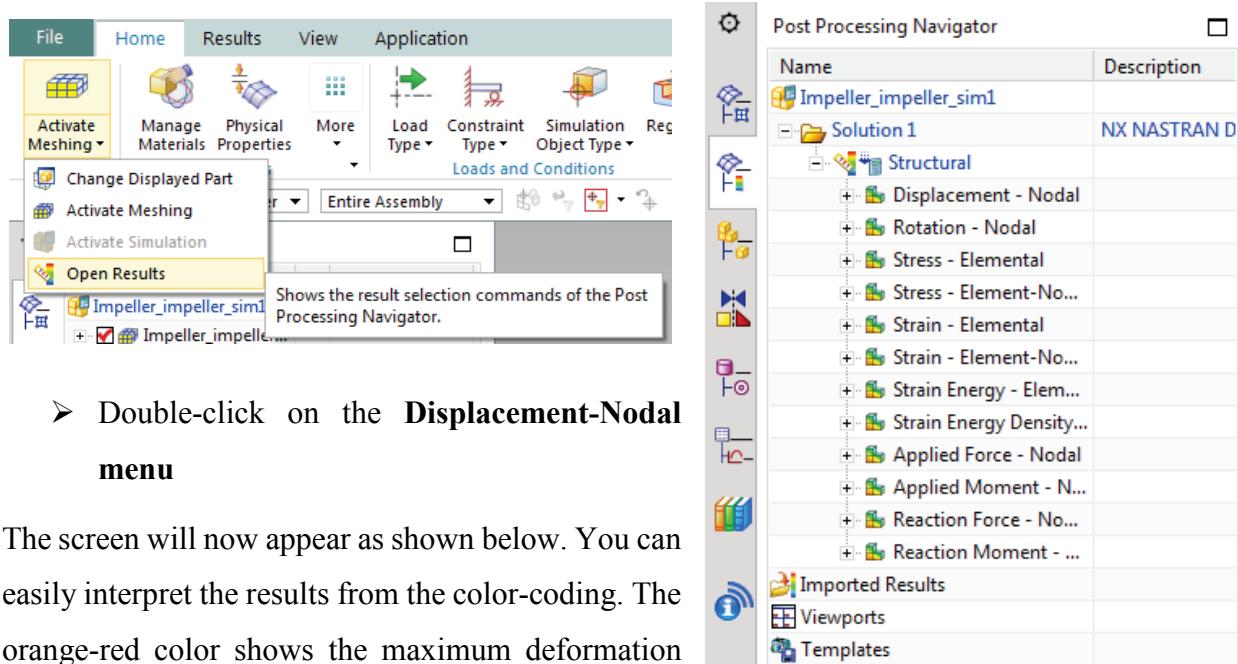
- Click on **Cancel** when the **Analysis Job Monitor** window shows **Completed**

8.7.2 FEA Result

- Click on **Open Results**
- Click on the **Post Processing Navigator**

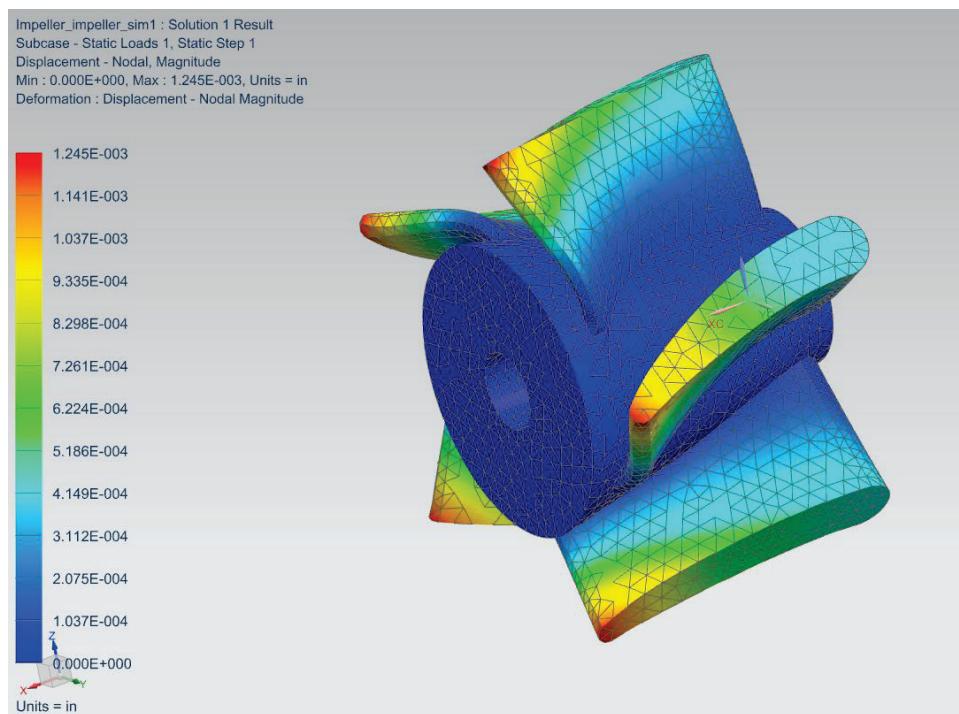


The *Post-Processing Navigator* shows all the *Solution* you created. If you click the '+' sign in front of the *Solution* you will see the different analyses that have been performed on the model.

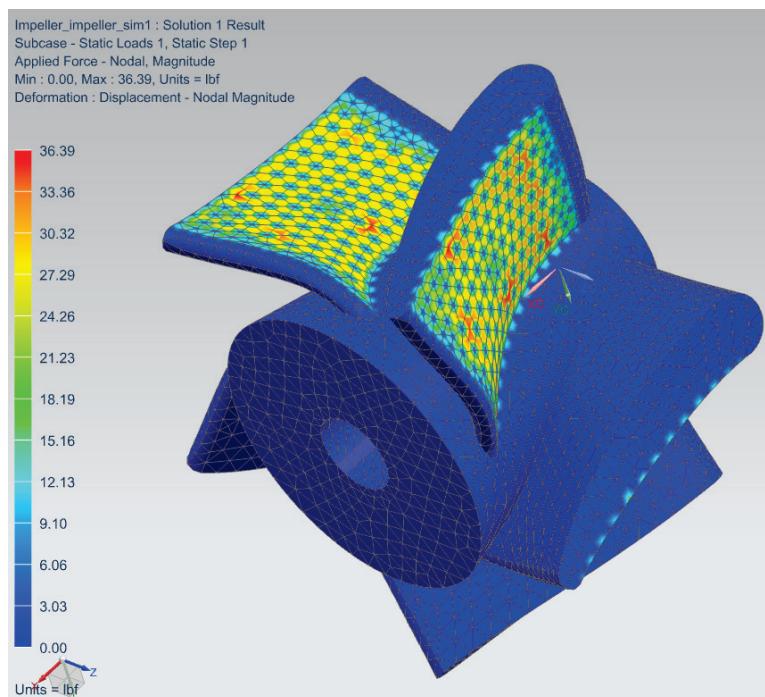
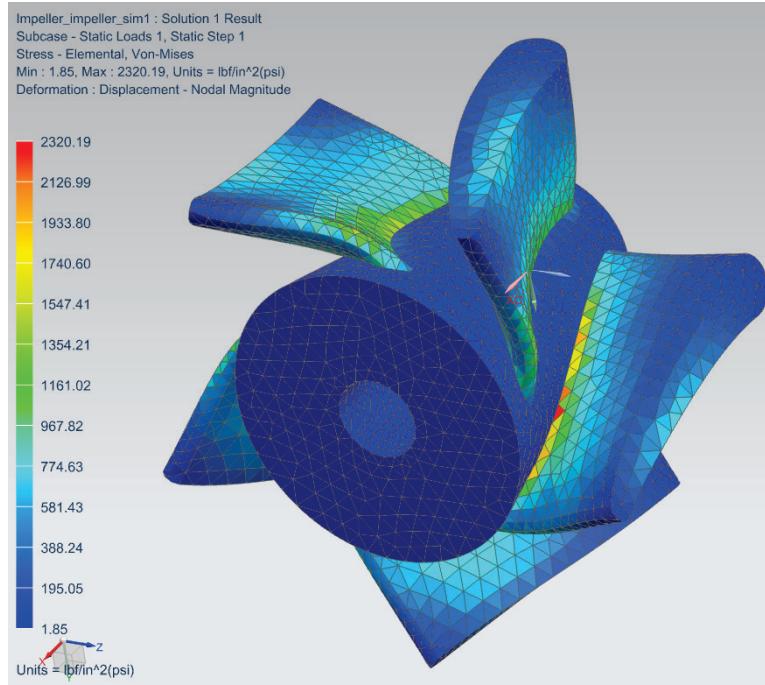


- Double-click on the **Displacement-Nodal** menu

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.



8.7.3 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.

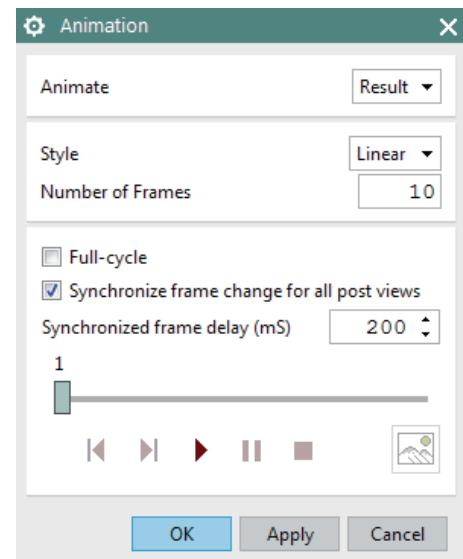
- Click on the **Results** tab. A group for **Animation** can be seen on it as follows



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

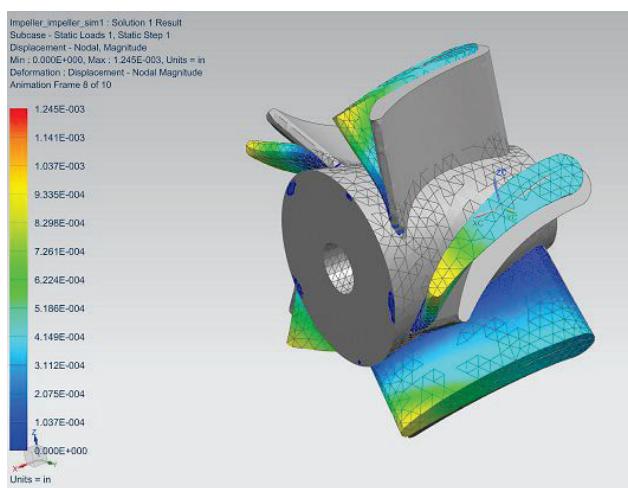
You can now see an animation of how the impeller is deformed as the loads are applied to the blades.

- To make any setting changes in the results display, click on the **Edit Post View** icon
- Check the **Show undeformed model** and click **OK**



Now press on the **Play** button to see the animation.

This will show the animation of deformation with the original shape in grey color, as shown in the figure below.



- Click on the **Stop** button
- Right-click on the **Return to Home** to go back to the meshed model

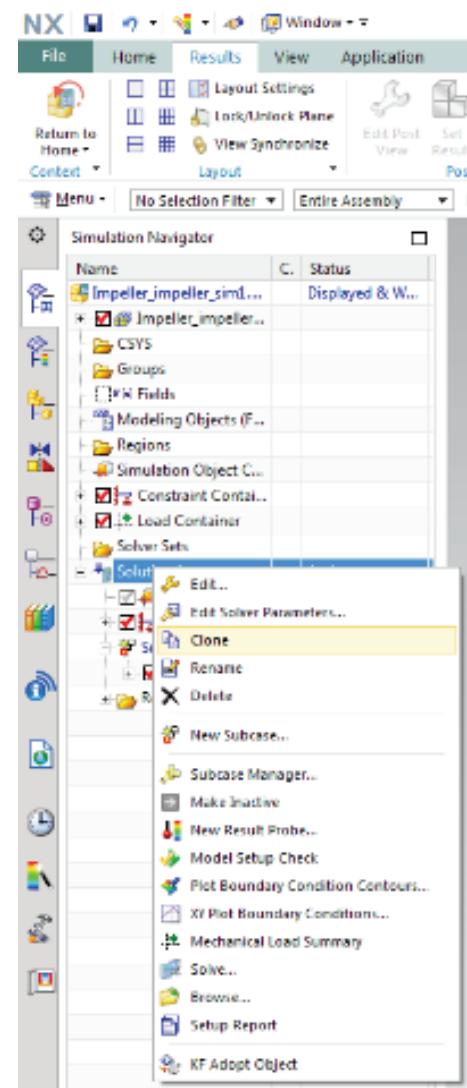


There are two ways to improve the accuracy of FEA results.

- 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 **Solution 1** in the **Simulation Navigator**



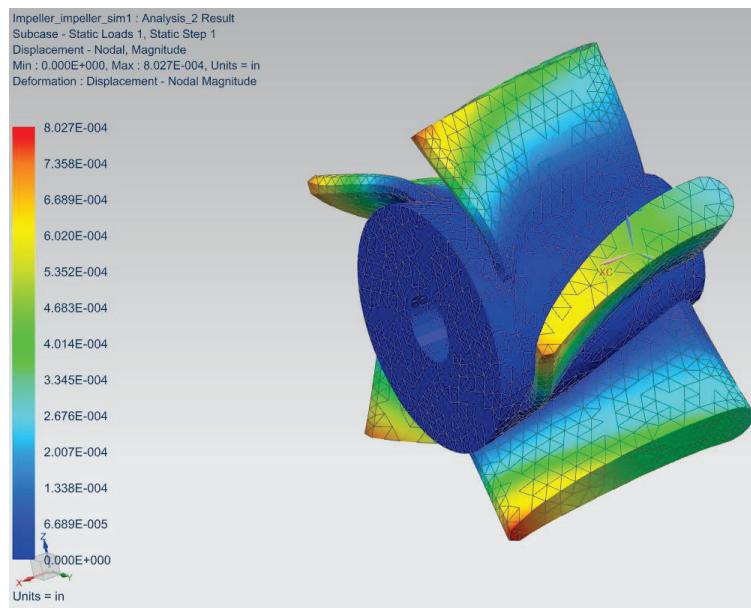
- Choose **Clone** to copy the first scenario
- Once **Copy of Solution 1** is created, rename it to **Solution 2**
- Go to **.fem1** file in the **Simulation File View**
- Right click on the **3D Mesh (1)** and click **Edit**
- In the dialog box shown, change the **Type** to **TETRA4**
- Choose **OK**
- 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 *Solution 2* to be *Completed*.

- Click **Cancel**
- In the **Simulation Navigator**, double-click on **Results for Solution 2**

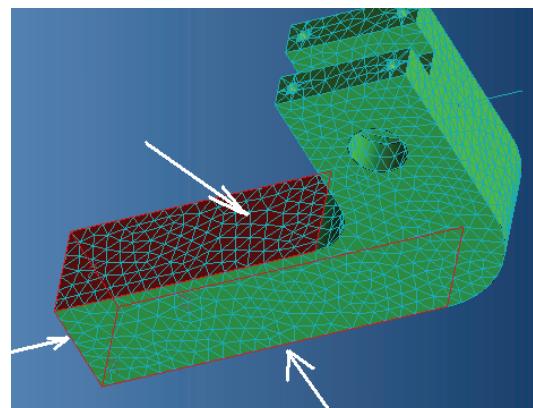
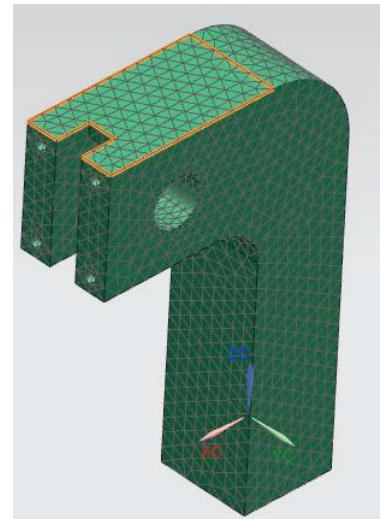
The figure below shows the analysis. You can observe the change in the maximum deviation. Save all the scenarios and close the files.



8.8 EXERCISE

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.

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 9 – 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 10 to generate CNC codes for *3-Axis Vertical Machining Centers*. This module allows you to program and do some post-processing on drilling, milling, turning and wire-cut EDM tool paths.

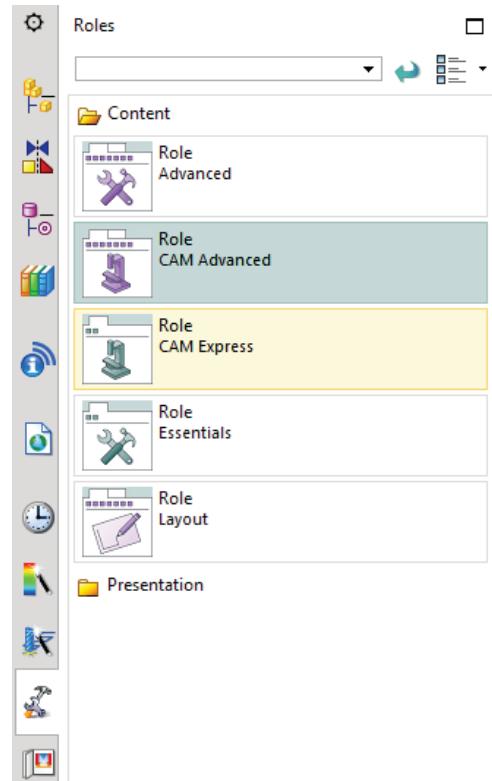
9.1 GETTING STARTED

A few preparatory steps need to be performed on every CAD model before moving it into the CAM environment. 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*. A drop-down menu will pop up in which the *CAM Advanced* role can be seen as shown in the figure.

9.1.1 Creation of a Blank

After completing the modeling, you should decide upon the raw material shape and size that needs to be loaded on the machine for the actual machining. This data has to be input in NX 10. 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 4

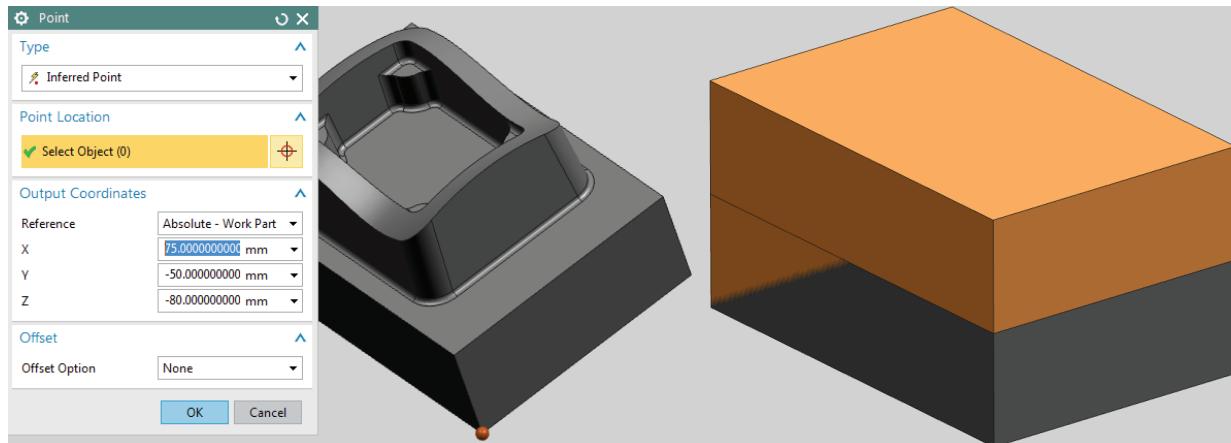
➤ **Insert** a block with the following dimensions and positioning

Length = **150** mm

Width = **100** mm

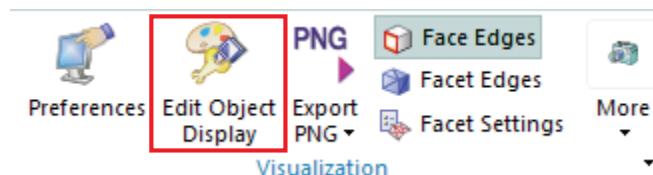
Height = **80** mm

➤ 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 as shown



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

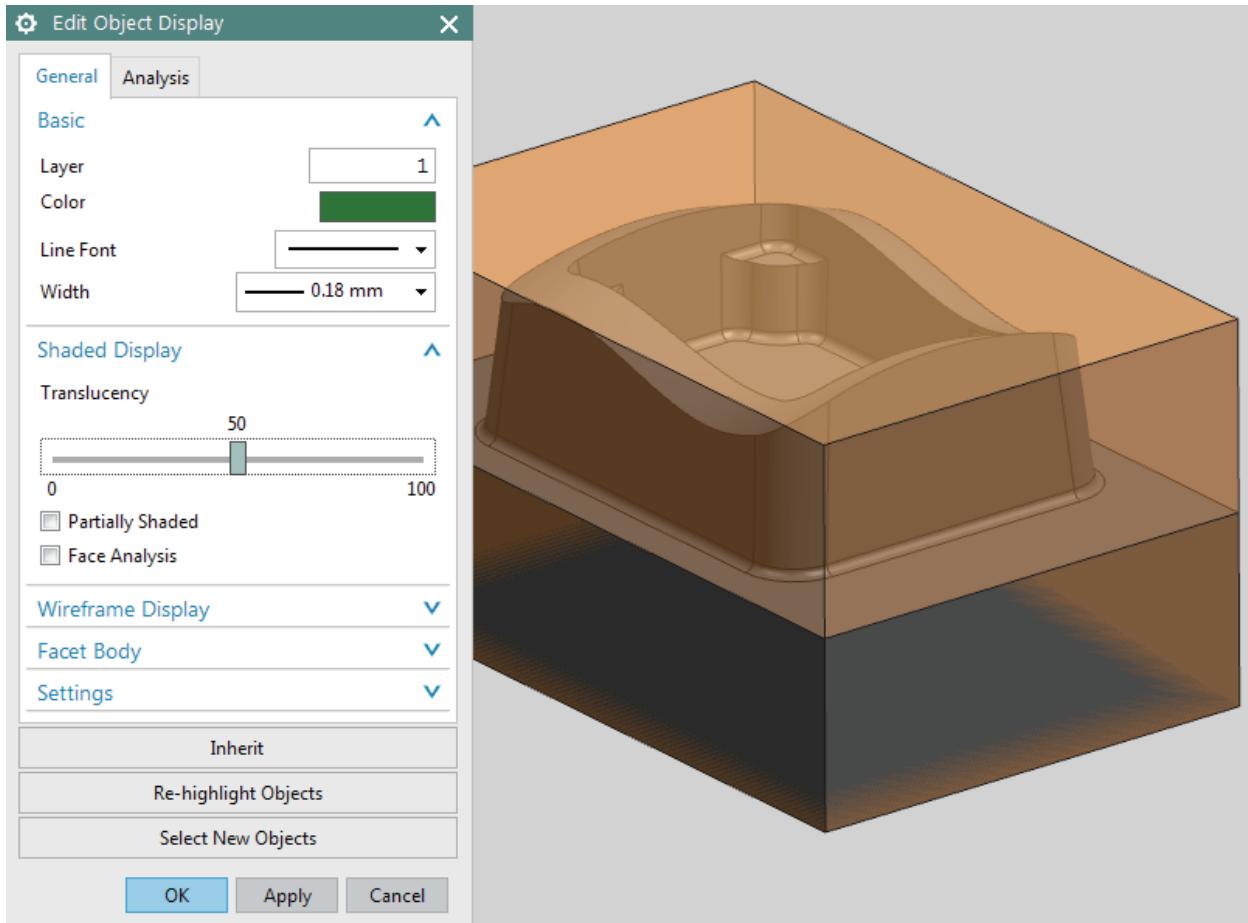
➤ Click on the **Edit Object Display** icon in the **Visualization** group of the **View** tab



➤ Select the block you created and click **OK**

➤ When the window pops up, change the display **Color** and change the **Translucency** to **50**

➤ click **OK**

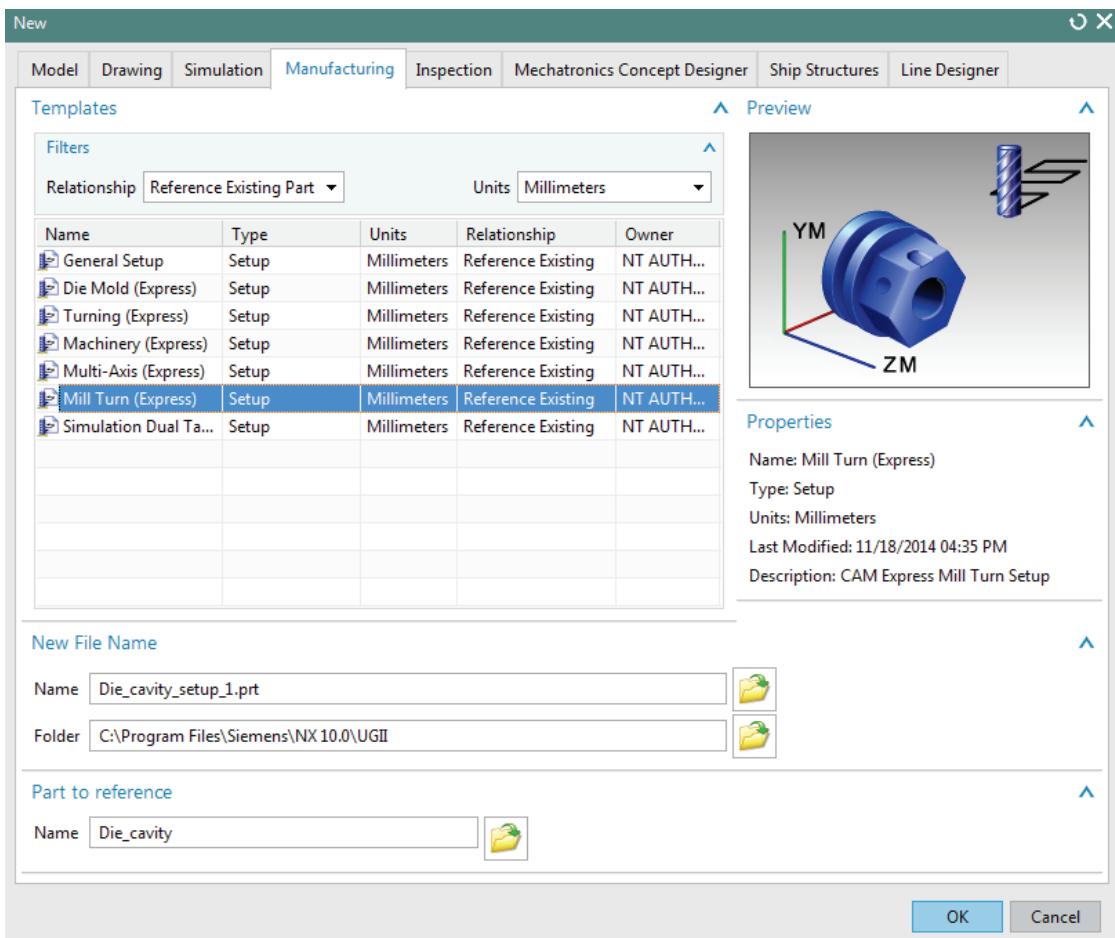


- **Hide** the block you just created by right clicking on the block in the **Part Navigator**. This will make the raw block disappear from the environment. Whenever you want to view or work on this solid, reverse the blanks. This is done by pressing **<Ctrl> + <Shift> + B**.

9.1.2 Setting Machining Environment

Now we are set to get into the Manufacturing module.

- Select **File → New → Manufacturing → Mill Turn**
- There are many different customized CAM sessions available for different machining operations. Here, we are only interested in the milling operation.



9.1.3 Operation Navigator

As soon as you get into the *Manufacturing* environment, you will notice many changes in the main screen such as new icons that are displayed.

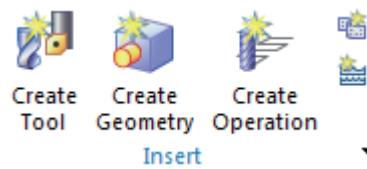
- Click on the **Operation Navigator** tab on the right on the **Resource Bar**

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**

9.1.4 Machine Coordinate System (MCS)

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

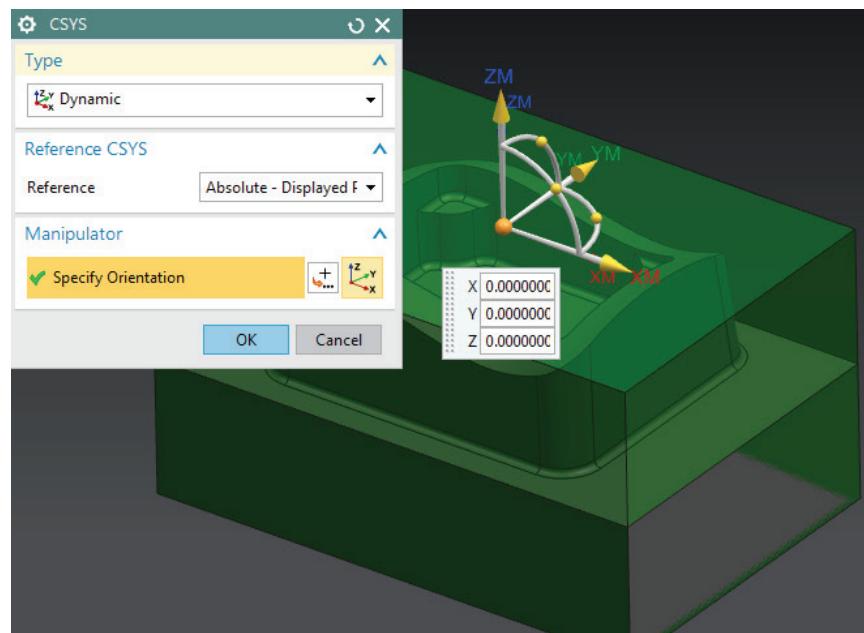
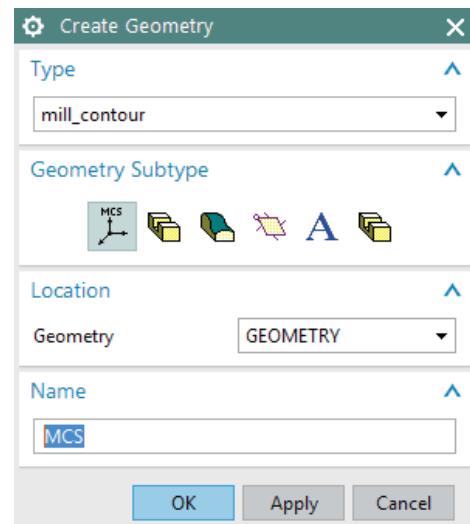


You will see a *Create Geometry* pop-up. You should be able to see the *mill_contour* as the program name in the *Operation Navigator*. If you do not see it, click on the *Geometry View* button in the *Toolbar* again.

- Click **OK**

Another pop-up window will allow you to set the *MCS* wherever you want. By default NX 10 takes the original AbsoluteCS as the *MCS*.

- Click on the **CSYS** button in **Specify MCS**. This will highlight the default WCS of the part and assign it as the **MCS**
- Click **OK** to select it as the **MCS**



9.1.5 Geometry Definition

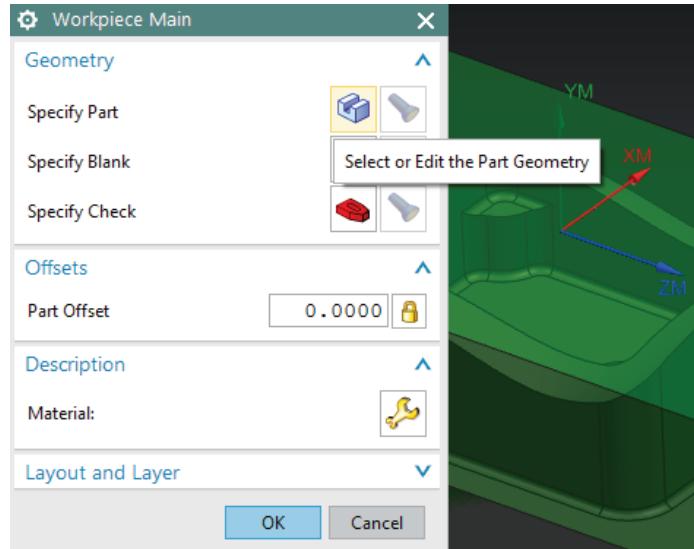
- Click on **Geometry View**

- Expand **MCS_MAIN_SPINDLE** by clicking on the plus signs in the **Operation Navigator**
- Double-click on **WORKPIECE_MAIN** in the **Operation Navigator**. If you do not see it, click on other plus signs

The pop up window *Workpiece Main* 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 several 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 block we can use that as the *Blank* geometry.

- Click on the **Block** and press **OK**

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

9.2 CREATING OPERATION

9.2.1 Creating a New Operation

The manufacturing setup is now ready for us to work further with *Programming Strategies*. There are many different manufacturing strategies involved in programming and it takes practice to know

which one is 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 

The *Create Operation* window will pop up.

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

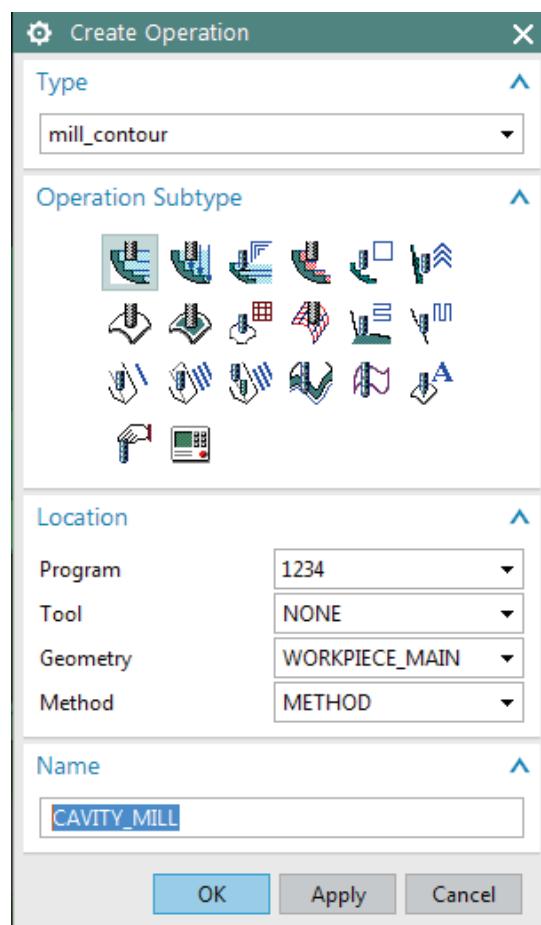
There are many different subtypes under *Mill-Contour*, namely *Cavity Mill*, *Z-Level Follow Cavity*, *Follow Core*, *Fixed Contour*, and so on. These different subtypes are used for different situations and profiles of the design part. As mentioned before, how you select a strategy for any situation 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 **1234**
- Change the **Geometry** to **WORKPIECE_MAIN**
- Click **OK**

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

9.2.2 Tool Creation and Selection

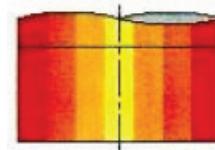
One of the most important decisions to make is to select the right shape and size of the tool to use. Before starting with the *Tool parameter* settings, we must first know about the types of *Tool cutters*. The *Milling Tool Cutters* are categorized into three forms of cutters. Hence, when selecting 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.



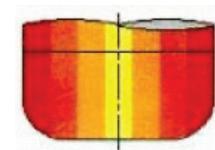
Ball End Mill

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

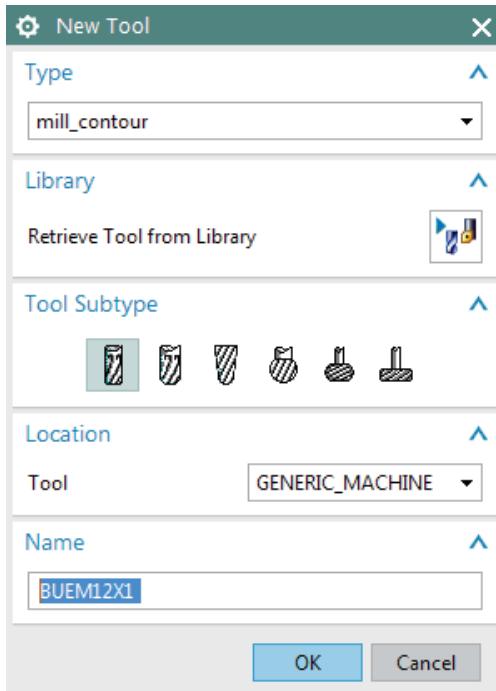
These cutters have small corner radii and are widely used for roughing and/or semi-finishing the parts as well as for finishing of inclined and tapered walls.



The cutter that we are going to use to rough out this huge volume is BUEN12X1 (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

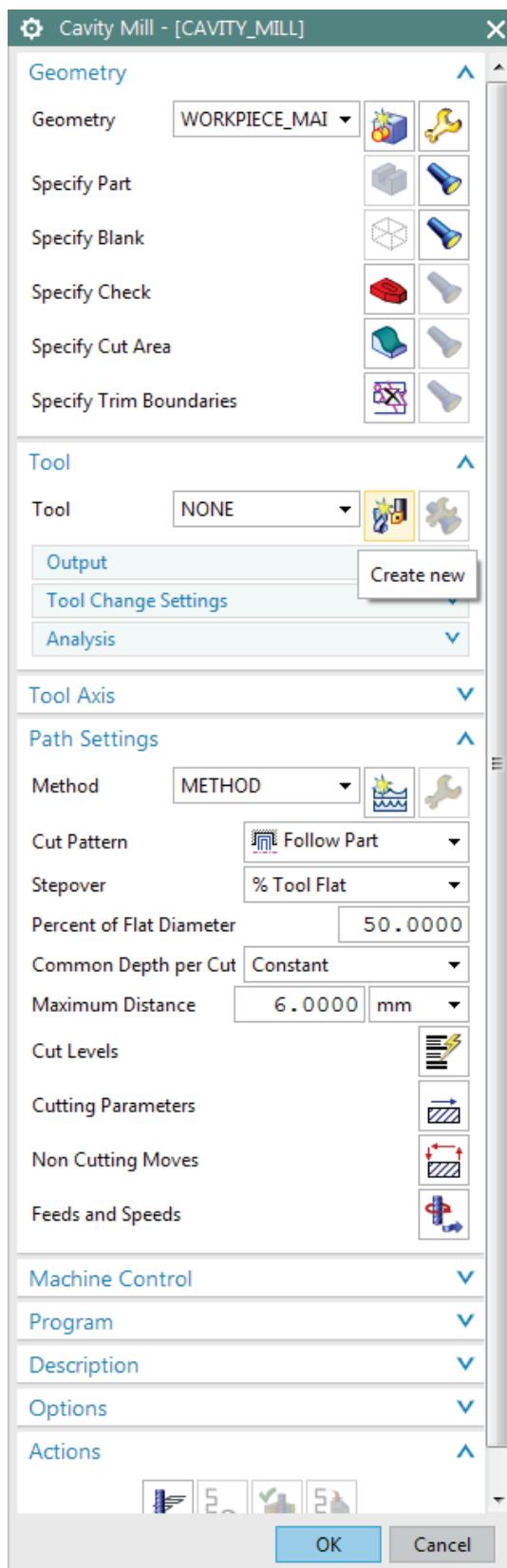


- Type in **BUEM12X1** as the **Name** and click **OK**

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

Dimensions	
(D) Diameter	12.0000
(R1) Lower Radius	1.0000
(B) Taper Angle	0.0000
(A) Tip Angle	0.0000
(L) Length	75.0000
(FL) Flute Length	50.0000
Flutes	2

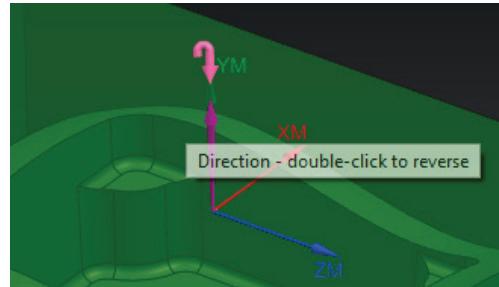
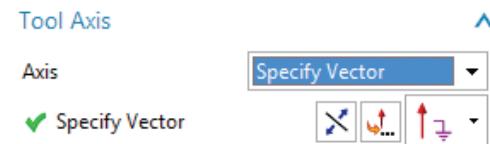


- Click OK

9.2.3 Tool Path Settings

Make sure that the *Tool Axis* is perpendicular to the top surface on the block.

- Click on **Tool Axis** and choose **Specify Vector**
- Select the appropriate axis as shown
- In the **Cavity Mill** menu click on the **Path Settings** option



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



Follow Part: This is the most optimal strategy where

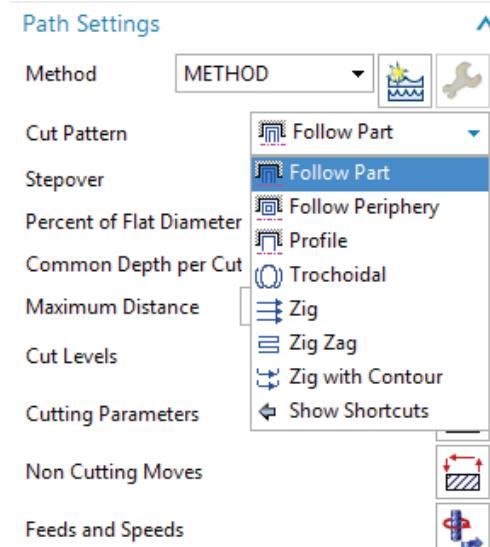
the tool path is manipulated depending on the part geometry. If there are cores and cavities in the part, the computer intelligently considers them to remove the materials in an optimal way. This is widely used for roughing operations.



Follow Periphery: This takes the path depending upon the periphery profile. For example, 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.



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



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

 **Zig Zag:** This tool takes a zigzag path at every level of depth. It saves time by reducing amount of air cutting time (idle running). The climb and conventional cuts alternate.

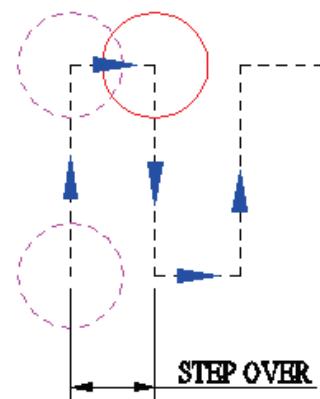
 **Zig with Contour:** This takes the path in one direction either climb or conventional. The unique thing is that it moves along the contour shape nonlinearly.

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

9.2.4 Step Over and Scallop Height

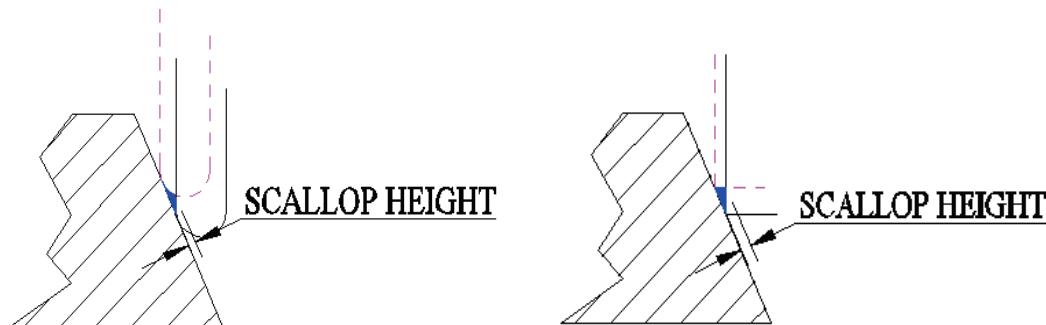
Step Over

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



Scallop Height

Scallop Height controls the distance between parallel passes according to the maximum height of material (scallop) you specify to be left between passes. This is affected by the cutter definition and the curvature of the surface. *Scallop* allows the system to determine the *Stepover* distance based on the scallop height you enter.



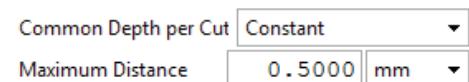
- For the **Stepover**, select **%Tool Flat** and change the **Percent** to **70**



9.2.5 Depth Per Cut

This is the value to be given between levels to slice the geometry into layers and the tool path cuts as per the geometry at every layer. The cut depth value can vary for each level. Levels are horizontal planes parallel to the XY plane. If we do not give cut levels, the software will unnecessarily try to calculate slices for the entire part and machine areas that are not in our interest.

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

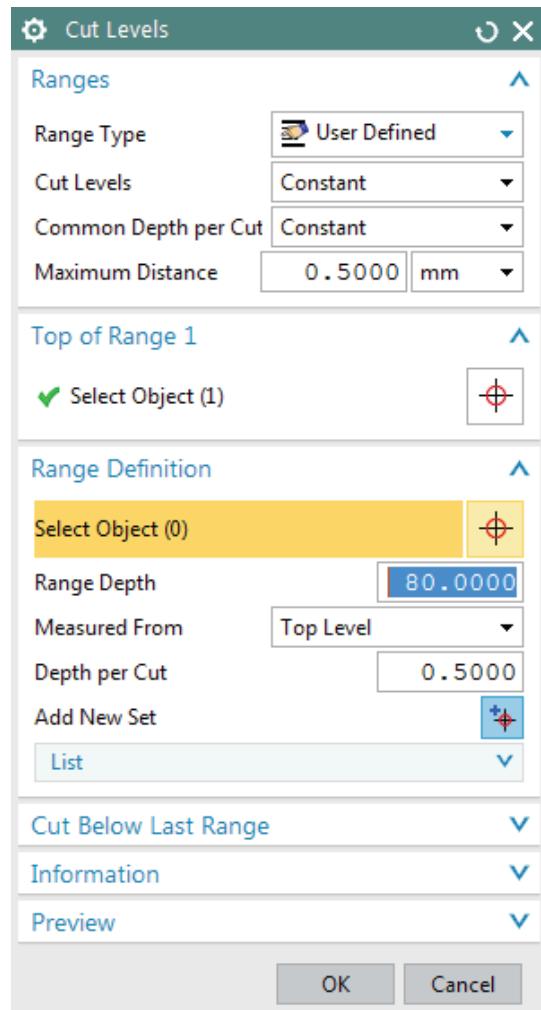


Now we will add the level ranges. This will split the part into different levels along the Z-direction to be machined.

- Click on **Cut Levels**

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. We are not going to mill up to the bottommost face of the part, but up to the floor at 40 mm from top. Therefore, we must delete the last level.

- Change the **Range Type** to **User Defined**
- Change the **Range Depth** to **80**
- Select **OK**



9.2.6 Cutting Parameters

- On the **Path Settings** menu, click **Cutting Parameters**
- Under the **Strategy** tab button, change the **Cut Order** from **Level First** to **Depth First**

Changing the cut order to *Depth First* orders the software to generate the tool path such that it will mill one island completely up to the bottom-most depth before jumping to another 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*.

- Click **OK**

9.2.7 Avoidance

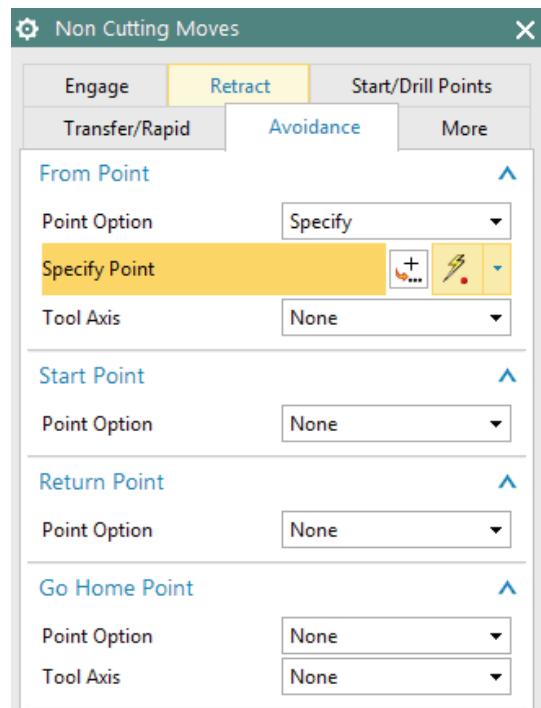
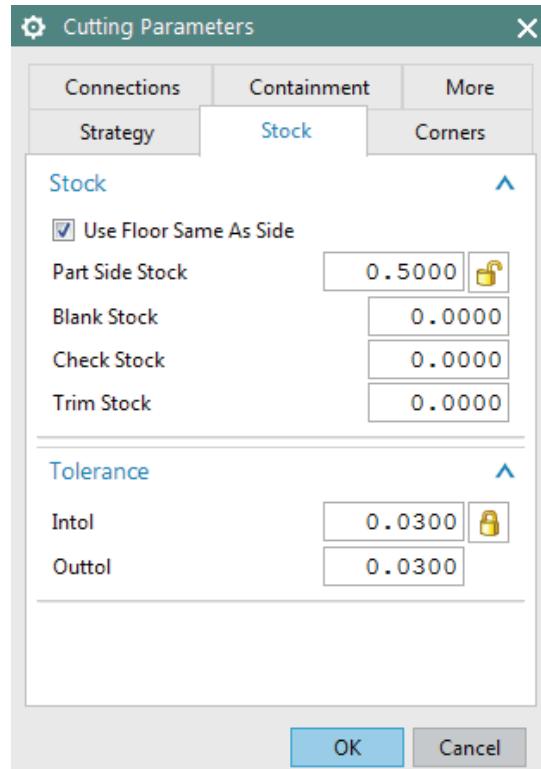
- Click the **Non Cutting Moves**
- Click the **Avoidance** tab

This window consists of several avoidance points of which we are concerned with the following points:

From Point

This is the point at which the tool change command will be carried out. The value is normally 50 or 100 mm above the Z=0 level to enhance the safety of the job when the cutter is changed by the Automatic Tool Changer (ATC).

- Click **From Point**
- Choose **Specify** in the **Point Option** field
- In the **Point Dialog**, enter the coordinates of XC, YC and ZC as **(0, 0, 50)**



- Choose **OK**

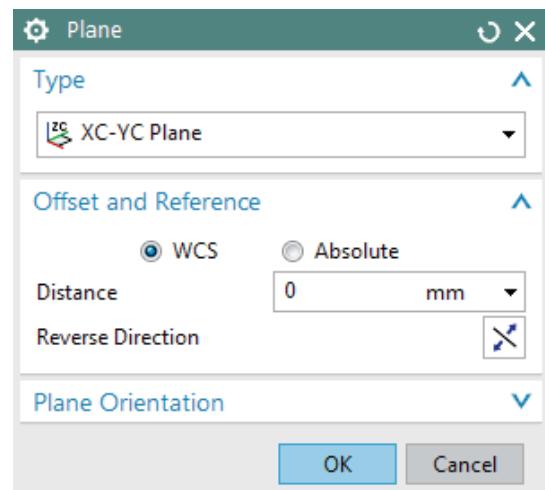
Start Point

This is the point at which the program starts and ends. This value is also 50 or 100 mm above the Z=0 level to enhance safety. It is also the point at which the machine operator checks the height of the tool mounted on the spindle with respect to the Z=0 level from the job. This cross checks the tool offset entered in the machine.

- Click on **Start Point**
- Choose **Specify**
- Enter the coordinates **(0, 0, 50)** in the **Point Dialog**
- Click **OK** to exit the **Point Constructor**

Clearance Plane is the plane on which the tool cutter will retract before moving to the next region or island. This is also known as *Retract Plane*. Sometimes the *Clearance Plane* can be the previous cutting plane. However, when the tool has to move from one region to another, it is necessary to move to the *Clearance Plane* before doing so. The value of the *Clearance Plane* should be at least 2 mm above the 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 **XC-YC Plane** from the drop-down menu in **Type** tab
- Under the **Offset and Reference** tab enter the value as **3** as the **Distance**
- Click **OK** twice to go back to the **Cavity Mill** parameters window



9.2.8 Speeds and Feeds

- Choose **Feeds and Speeds** to enter the feed and speed parameters

Speed

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

- Enter the **Spindle Speed** value as **4500 rpm**

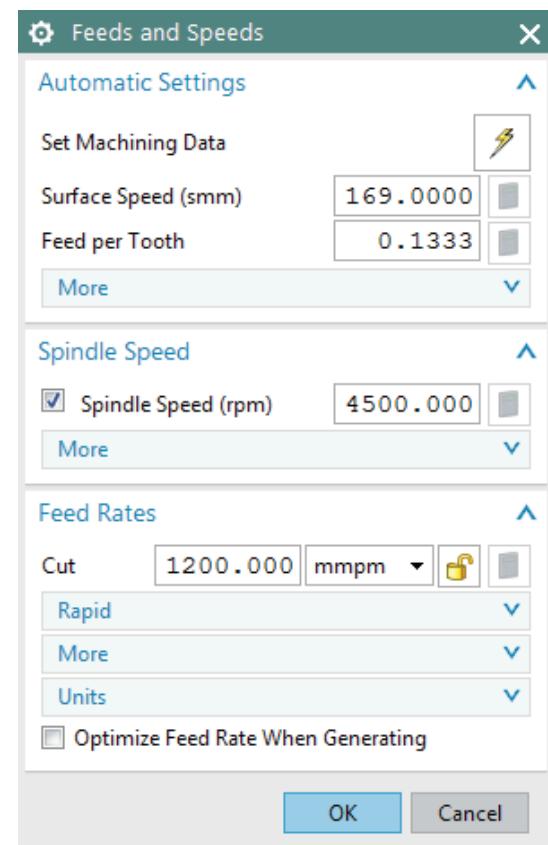
For the Surface Speed and the Feed per Tooth, you should enter the recommended values given by the manufacturers of the cutter (for this example, click on the calculator button near spindle speed). By entering these values, the software will automatically calculate the cutting feed rate and spindle speed. You can also enter your own values for feed rates and spindle speeds.

Feeds

There are many feeds involved in a single program. The most important is the *Cutting* feed. This is the feed at which, the tool will be in engagement with the raw work-piece and actually cutting the material off the work-piece. It is the relative linear velocity, at which the cutter moves with respect to the job.

The other feeds are optional. Some machine control systems use their default retracts and traverse feed. In those cases, even if you do not enter the values of other feeds, there would not be any problems. Some control systems may look for these feed rates from the program. It can be slightly less than the machine's maximum feed rate.

- Enter the **Cut** value as **1200 mppm**
- Click **OK**



9.3 PROGRAM GENERATION AND VERIFICATION

9.3.1 Generating Program

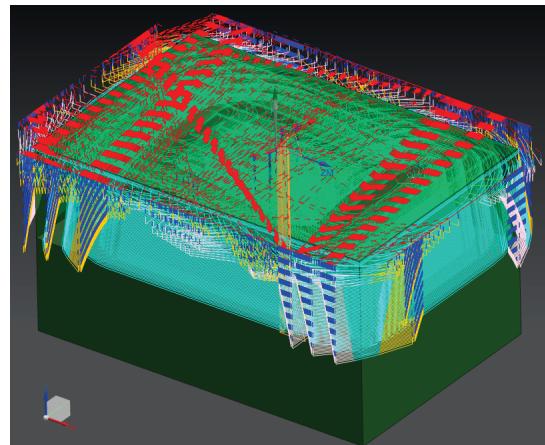
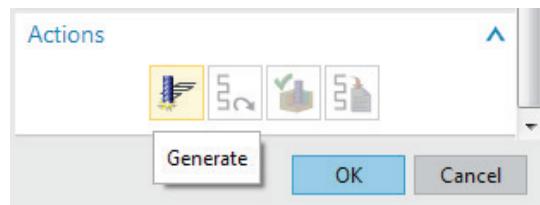
Now we are done entering all the parameters required for the roughing program. It is time to generate the program.

- Click on the **Generate** icon at the bottom of the window

You can now observe the software slicing the model into depths of cuts and creating tool-path at every level. You can find on the model cyan, blue, red and yellow lines as shown in the figure.

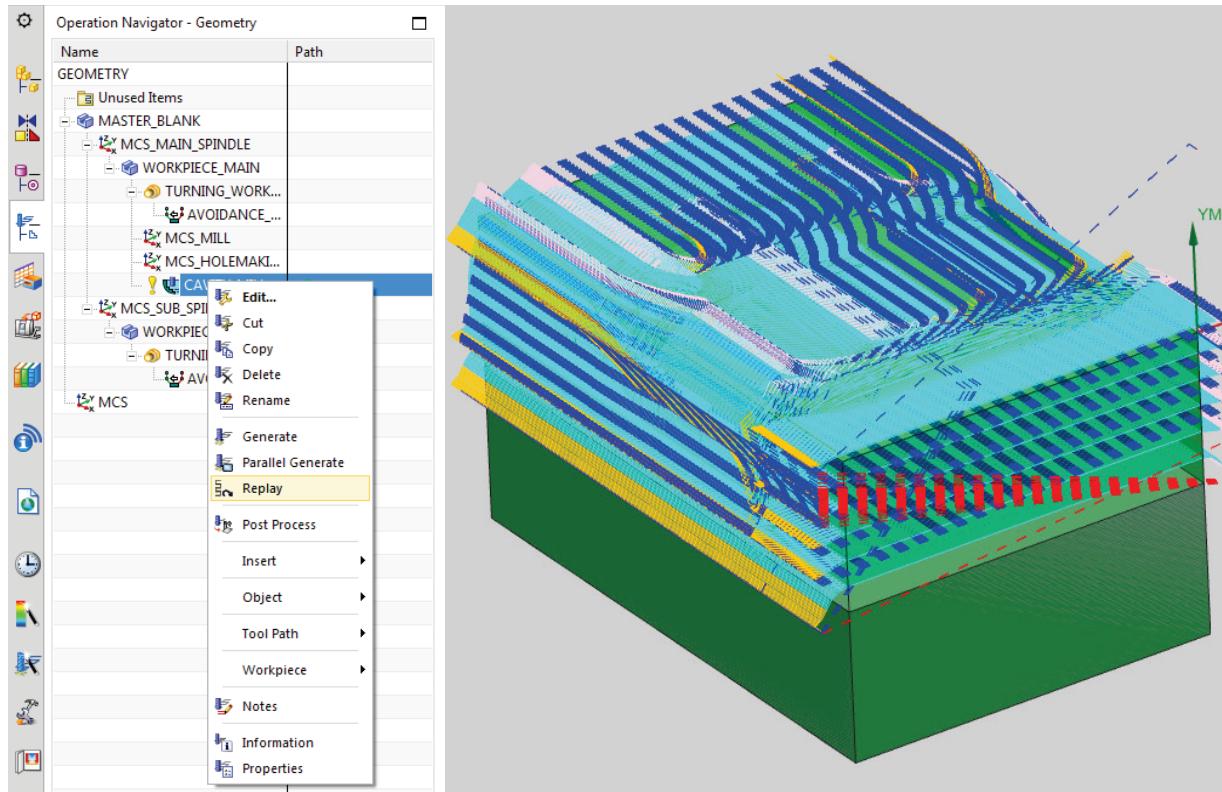
During the generation, you may be prompted with a *Display Parameters* window.

- 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



9.3.2 Tool Path Display

Whenever you want to view the entire tool-path of the program, right-click on the program in *Operation Navigator* and click *Replay*. It will give the display as shown in the Figure.



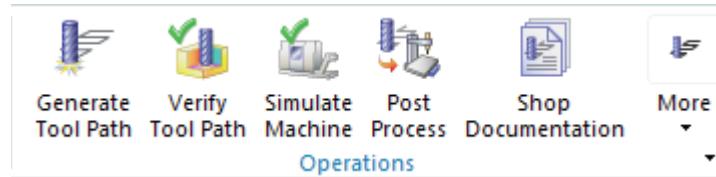
You can now observe that next to the program in the *Operation Navigator* is a yellow exclamation point instead of a red mark. This means that program has been generated successfully but has not been post-processed. If any change is made in the model, the program will again have a red mark next to it. This implies that the program has to be generated again. However, there is no need to change any parameters in the program.

9.3.3 Tool Path Simulation

It is very important to check the programs you have created. This prevents any improper and dangerous motions from being made in the cutting path. It is possible that wrong parameters and settings will be given that cause costly damages to the work piece. To avoid such mistakes, NX 10 and other CAM software provide *Tool-path verification* and a *Gouge* check.

The *Tool-Path verification* can be used to view the cutter motion in the entire program. You can observe how the tool is engaged and how it retracts after cutting. It also shows the actual material being removed through graphical simulation. You can also view the specific zone of interest by moving the line of the program.

- Right-click on the program in the **Operation Navigator** and choose **Tool Path → Verify** or click on the **Verify Tool Path** button in the toolbar



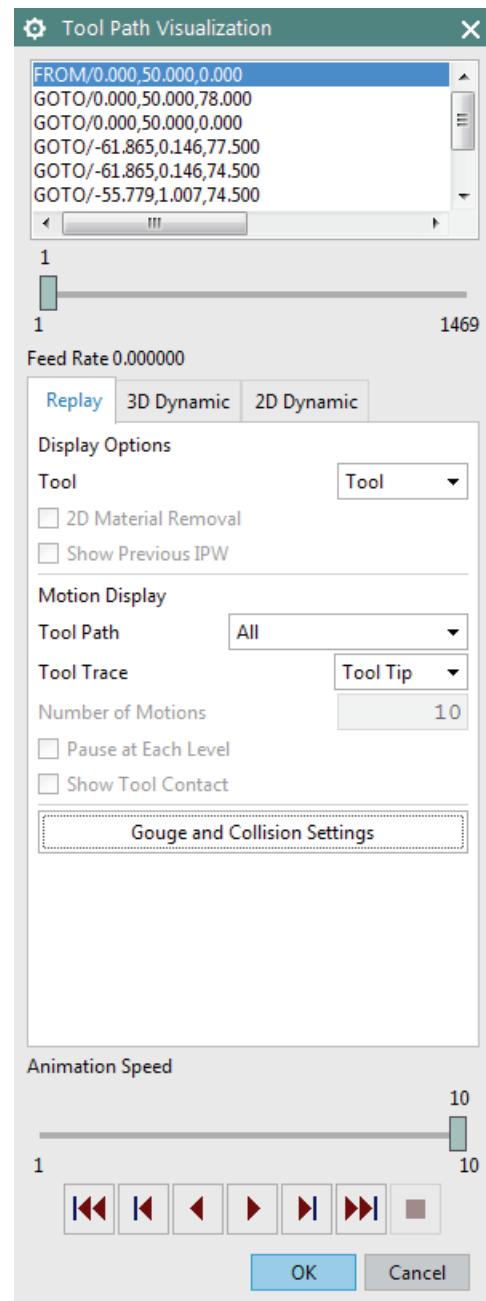
This will allow you to set the parameters for visualization of the *Tool-Path*.

- On the **Tool Path Visualization** window, click on

the **Play** icon to view the motion

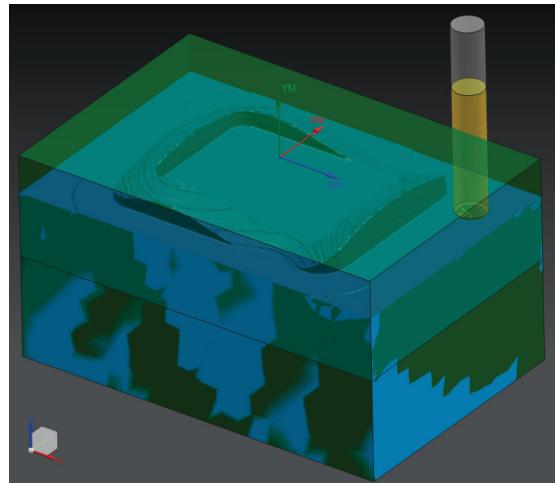
You can also view the visualization in different modes by changing the options in the drop-down menu next to *Display*.

- Click on the **3D Dynamic** tab on the same window
- Click on the **Display Options** button on the same window
- Change the **Number of Motions** to **50**
- Change the **Animation Accuracy** to **Fine**
- Change the **IPW Color** to **Green**
- Click **OK**



- Click on the Play button  again

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



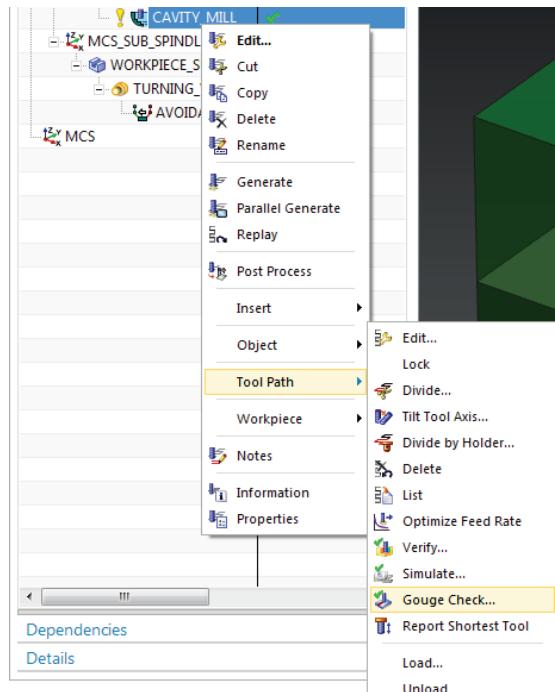
9.3.4 Gouge Check

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**
- Click **OK**

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

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



9.4 OPERATION METHODS

9.4.1 Roughing

In case of milling operation, the first operation should be rough milling before finishing the job. The main purpose of roughing is to remove bulk material at a faster rate, without affecting the accuracy and finish of the job. Stock allowances are given to provide enough material for the finishing operation to get an accurate and good finish job. What we did in the 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.

9.4.2 Semi-Finishing

Semi-Finishing programs are intended to remove the unevenness due to the roughing operation and keep even part stock allowance for the *Finishing* operations. Once we are done with the first roughing program, semi-finishing is always easier and simpler to perform.

Now we will copy and paste the first program in the *Operation Navigator*. In the new program, you only have to change a few parameters and cutting tool dimensions and just regenerate the program.

- Right-click **CAVITY_MILL** program in the **Operation Navigator** and click **Copy**
- Right-click **CAVITY_MILL** again and choose **Paste**
- Right-click the second **CAVITY_MILL_COPY** you just made and click **Rename**
- Rename the second program as **CAVITY_MILL_1**

Operation Navigator - Geometry		
Name	Path	Toc
GEOMETRY		
Unused Items		
MASTER_BLANK		
MCS_MAIN_SPIN...		
WORKPIECE_M...		
TURNING...		
MCS_MILL		
MCS_HOLE...		
CAVITY...	X	BUE
CAVITY...	✓	BUE

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

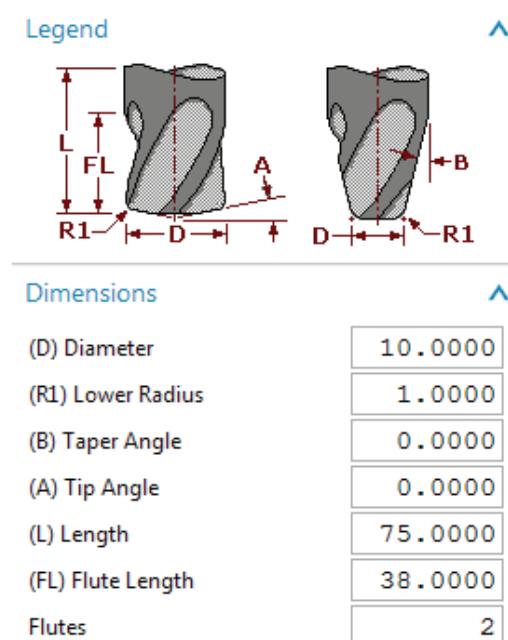
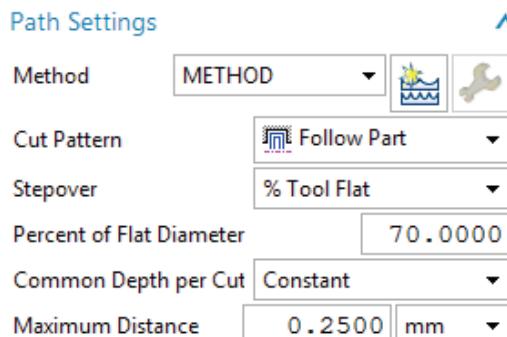
Let us now set the parameters that need to be changed for the second program. Before we even start, we should analyze the part geometry to figure out the minimum corner radius for the cutter diameter. In our model, it is 5 mm and at the floor edges, it is 1 mm. Therefore, the cutter diameter

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.

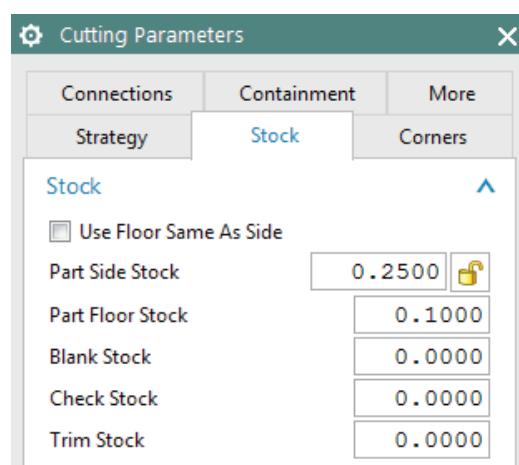
- Double-click **CAVITY_MILL_1** on **Operation Navigator** to open the parameters window

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 *BUEM10X1* as the current tool.

- Create a new **Mill** and name it **BUEM10X1**
- It should have a **Diameter** of **10**, a **Lower Radius** of **1** and a **Flute Length** of **38**
- Click **OK**
- Click the **Common Depth per Cut** as **0.25** in the **Path Settings**



- Then click on **Cutting Parameters** button
- Click on the **Stock** tab
- Uncheck the box next to **Use Floor Same As Side**
- Enter **0.25** for **Part Side Stock**
- Enter **0.1** for **Part Floor Stock**
- Click on the **Containment** tab button



- In the drop-down menu next to **In Process Workpiece**, choose **Use 3D**

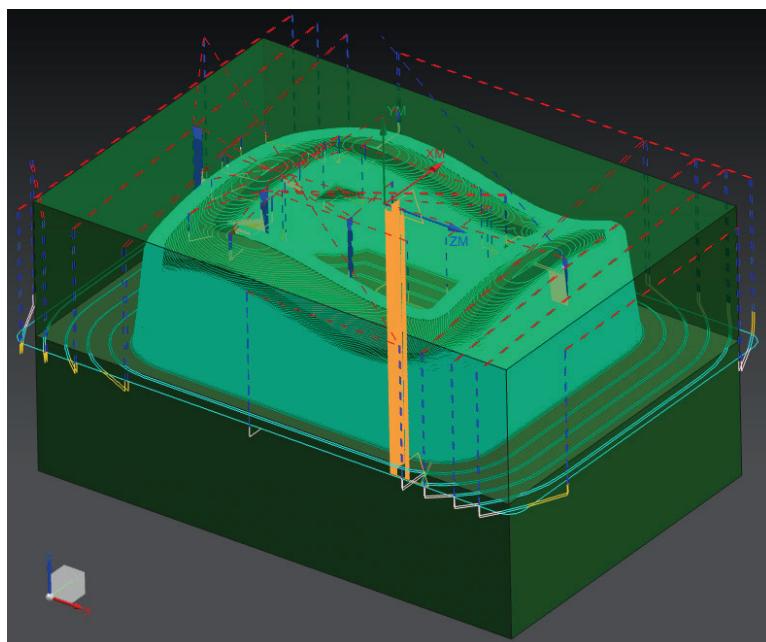
In Process Workpiece is a very useful option in NX. The software considers the previous program and generates the current program such that there is no unnecessary cutting motion in the *No-material* zone. This strategy reduces the cutting time and air cutting motion drastically. The algorithm will force the cutter to only remove that material, which 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 **Spindle Speed** as **5000** and click on the **Calculator**
- Then click **OK**

The parameters and settings are finished for the semi-finishing program.

- Regenerate the program by clicking on the **Generate** icon 
- After the software finishes generating click **OK**

Then replay the *Tool Path Visualization*. The overall *Tool Path* generated in the second program will look like the following figure. You can replay it or check for the gouging in a similar way.



9.4.3 Finishing Profile

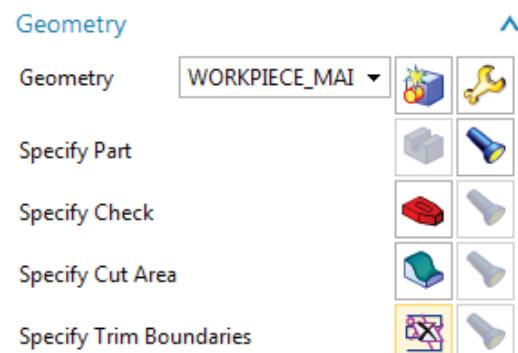
So far, we are done with the roughing and semi-finishing programs for the part. There is a small amount of material left in the *Workpiece* to be removed in the finishing programs to obtain the accurate part geometry as intended in the design. The finishing programs should be generated such that every 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.

9.4.3.1 Outer Profile

This program is intended to finish the outer inclined walls onto the bottom of the floor. Because the program should not touch the contour surface on the top, we 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**
- Click on the **Specify Trim Boundaries** tab

The *Trim Boundaries* window will pop up. Make sure to carry out the following procedure in the right sequence. Keep the default setting of *Trim Side* to *Inside*. This tells the software that the cutter should not cut material anywhere inside the boundary. Trim allows you to specify boundaries that will further constrain the cut regions at each cut level.



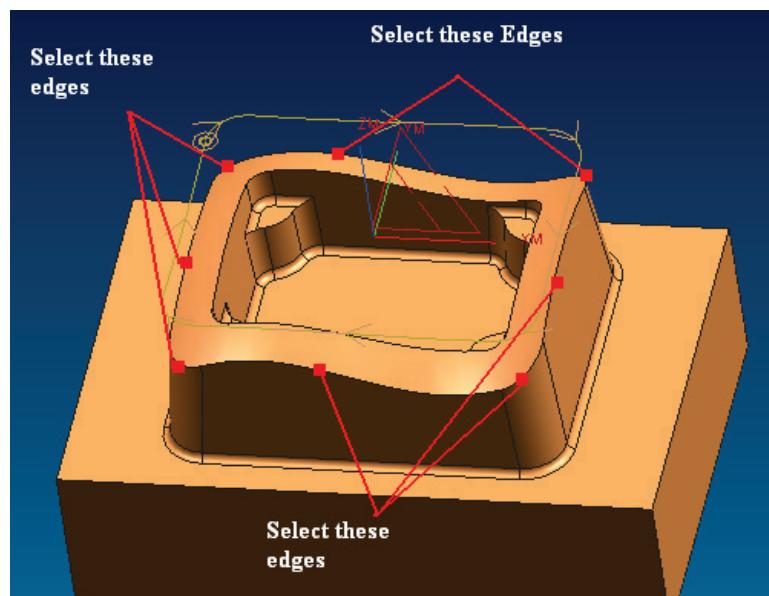
- Change the **Selection Method** to **Curves**
- Change the **Plane** from **Automatic** to **Specify** and click on the **Plane Dialog**

A new window will pop up. The window will ask for the mode of selection of the plane on which the curves should be projected. This should normally be over the topmost point of the part geometry. Precisely, it should be over the *MCS*.

- Choose the **XC-YC Plane** from the drop-down menu under **Type**
- Enter a value of **3** next to **Distance**
- Click **OK**

Now we will start selecting edges from the part. These selected edges will be projected on the $Z = 3$ plane as curves and used as the boundary.

- Select all the top outer edges on the wall along the contour surface as shown in the figure.
Make sure to select all **8 edges** and in a **continuous order**
- Choose **OK**

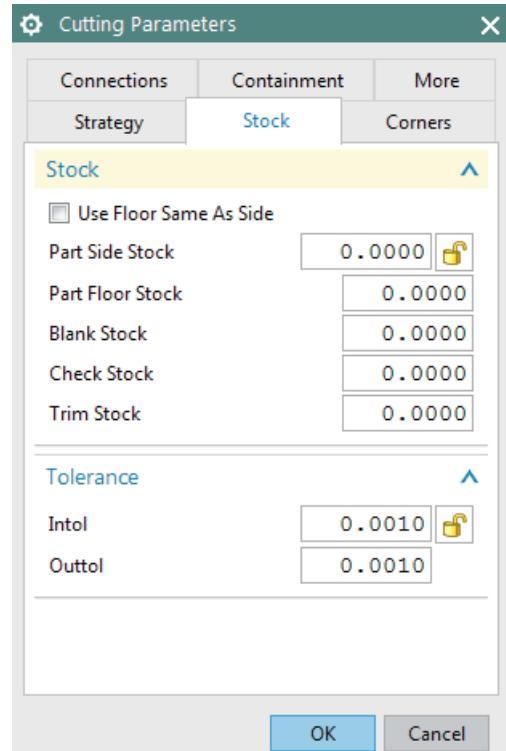


- Enter the **Common Depth per Cut** as **0.2**
- Click **Cutting Parameters**
- In the pop up dialog box, click on **Stock** tab
- Enter the **Part Side Stock** and **Part Floor Stock** values to be **0.00**

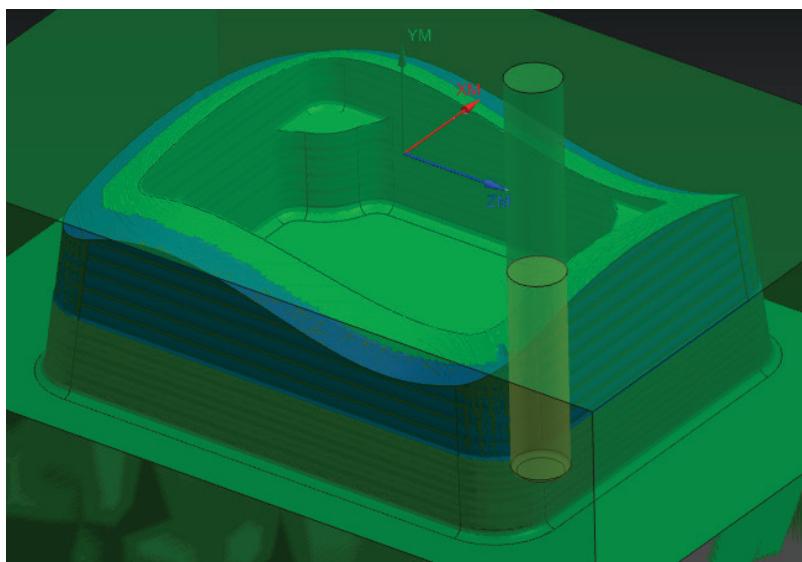
Intol allows you to specify the maximum distance that a cutter can deviate from the intended path into the workpiece.

Outtol allows you to specify the maximum distance that a cutter can deviate from the intended path away from the workpiece.

- Enter the **Intol** and **Outtol** values to be **0.001** as shown in the figure
- Click 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.

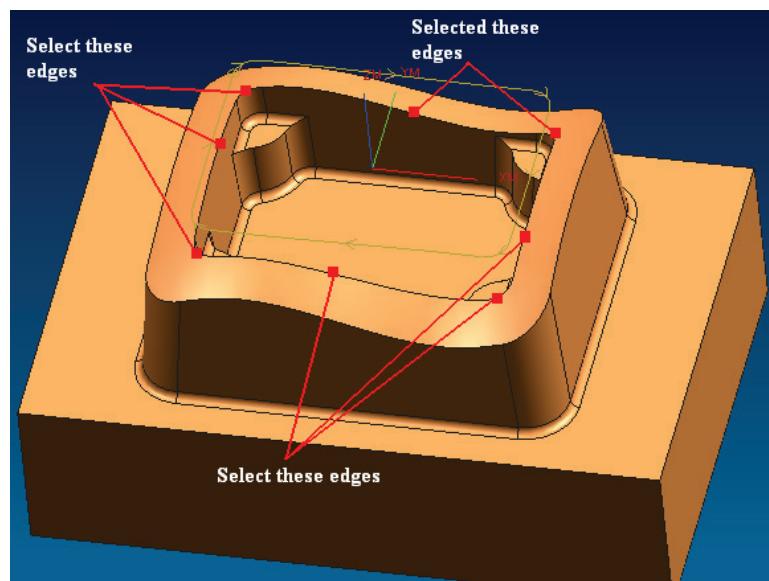


9.4.3.2 Inner profile

- Repeat the same procedure as before to copy and paste **CAVITY_MILL_2** on **Operation Navigator** and rename it as **CAVITY_MILL_3**
- Double-click **CAVITY_MILL_3** to edit the parameters or right click on it and choose **Edit**
- Select the **Specify Trim Boundaries** tab and choose **Trim Side** to be **Outside** in the pop up dialog box

This will prevent the cutter from passing outside the boundary.

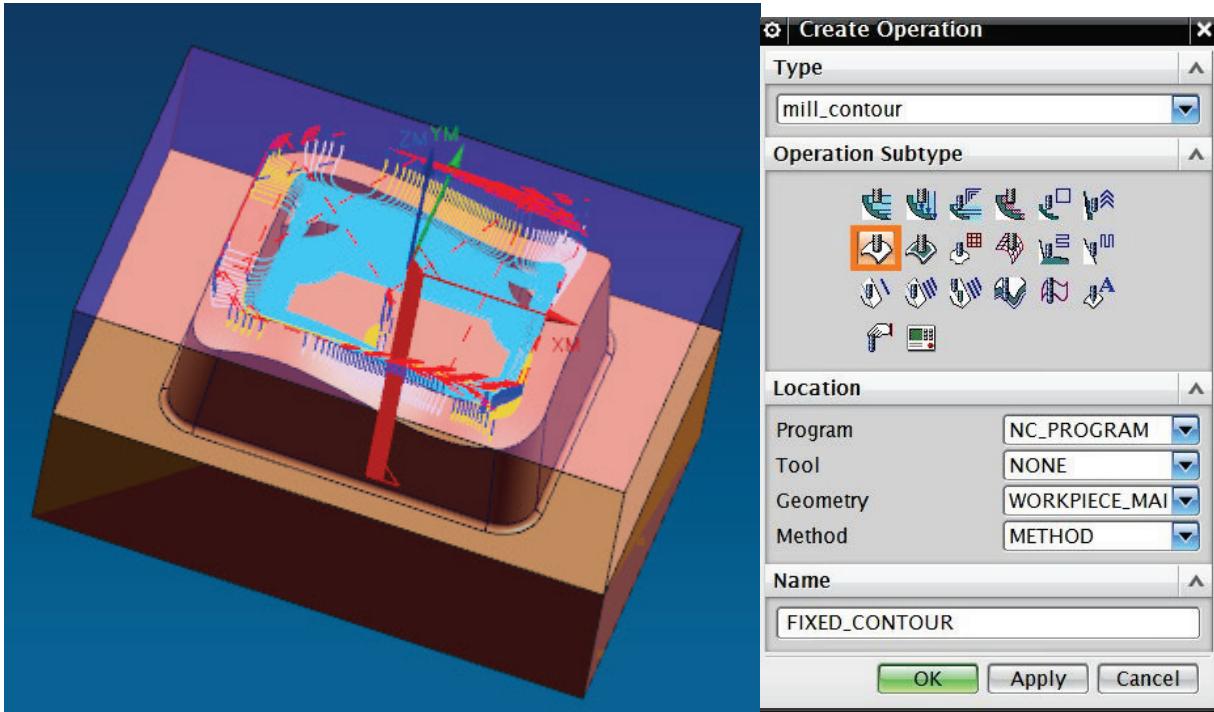
- Change **Selection Method** to **Curves**
- Change the plane manually to be the **XC-YC** plane and enter the offset distance as **3**
- Click **OK**
- Select all the top inner edges along the contour surface as shown in the figure. Again, make sure all 8 edges are selected in a continuous order.
- 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

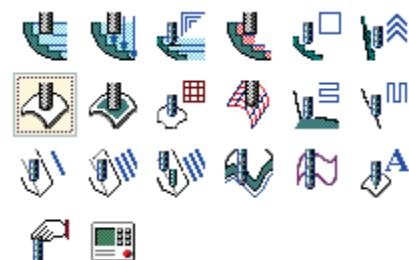
The finishing program for the outer profile is now ready. By replaying the tool path, you can observe that the cutter never crosses the boundary that has been given for trim and check.



9.4.4 Finishing Contour Surface

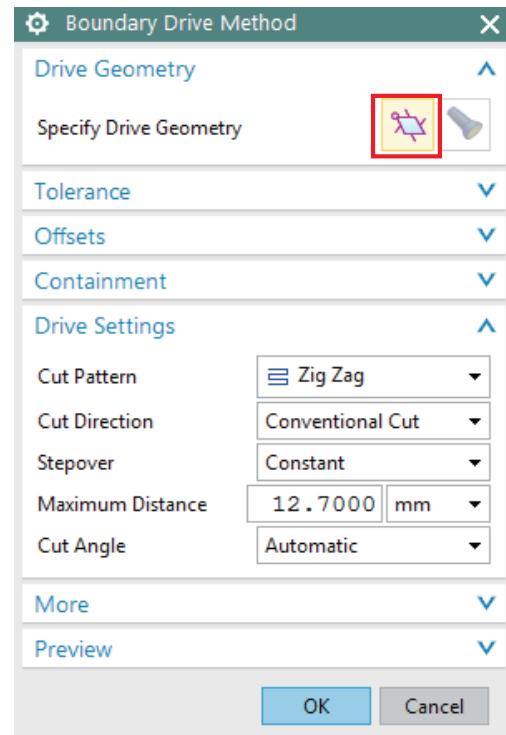
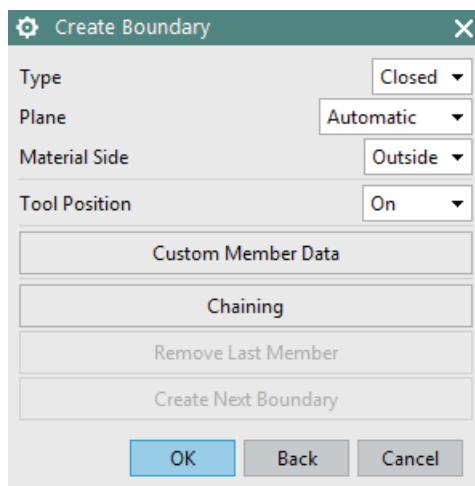
Now we have to use a different type of strategy to finish the top freeform surface.

- Click on the **Create Operation** icon  in the Toolbar
- Click on the **Fixed Contour** icon as shown in the figure
- Choose **1234** for Program
- Choose **WORKPIECE_MAIN** for Geometry
- Keep the default name of program
- Click **OK**
- On the **Parameters** window, under **Drive Method**, select **Boundary** even if it is already shown



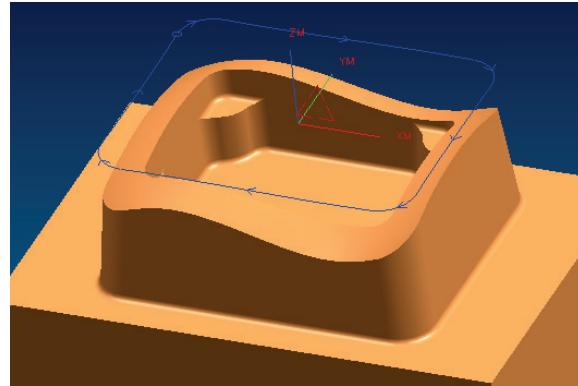
If the *Boundary Drive Method* window still does not show up, select another *Drive Method* other than *Boundary*, then cancel it and choose *Boundary* again!

- When **Boundary Drive Method** pups up, click on the **Spanner** icon as shown in the figure to open the **Boundary Geometry** menu
- Change the **Mode** to **Curves/Edges**
- Select the **Material Side** to be **Outside**
- Select the **Tool Position** to be **On**



The *Tool Position* determines how the tool will position itself when it approaches the *Boundary Member*. *Boundary Members* may be assigned one of three tool positions: *On*, *Tanto*, or *Contact*.

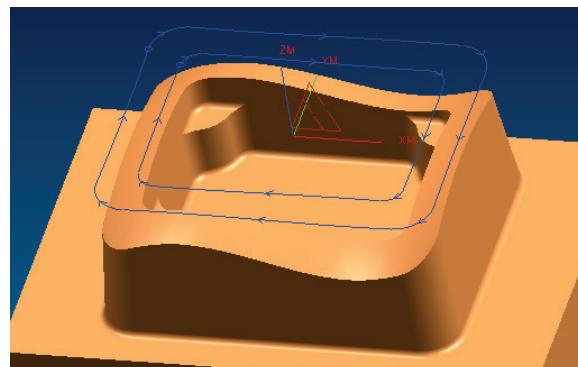
- In **On** position, the center point of the tool aligns with the boundary along the tool axis or projection vector.
- In **Tanto** position, the side of the tool aligns with the boundary.
- In **Contact** position, the tool contacts the boundary.
- For the **Plane**, choose **User-Defined**
- Again, set the plane to be **XC-YC** with a **Distance** of **3**
- Click **OK**
- Select the outer loop of the top contour surface as shown in the figure. Remember to select the edges in a continuous order



- Click **OK**

We have trimmed the geometry outside the loop. Now we have to trim the geometry inside the inner loop so that the only geometry left will be the area between the two loops.

- Choose the **Mode** to be **Curves/Edges**
- Choose the **Material Side** to be **Inside** and **Tool Position** to be **On**
- Choose the plane to be user-defined at **XC-YC** with a **Distance** of **3**
- Select the inner edges of the contour surface as shown
- Click **OK** to return to the **Boundary Drive Method** window



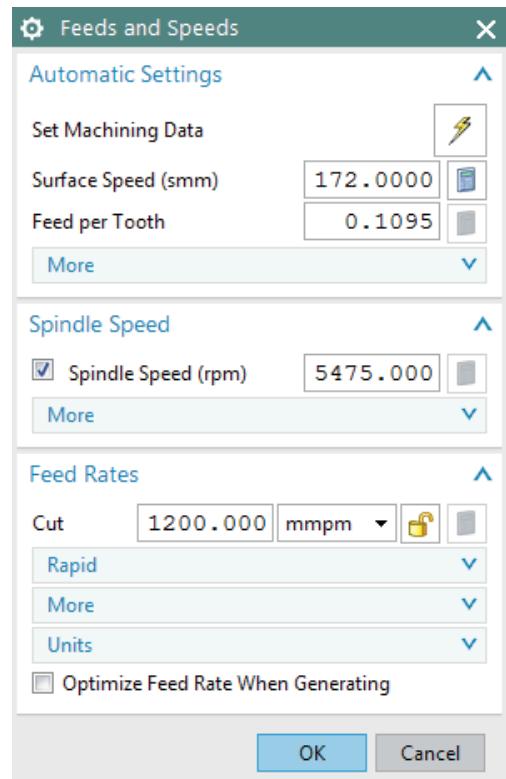
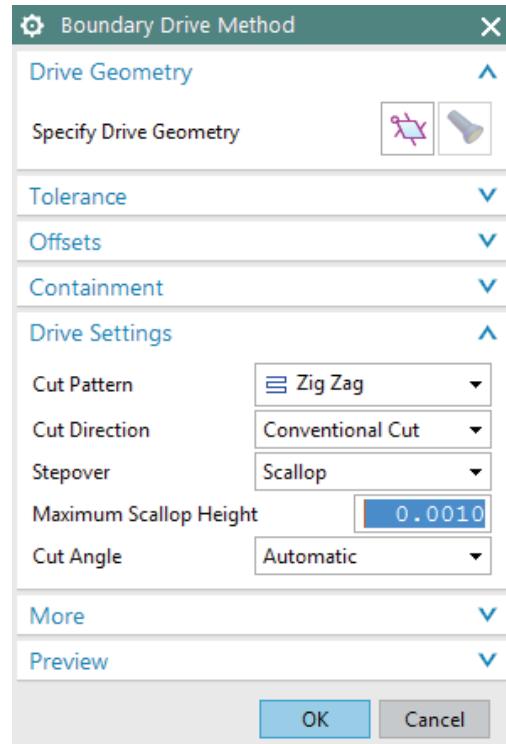
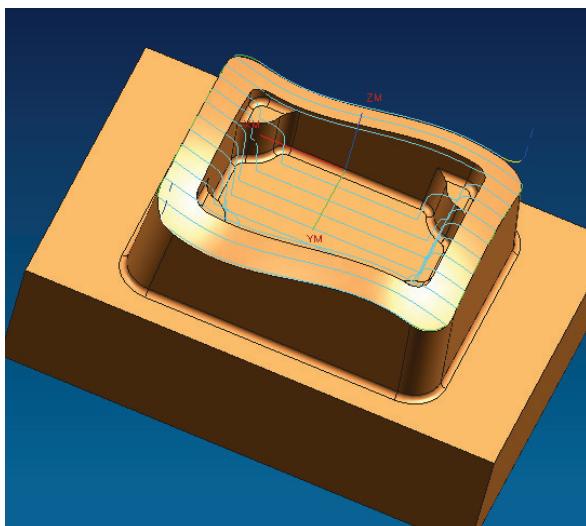
- Change the **Stepover** method to **Scallop** and enter the height to be **0.001** and click **OK**

- Click on **Cutting Parameters**
- Change the **Tolerance** values in the **Stock** tab so that the **Part Intol** and **Part Outtol** is **0.001**
- Click on the **More** tab button and enter the value of **Max Step** as **1.0**
- Click **OK**
- Click on the **Feeds and Speeds** icon on the parameters window
- Enter the parameters as shown in the figure on right (do not let the software calculate it)
- 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*.



9.4.5 Flooring

Flooring is the finishing operation performed on the horizontal flat surfaces (*Floors*) of the part. In most of the milling processes, flooring will be the final operation of the process. All the horizontal surfaces have to be finished. This planar operation runs the cutter in a single pass on every face.

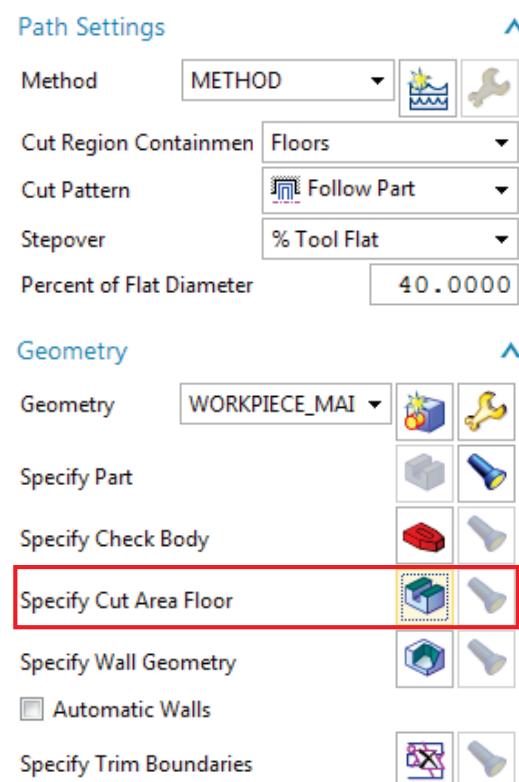
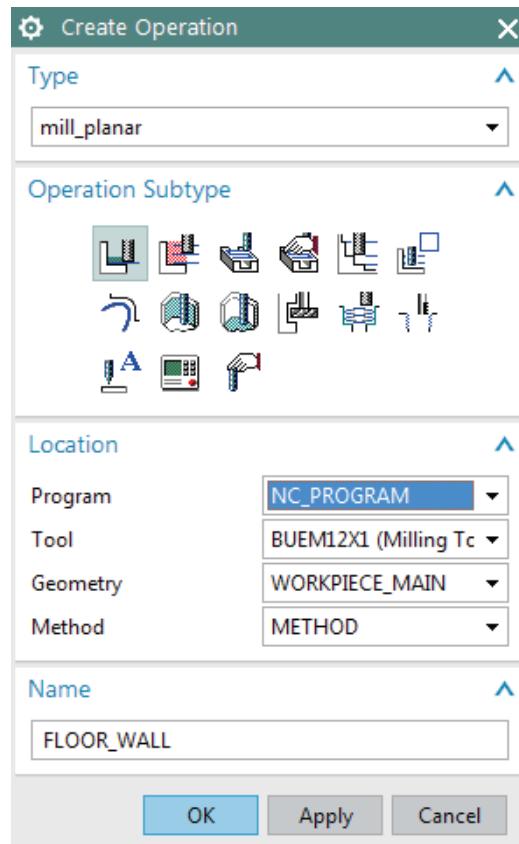
- Click on the **Create Operation** icon 
- Change the **Type** to be **mill_planar** at the top of the window
- Change all the options as shown in the figure
- Click **OK**
- In the parameters window, change the **Cut Pattern** to be **Follow Part**
- Change the percent of the tool diameter for **Stepover** to be **40**

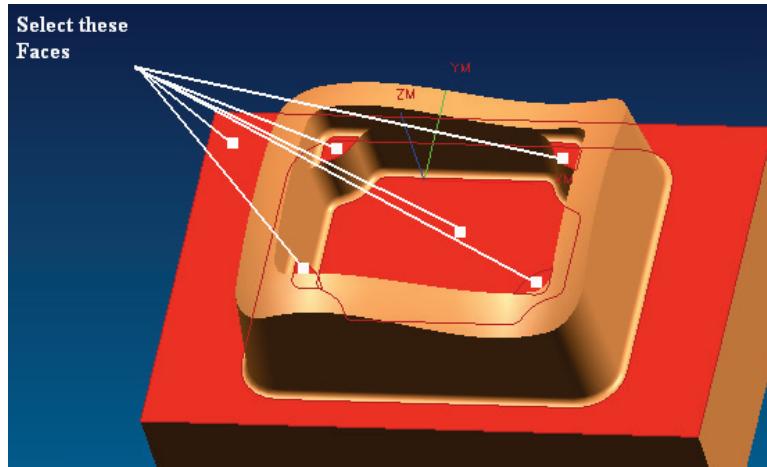
In flooring operations, it is always better to keep the *Stepover* value to be less than half of the diameter of the cutter in order to achieve more flatness on the planar surfaces.

Unlike previous programs, we have to select a cut area.

- Click on the **Specify Cut Area Floor** as shown
- 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.





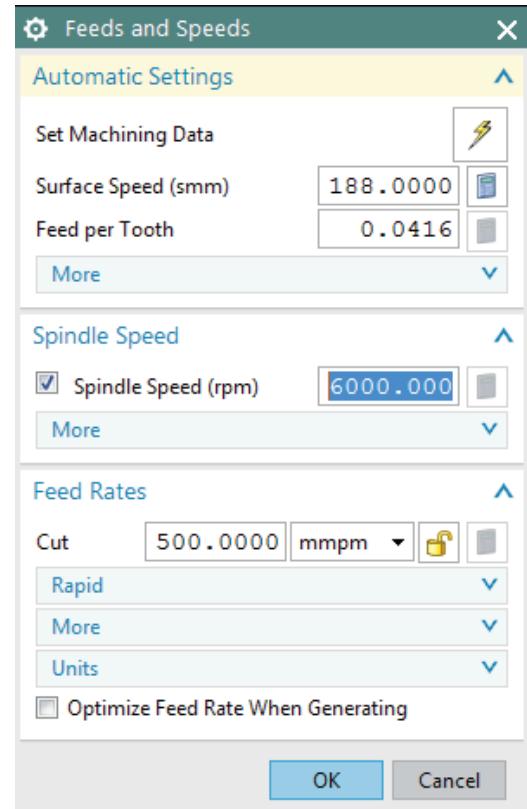
- Click **OK**
- Click on **Cutting Parameters** in the main parameter window
- Choose the **Stock** tab button and enter the **Intol** and **Outtol** values as **0.001**
- Click **OK**
- Click on **Feeds And Speeds**

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.

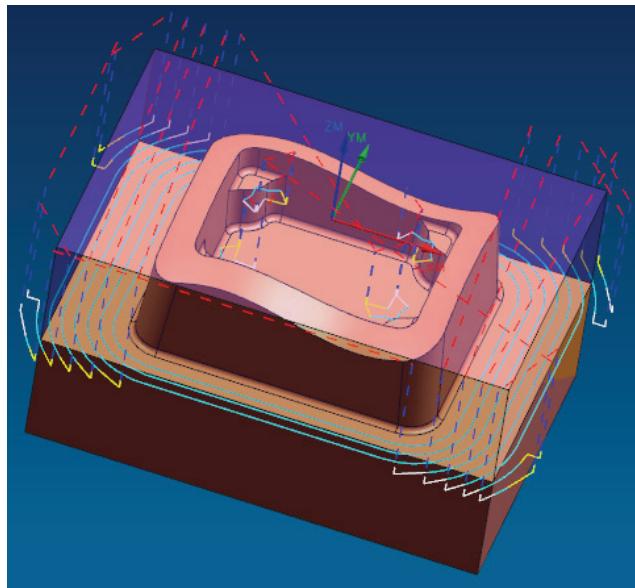
- Enter the values exactly as shown in the figure
- Choose **OK**

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 *Tool Path* display for the flooring.



9.5 POST PROCESSING

The primary use of the *Manufacturing Application* is to generate tool paths for manufacturing parts. Generally, we cannot just send an unmodified tool path file to a machine and start cutting because there are many different types of machines. Each type of machine has unique hardware capabilities, requirements and control systems. For instance, the machine may have a vertical or a horizontal spindle; it can cut while moving several axes simultaneously, etc. The controller accepts a tool path file and directs tool motion and other machine activity (such as turning the coolant or air on and off).

Naturally, as each type of machine has unique hardware characteristics; controllers also differ in software characteristics. For 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 NX 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)*.

- Post process the *CLSF* into machine CNC code (*Post Processed* file). This program reads the tool path data and reformats it for use with a particular machine and its accompanying controller.

9.5.1 Creating CLSF

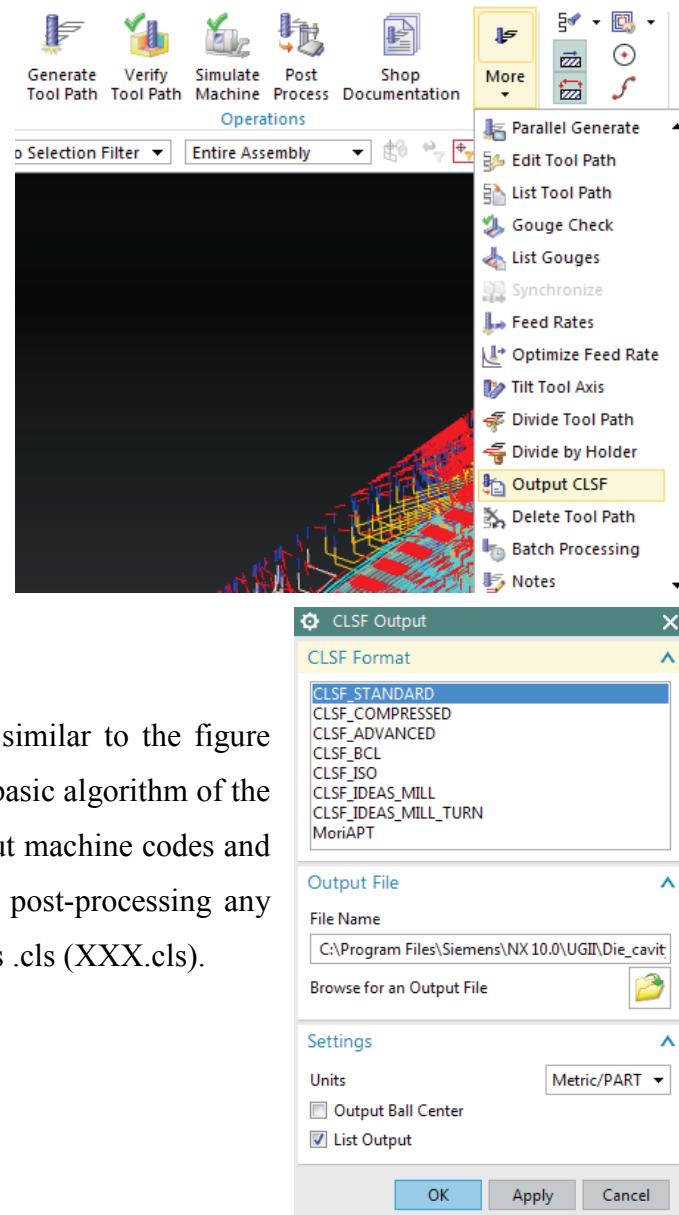
After an operation is generated and saved, the resulting tool path is stored as part of the operation within the part file. *CLSF* (*Cutter Location Source File*) provides methods to copy these internal paths from the operations in the part file to tool paths within the *CLSF*, which is a text file. The *GOTO* values are a "snapshot" of the current tool path. The values exported are referenced from the MCS stored in the operation. The *CLS* file is the required input for some subsequent programs, such as postprocessors.

- Click on one of the programs that you want to post process in the **Operation Navigator**
- Click on **Output CLSF** in the **Operations** toolbar

A window will pop up to select the *CLSF Format*.

- Choose **CLSF_STANDARD** and enter a location for the file
- Choose **OK**

The *CLSF* file will be created. It will be similar to the figure below. The contents of the file contain the basic 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).

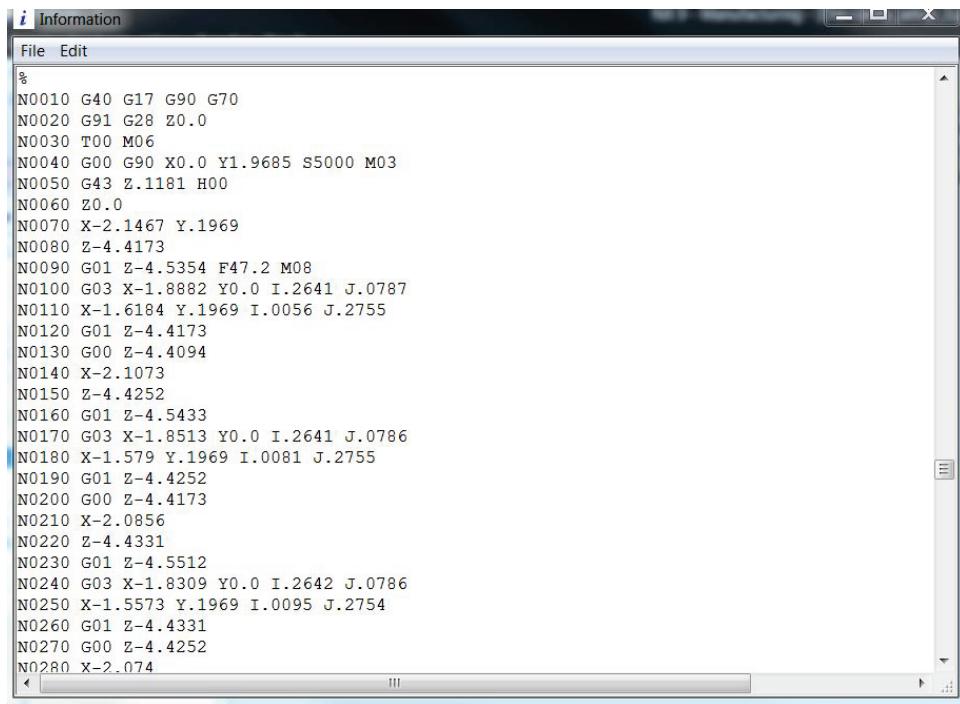


Any program that has been output to *CLSF* or *Post Processed* will have a green checkmark next to it in the *Operation Navigator*.

9.5.2 Post Processing

- Click on a program in the **Operation Navigator** that you want to post process
 - Click **Menu → Tools → Operation Navigator → Output → Postprocess** or from the **Home** tab as shown 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 screenshot shows a software window titled "Information". The menu bar includes "File" and "Edit". The main area displays a list of G-code commands:

```
%  
N0010 G40 G17 G90 G70  
N0020 G91 G28 Z0.0  
N0030 T00 M06  
N0040 G00 G90 X0.0 Y1.9685 S5000 M03  
N0050 G43 Z.1181 H00  
N0060 Z0.0  
N0070 X-2.1467 Y.1969  
N0080 Z-4.4173  
N0090 G01 Z-4.5354 F47.2 M08  
N0100 G03 X-1.8882 Y0.0 I.2641 J.0787  
N0110 X-1.6184 Y.1969 I.0056 J.2755  
N0120 G01 Z-4.4173  
N0130 G00 Z-4.4094  
N0140 X-2.1073  
N0150 Z-4.4252  
N0160 G01 Z-4.5433  
N0170 G03 X-1.8513 Y0.0 I.2641 J.0786  
N0180 X-1.579 Y.1969 I.0081 J.2755  
N0190 G01 Z-4.4252  
N0200 G00 Z-4.4173  
N0210 X-2.0856  
N0220 Z-4.4331  
N0230 G01 Z-4.5512  
N0240 G03 X-1.8309 Y0.0 I.2642 J.0786  
N0250 X-1.5573 Y.1969 I.0095 J.2754  
N0260 G01 Z-4.4331  
N0270 G00 Z-4.4252  
N0280 X-2.074
```

The final output (XXX.ptp) file can be transferred to the machine 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*.