CHAPTER 4 – THREE DIMENSIONAL MODELING

This chapter discusses the basics of three dimensional (3D) modeling in NX 12. We will discuss what a feature is, what the different types of features are, what primitives are, and how to model features in NX 12 using primitives. This will give a head start to the modeling portion of NX 12 and develop an understanding of the use of *Form Features* for 3D modeling. Once these features are introduced, we will focus on *Feature Operations* which are functions that can be applied to the faces and edges of a solid body or features you have created. These include taper, edge blend, face blend, chamfer, trim, etc. After explaining the feature operations, the chapter will walk you through some examples.

In NX 12, *Features* are a class of objects that have a defined parent. Features are associatively defined by one or more parents and the order of their creation and modification retain within the model, thus capturing it through the *History*. Parents can be geometrical objects or numerical variables. Features include primitives, surfaces and/or solids and certain wire frame objects (such as curves and associative trim and bridge curves). For example, some common features include blocks, cylinders, cones, spheres, extruded bodies, and revolved bodies.

Commonly *Features* can be classified as follows:

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

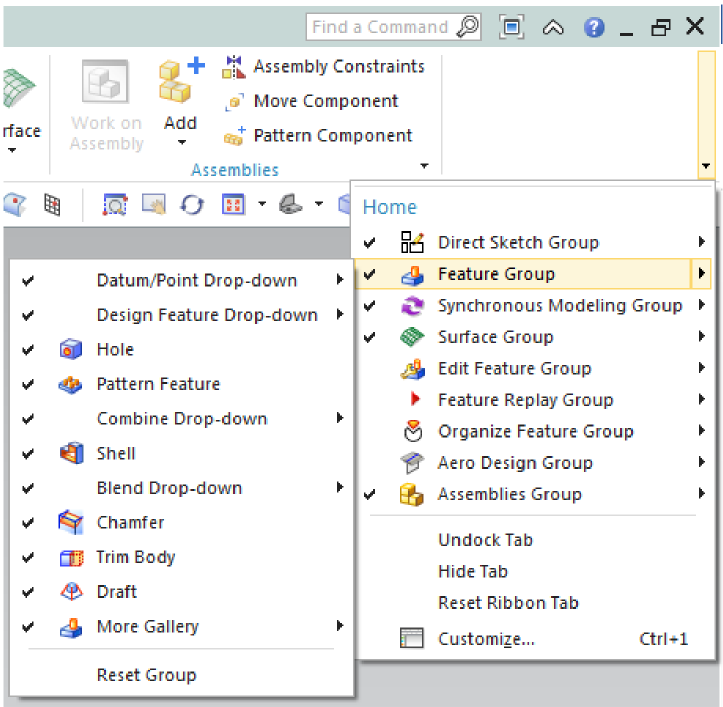
4.1 TYPES OF FEATURES

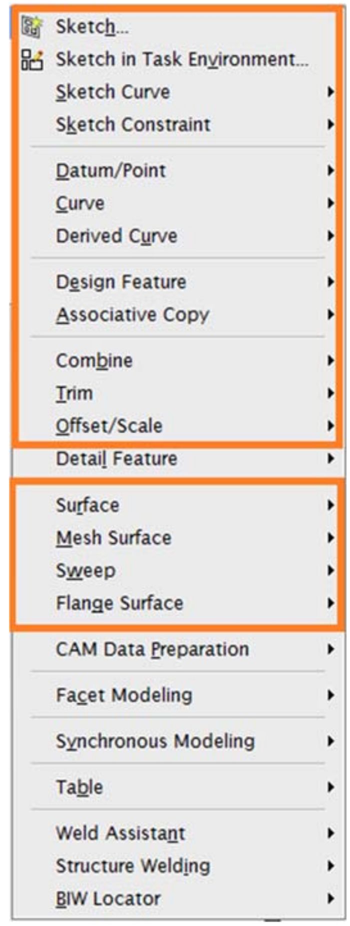
There are six types of *Form Features*: Primitives, Reference features, Swept features, Remove features, Extract features, and User-defined features. Similar to previous versions, NX 12 stores all the *Form Features* under the *Insert* menu option. The form features are also available in the *Form Features 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:

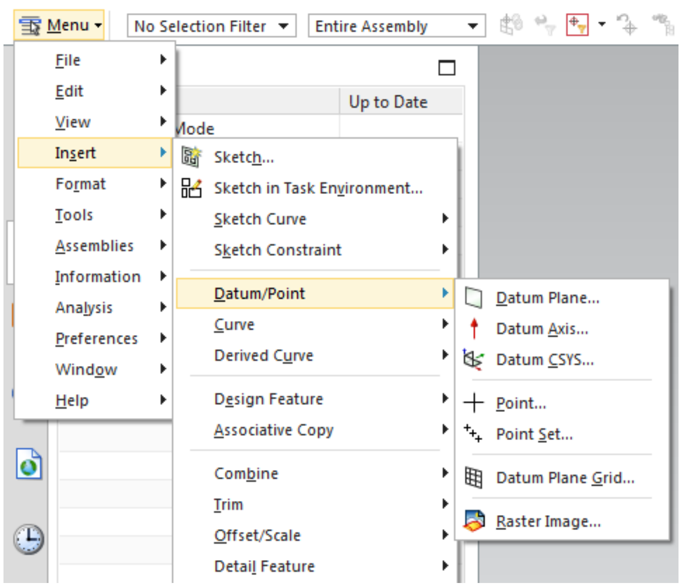
- Block
- Cylinder
- Cone
- Sphere

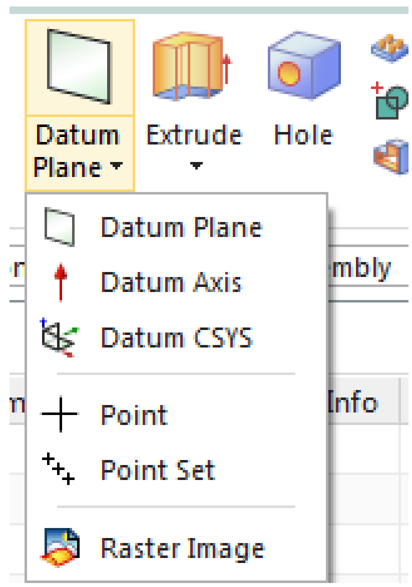
Primitives are the primary entities. Hence, we will begin with a short description of primitives and then proceed to modeling various objects.

4.1.2 Reference Features

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

➢ 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 sections. Swept Features include:

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

To select a swept feature you can do the following:

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

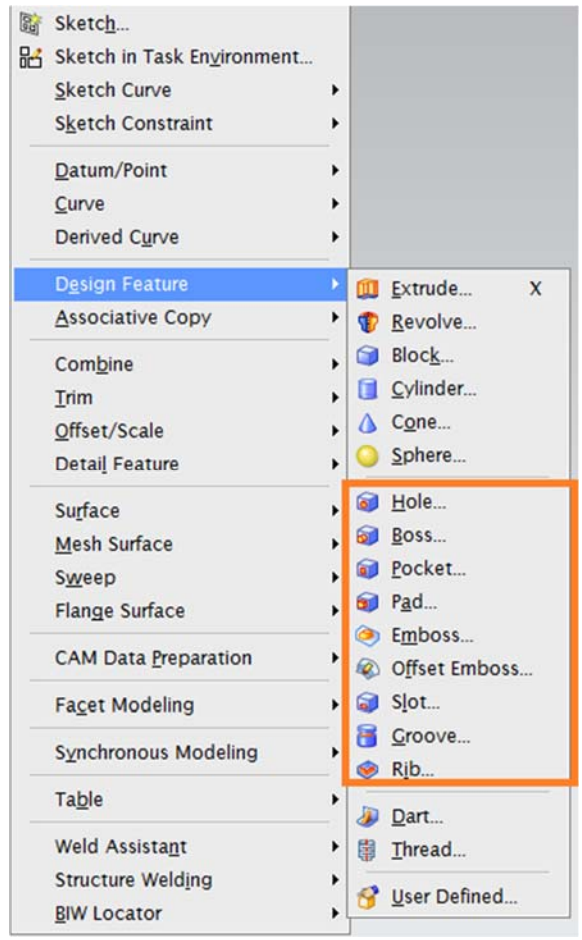
OR

➢ 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

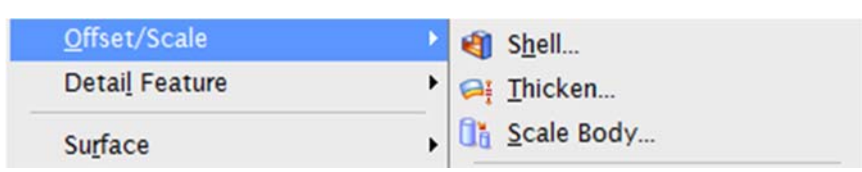
4.1.5 Extract Features

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:

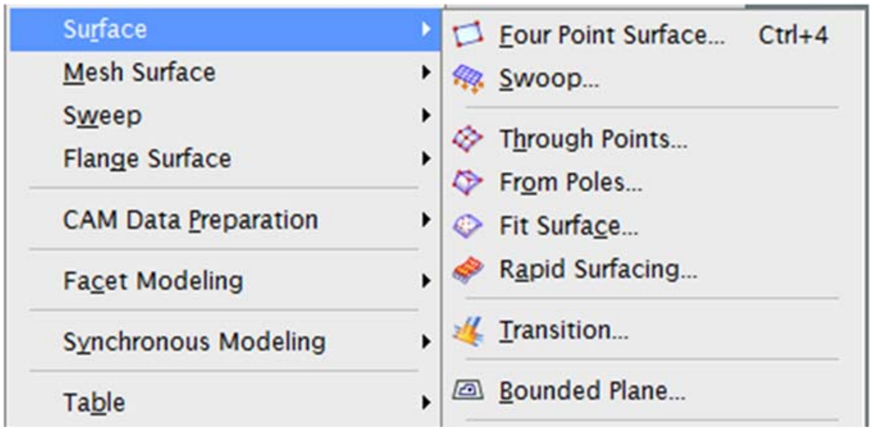
- Extract
- Solid to Shell
- Thicken Sheet
- Bounded plane
- Sheet from curves

➢ 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**

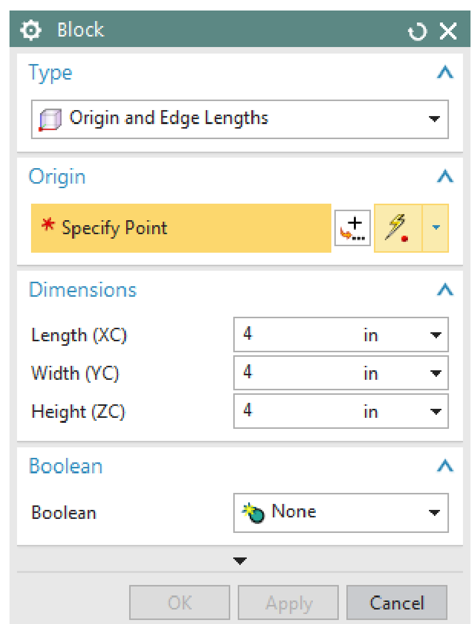


4.1.6 User-Defined features

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**

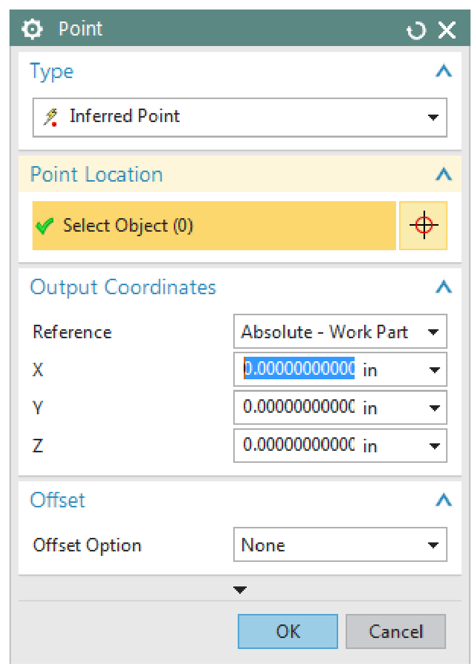
➢ Choose **Insert →Design Feature →Block** or click on the **Block** icon in the **Form Feature Toolbar**

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

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

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

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



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

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

➢ Click **OK**

The *Block* window will reappear again.

➢ Type the following dimensions in the window

Length (XC) = **65** inches

Width (YC) = **85** inches

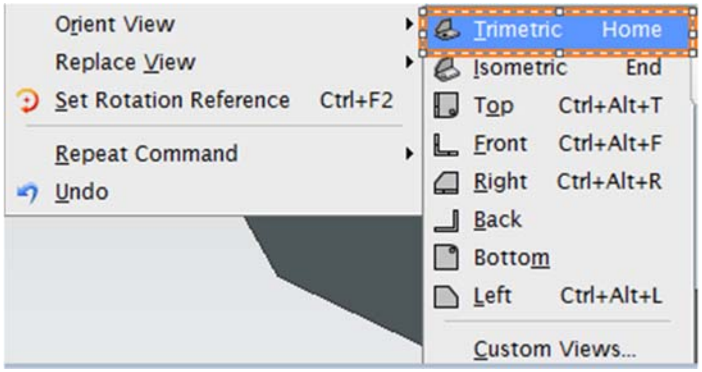
Height (ZC) = **20** inches

➢ Click **OK**

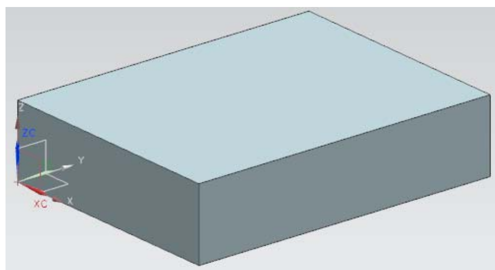
If you do not see anything on the screen,

➢ Right-click and select **FIT**. You can also press **<Ctrl> + F**

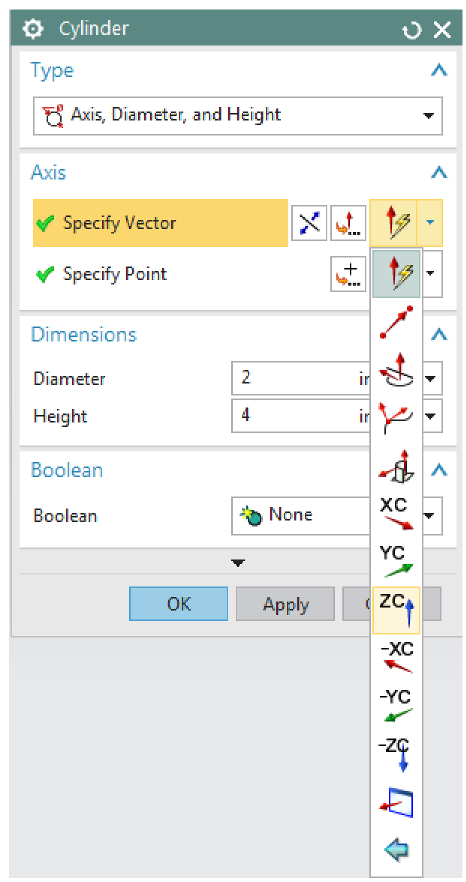
➢ Right-click on the screen and click on **Orient View →Trimetric**



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



4.2.2 Model a Shaft



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

- ➢ Create a new file and save it as **Impeller_shaft.prt**
- ➢ Choose **Insert** →**Design Feature** →**Cylinder** or click on **More** in **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

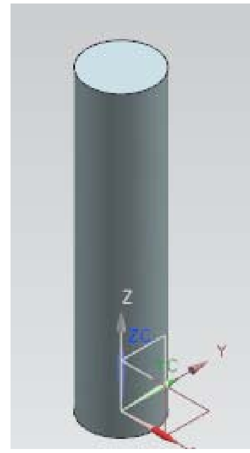
➢ In the next dialog box of the window, type in the following values

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.

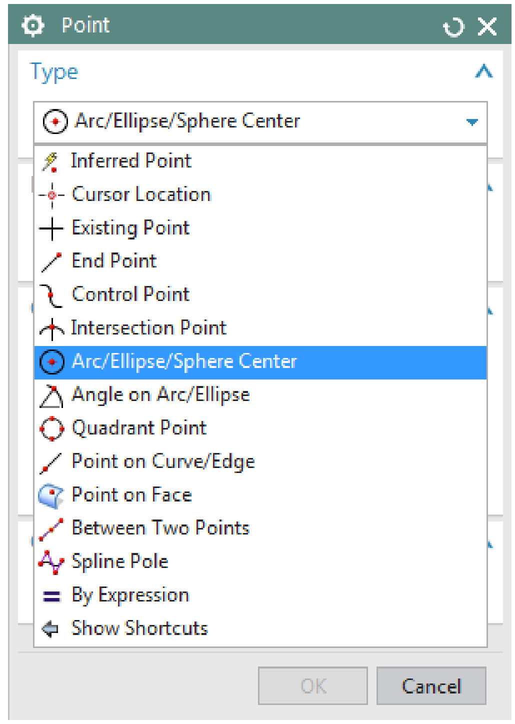
- Diameters and Height
- Diameters and Half Angle
- Base Diameter, Height, and Half Angle
- Top Diameter, Height, and Half Angle
- Two Coaxial Arcs

➢ 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.

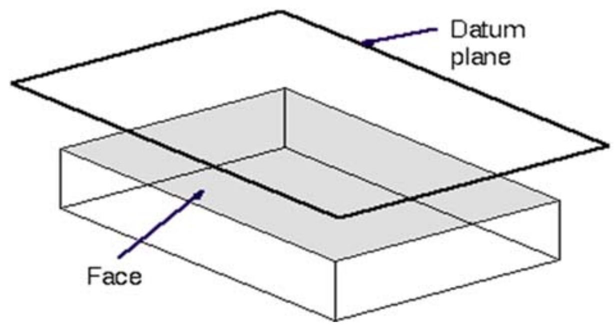
Now we will create one more cylinder on top of the cone.

➢ 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 below. Do not forget to save the model.

4.3 REFERENCE FEATURES

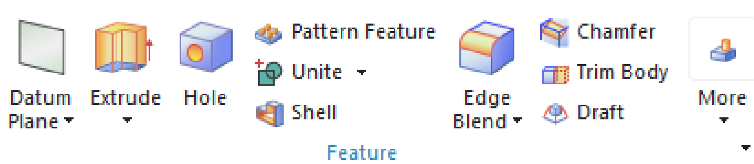
4.3.1 Datum Plane



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

- Open the model **Arborpress_plate.prt**
- Choose **Insert →Datum/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*.

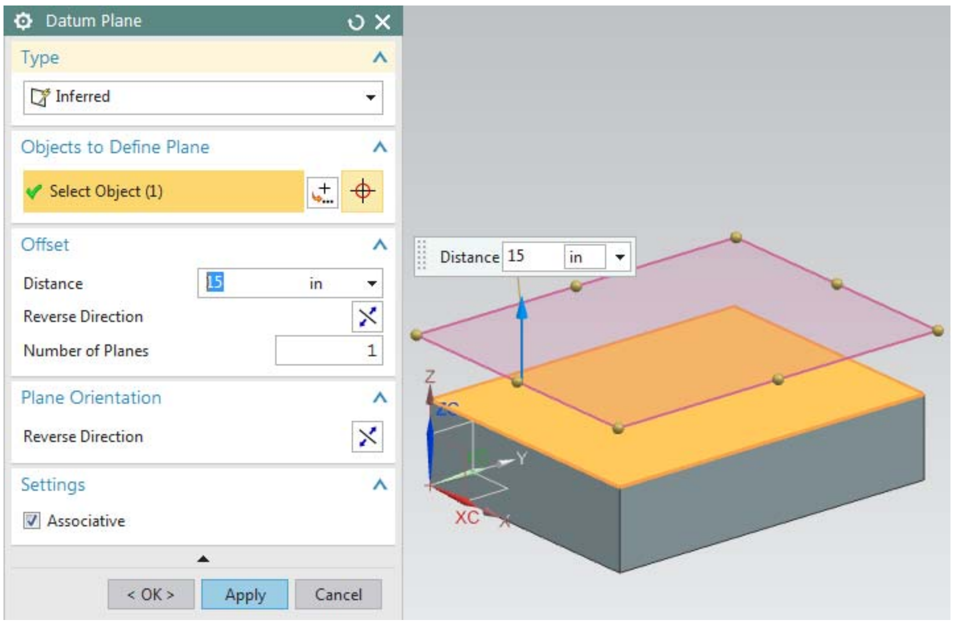


The *Datum Plane* window allows you to choose the method of selection. However, NX 12 is smart enough to judge the method depending on the entity you select if you keep in *Inferred* option, which is also the *Default* option.

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

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

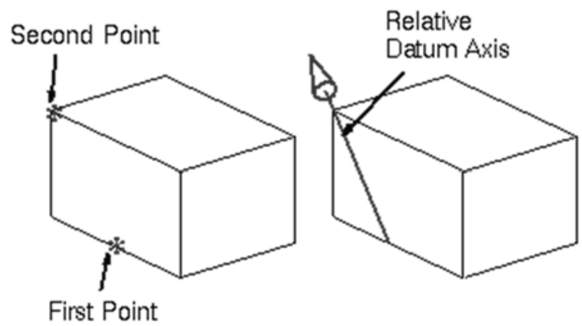
➢ Set the **Offset Distance** value as **15** inches in the dialog box and click **OK**



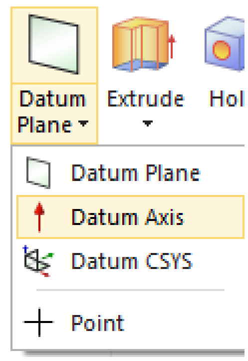
➢ If you don't see the complete model and plane, right-click and select ***FIT***

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).



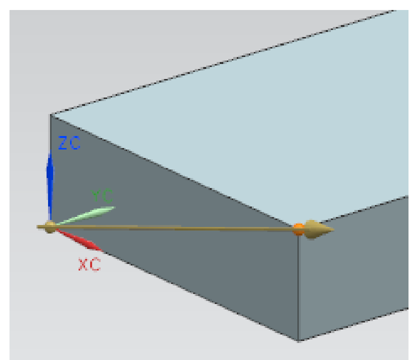
➢ Choose **Insert →Datum/Point →Datum Axis**



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

The next window allows you to choose the method of selecting the axis. Note that NX 12 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**

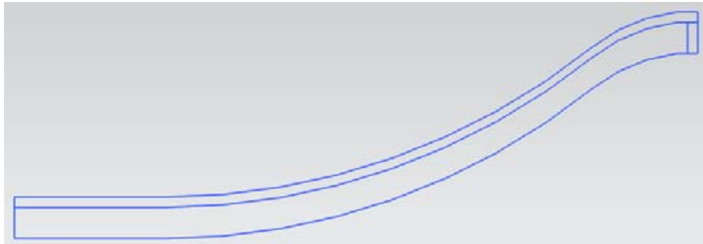
The *Datum Axis* will be a diagonal as shown.

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**

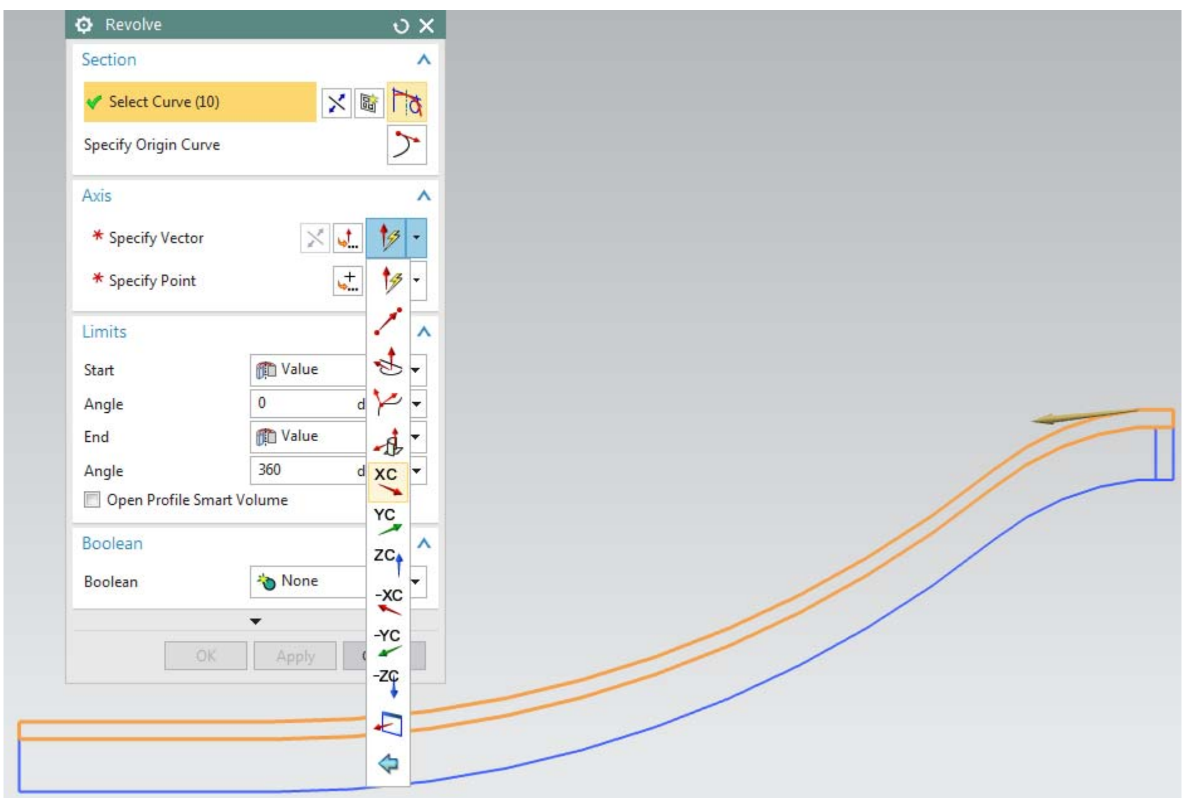
OR

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

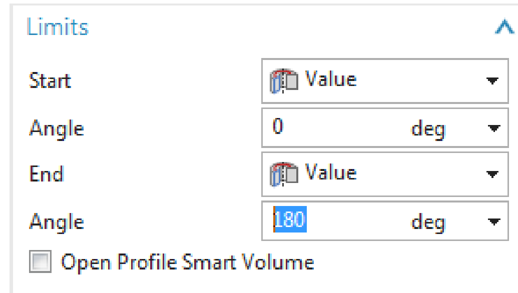
Make sure that the *Selection Filter* is set to *Single Curve* as shown below on the *Selection Filter Toolbar*

➢ 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**

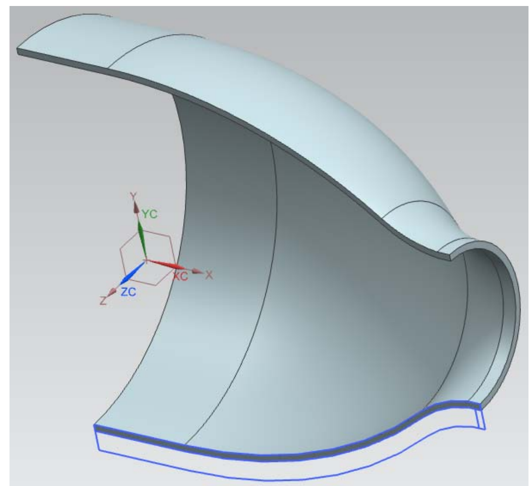


- In the **Specify Point** option, enter the coordinates **(0, 0, 0)** so the curve revolves around XC-axis with respect to the origin



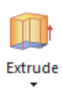
- Keep the **Start Angle** as **0** and enter **180** as the value for the **End Angle**
- Click **OK**

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

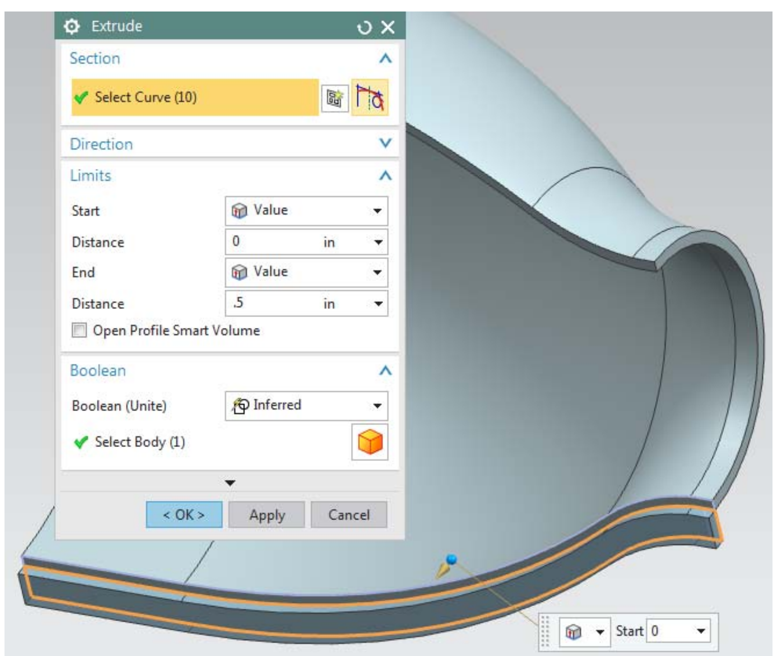


- Click on **Insert** →**Design Feature** →**Extrude**

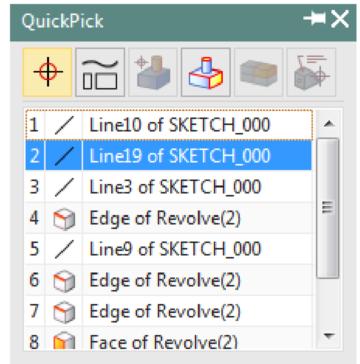
OR

- 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**).



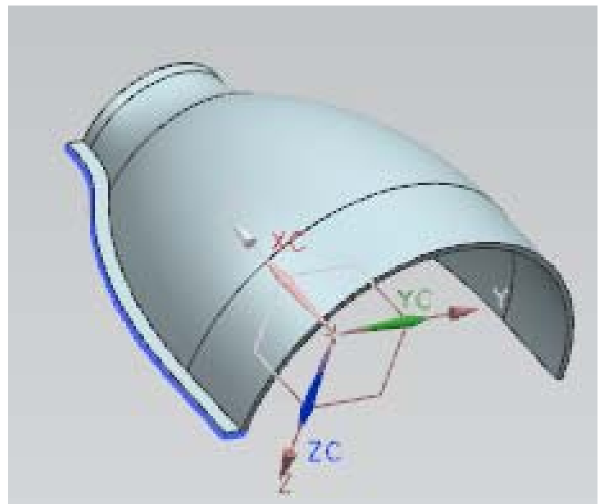
Note: In case you are not able to select the proper lines, left-click and hold the mouse button and you will see a dialog box pop-up as shown which will provide you the options of which curve to select.



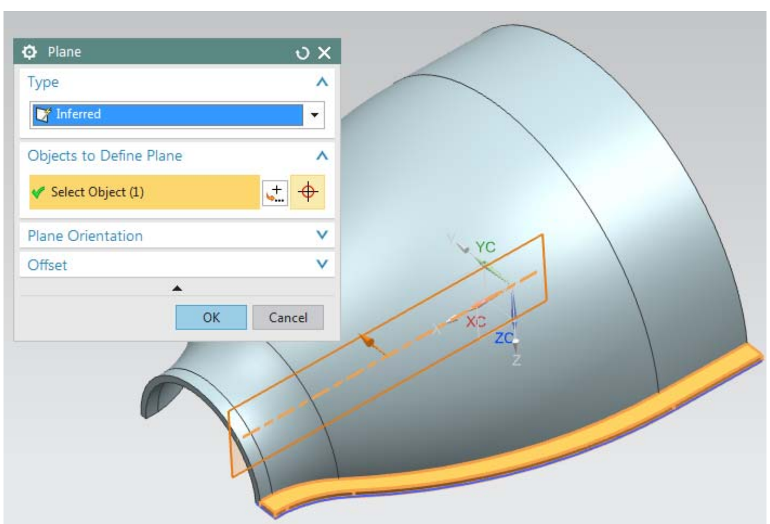
- Extrude this piece in the **negative Z-direction** by **0.5** inches

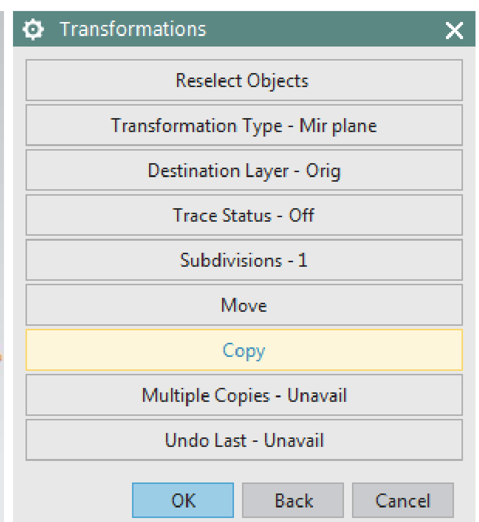
The final solid will be seen as follows.

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



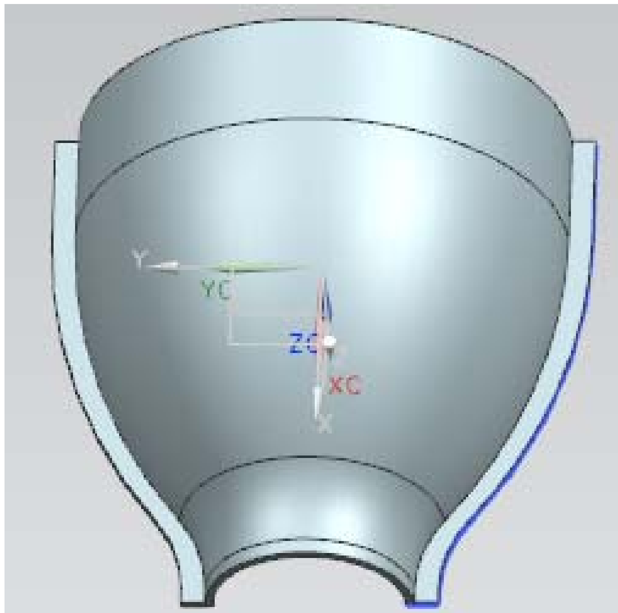
- Choose **Edit** →**Transform**
- Select the solid edge as shown. For this you will have to change the Filter in the dialog box to **Solid Body**
- Click **OK**
- Choose **Mirror Through a Plane**
- Select the **Center Line** as shown below
- Click **OK**
- Select **Copy**



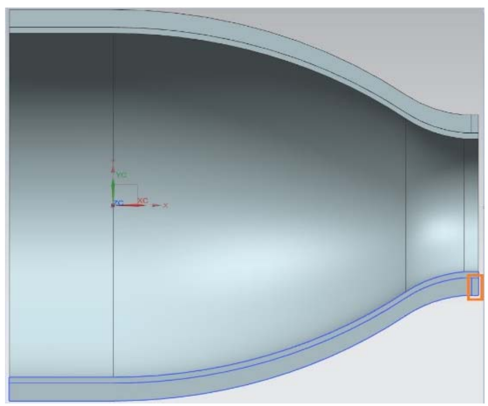


- Click **Cancel**

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.

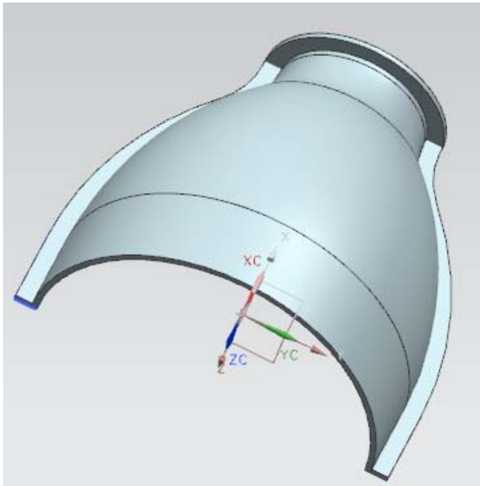


- ➢ Click on **Insert →Design Feature →Revolve**

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.



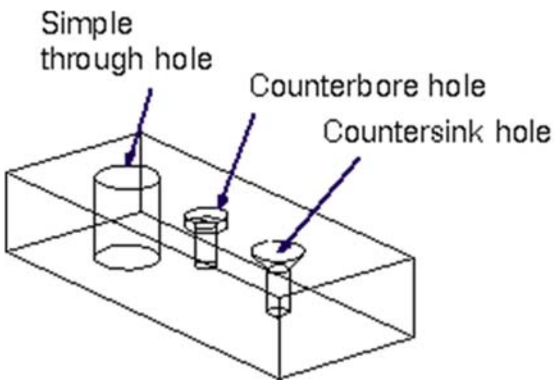
The lower casing is completed. Do not forget to save the model.

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, which are illustrated below.

4.5.1 General Hole

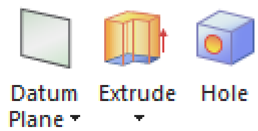
This option lets you create *Simple, Counterbored, Countersunk* and *Tapered* holes in solid bodies.



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

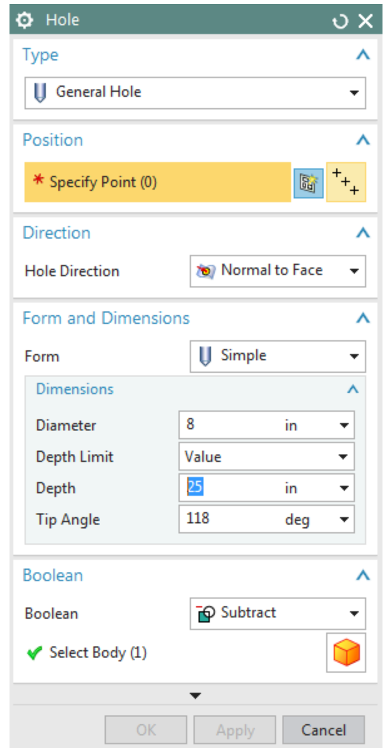
OR

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



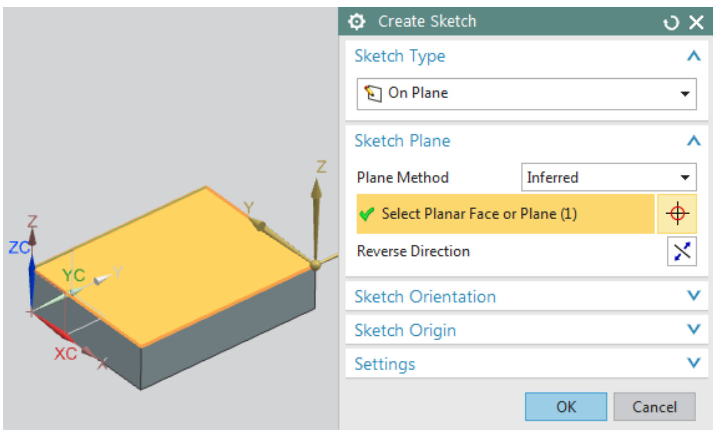
The *Hole* window will open. There are various selections that need to be done prior to making the holes. First you need to select the *Type* of the hole.

- ➢ Select the default **General Hole**



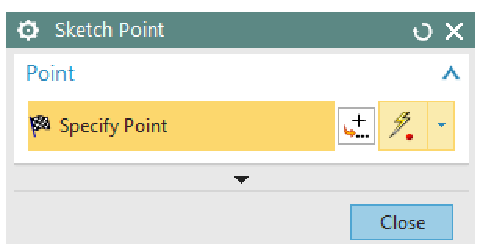
Next, you need to define the points at which you need to make the holes.

- ➢ 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

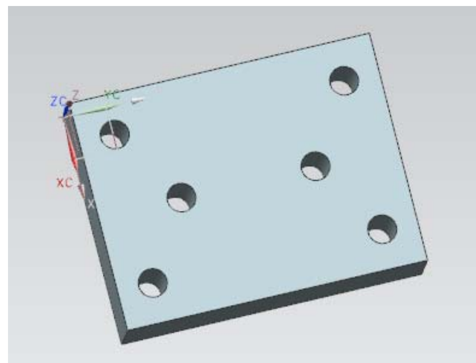
This will take you out of the *Sketch* mode and bring back to the original *Hole* window on the graphics screen.

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

  Diameter = **8** inches

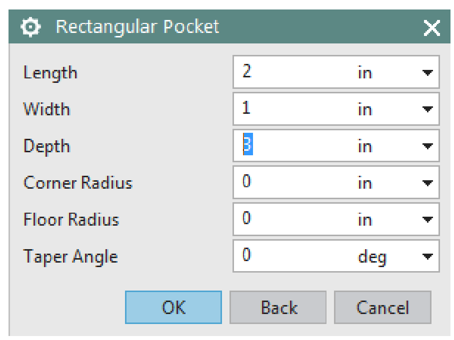
  Depth = **25** inches

  Tip Angle = **118** degrees



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

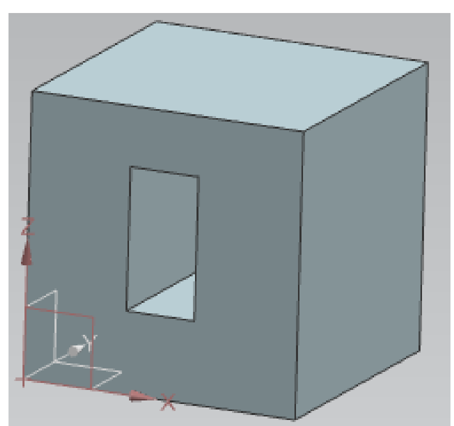
Make sure to save the model.



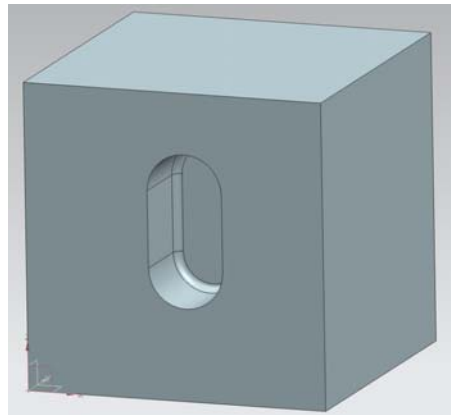
4.5.2 Pocket

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
- 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

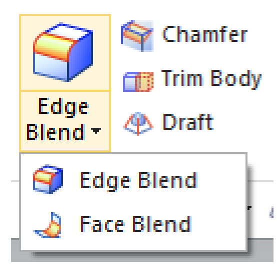
This option lets you create a groove in a solid body, as if a form tool moved inward (from an external placement face) or outward (from an internal placement face) on a revolving part, as with a turning operation. An example is shown on the right.

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

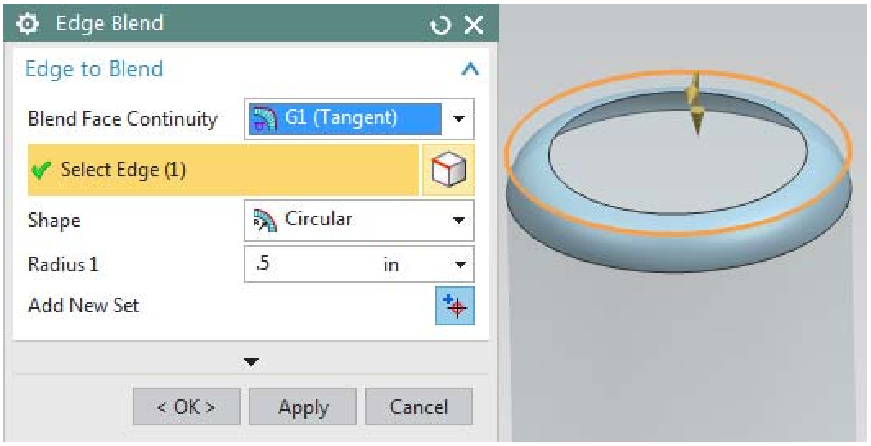
4.6 FEATURE OPERATIONS

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



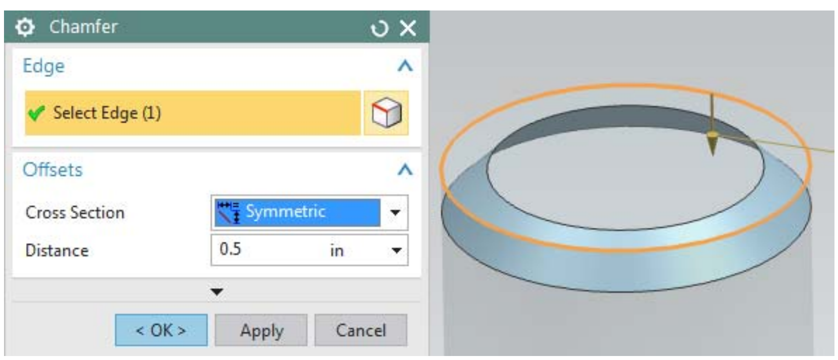
An *Edge Blend* is a radius blend that is tangent to the blended faces. This feature modifies a solid body by rounding selected edges. This command can be found under *Insert →Detail Feature →Edge Blend*. You can also click on its icon in the *Feature Group.* You need to select the edges to be blended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



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

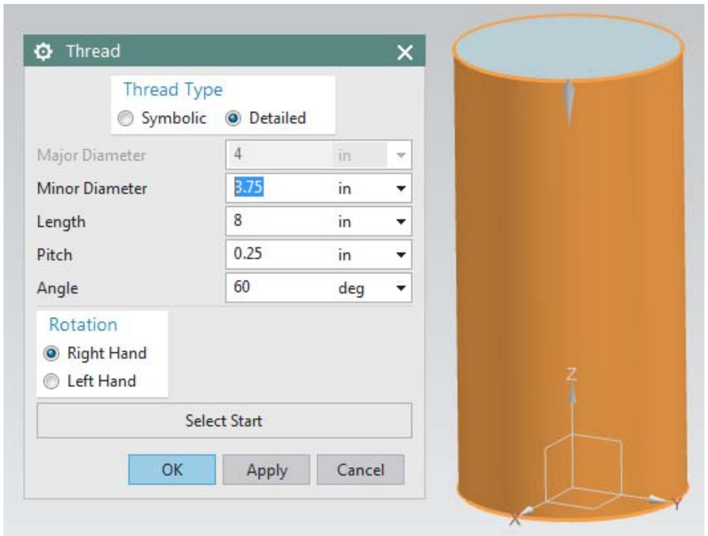
4.6.2 Chamfer

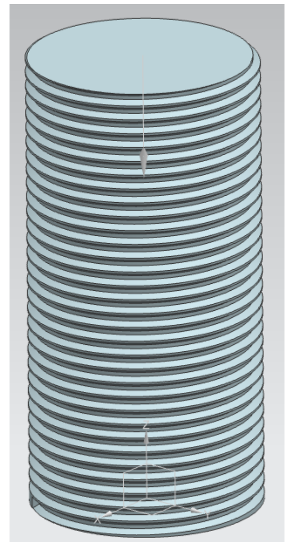
The *Chamfer Function* operates very similarly to the *Blend Function* by adding or subtracting material relative to whether the edge is an outside chamfer or an inside chamfer. This command can be found under *Insert →Detail Feature →Chamfer*. You can also click on its icon in the *Feature Group*. You need to select the edges to be chamfered and define the *Distance* of the *Chamfer* as shown below.



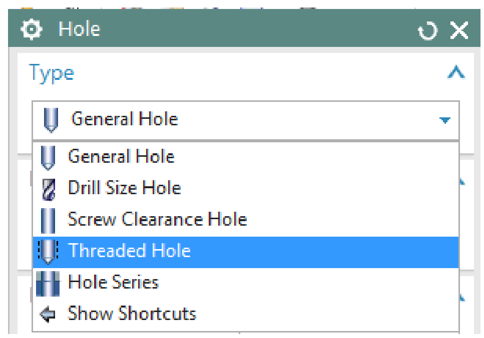
4.6.3 Thread

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.



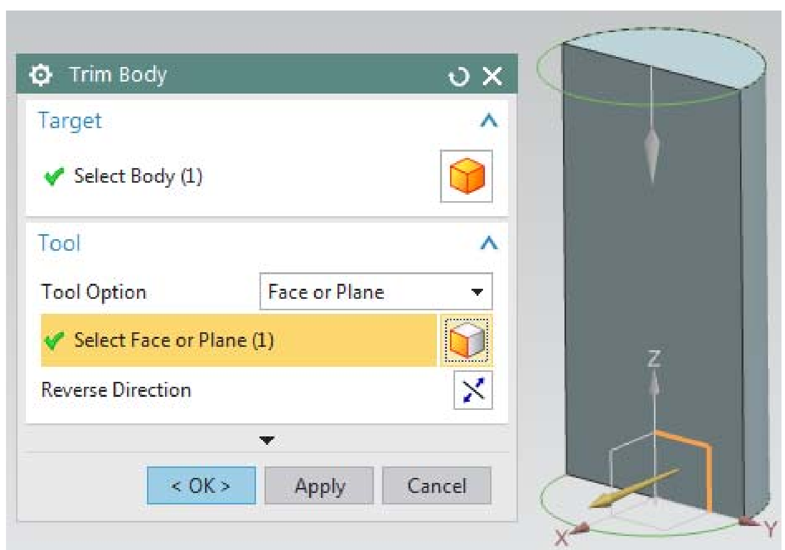


For *Threaded Holes,* it is recommended to use the *Threaded Hole* command instead of the *Thread* command: *Insert →Design Feature →Hole*



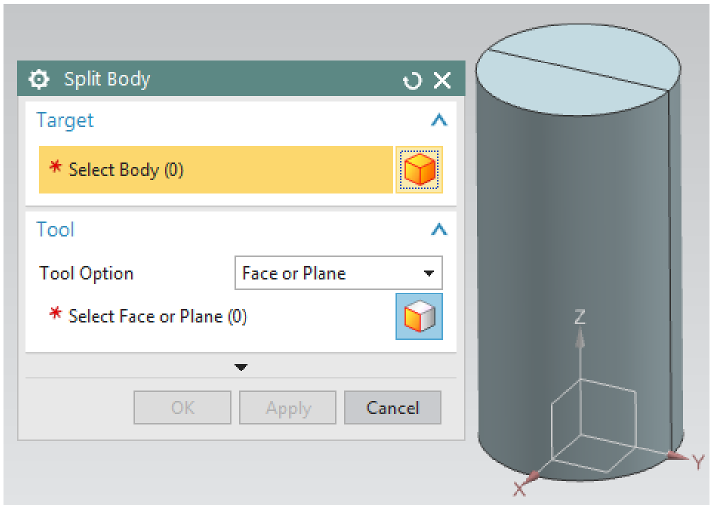
4.6.4 Trim Body

You can use the *Trim Body* function to trim a solid body with a sheet body or a *Datum Plane*, and at the same time retain parameters and associativity. To use this command, go to *Insert →Trim →Trim Body* or click on its icon in the *Feature Group.* An example is shown on the right.



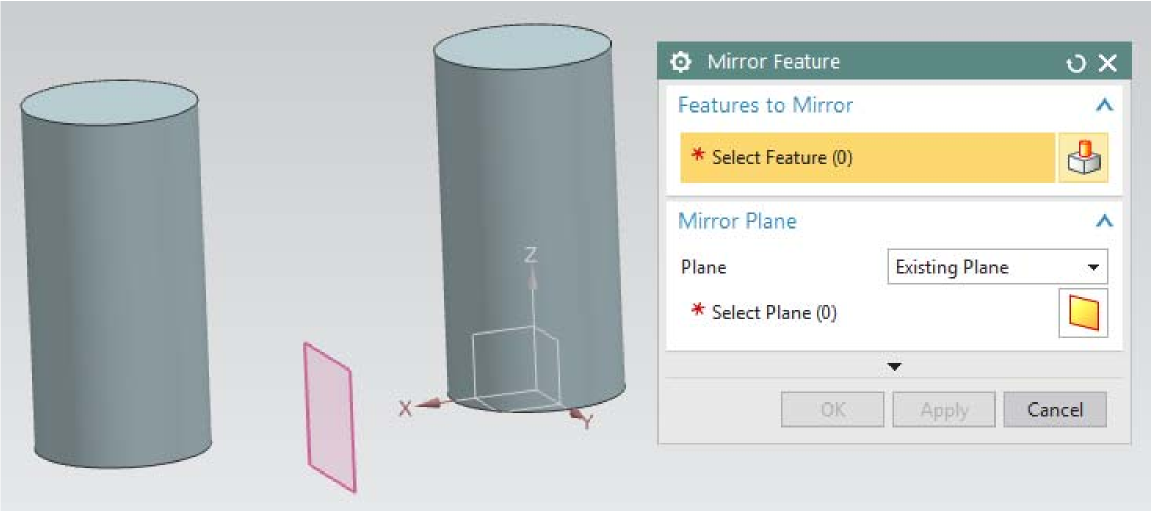
4.6.5 Split Body

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



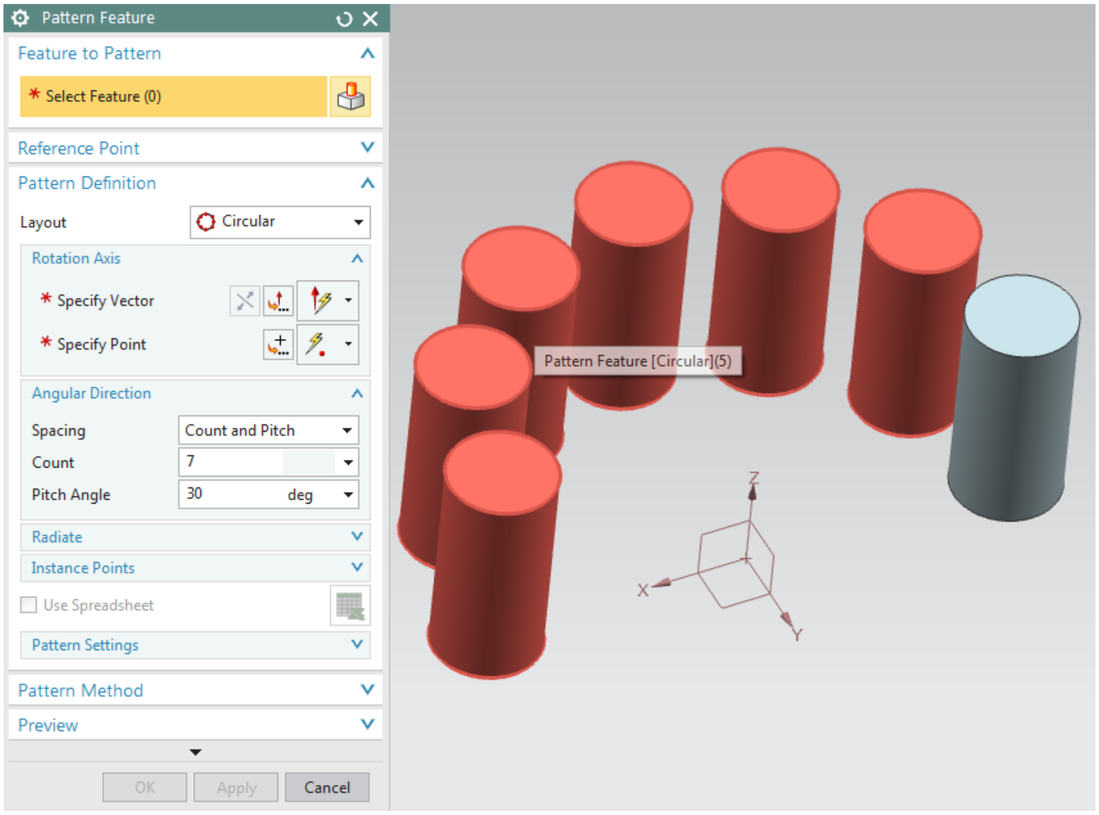
4.6.6 Mirror

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

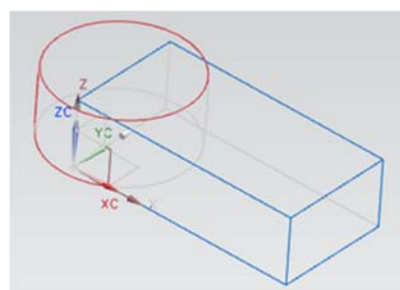


4.6.7 Pattern

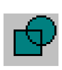
A *Design Feature* or a *Detail Feature* can be made into dependent copies in the form of an *Array*. It can be *Linear, Circular, Polygon, Spiral,* etc. This particularly helpful feature saves plenty of time and modeling when you have similar features. For example threads of a gear or holes on a mounting 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.



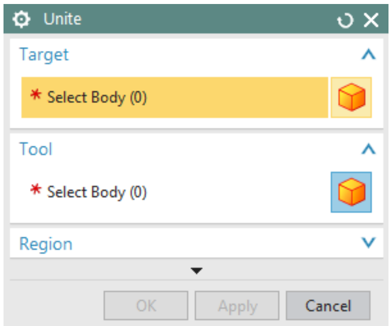
4.6.8 Boolean Operations

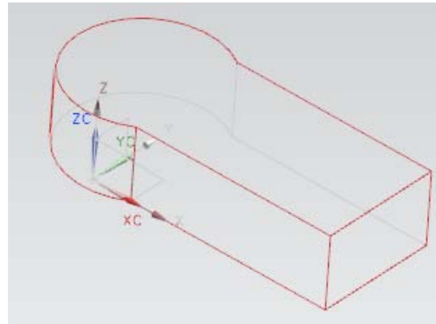


There are three types of *Boolean Operations: Unite, Subtract,* and *Intersect*. These options can be used when two or more solid bodies share the same model space in the part file. To use this command, go to *Insert* → *Combine* or click on their icons in the *Feature Group*. Consider two solids given: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

4.6.8.1 Unite 

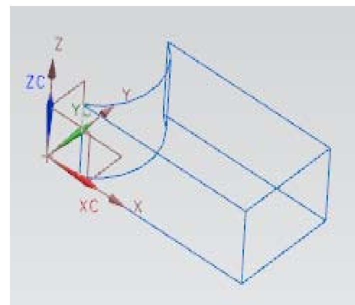
The unite command adds the *Tool body* with the *Target body*. For the above example, the output will be as follows if *Unite* option is used.

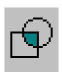




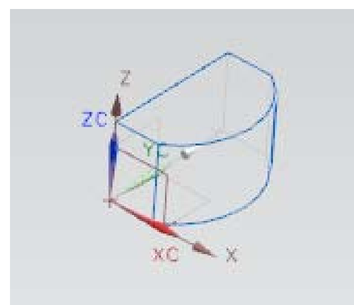
4.6.8.2 Subtract 

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



4.6.8.3 Intersect 

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

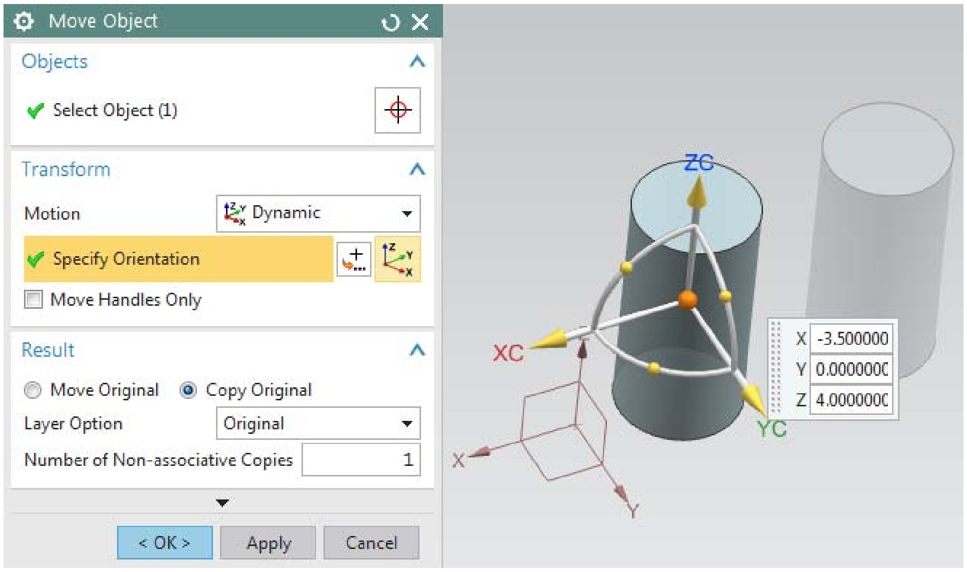


4.6.9 Move

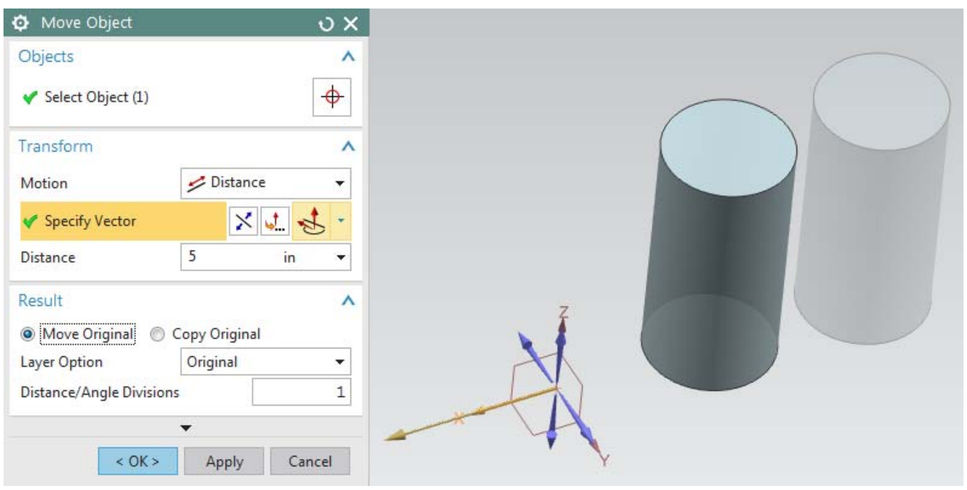
If you want to *Move* an object with respect to a fixed entity,

➢ Click on **Edit →Move Object**

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



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



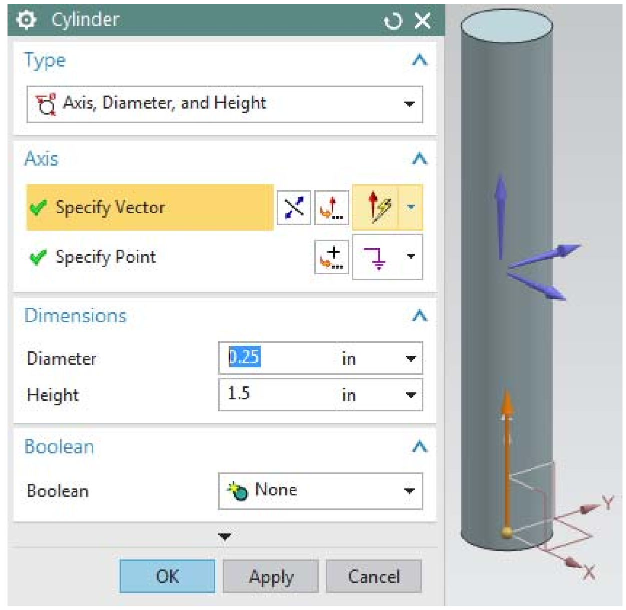
- ➢ Click on **Specify Vector** and select the direction.
- ➢ For example, type **5** in the **Distance** box. This will translate the cylinder a distance of 5 inches along X-Axis
- ➢ Click **OK**

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

4.7 EXAMPLES

4.7.1 Hexagonal Screw

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



- Choose **Insert** →**Design Feature** → **Cylinder**
- The cylinder should be pointing in the **Positive ZC-Direction** with the center set at the **Origin** and with the following dimensions:

  Diameter = **0.25** inches

  Height = **1.5** inches

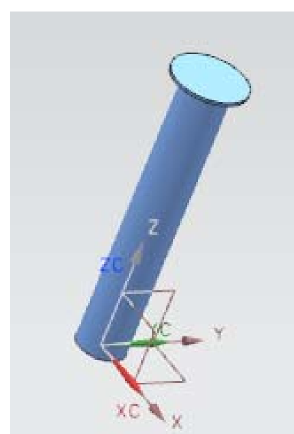
Now create a small step cylinder on top of the existing cylinder.

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

  Diameter = **0.387** inches

  Height = **0.0156** inches

- Click on the top face of the existing cylinder



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

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

- 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.

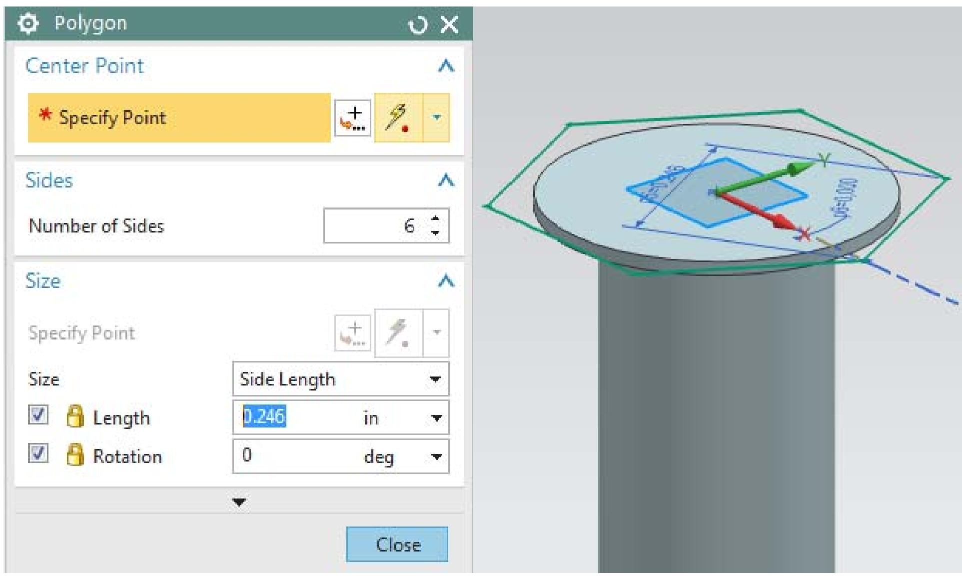
- *Inscribed Radius*
- *Circumscribed Radius*
- *Side Length*

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

Length = **0.246** inches

Rotation = **0.00** degree

➢ Click **OK**



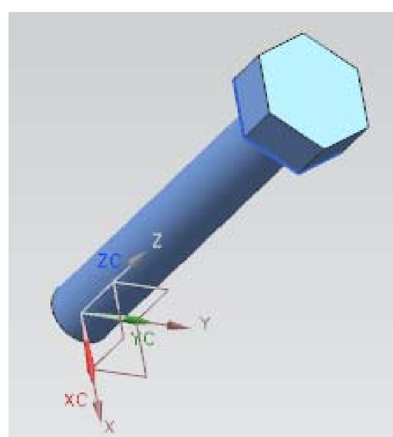
Now we will extrude this polygon.

➢ Choose **Insert** →**Design Feature** →**Extrude**

➢ Choose the **Hexagon** to be extruded

➢ Enter the **End Distance** as **0.1876** inches

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

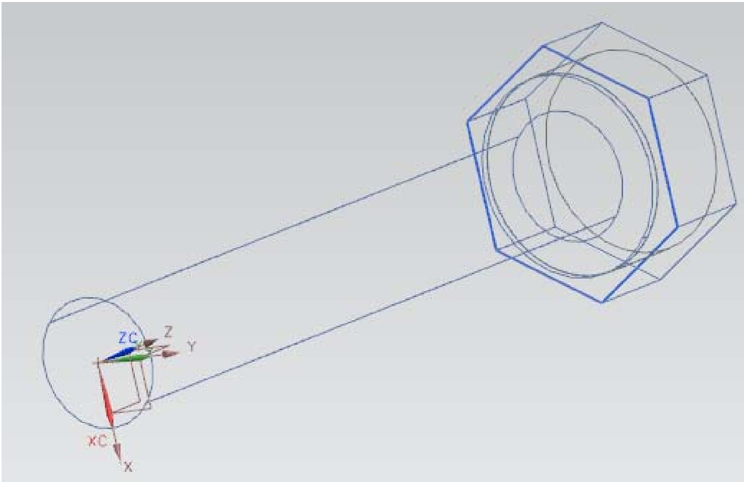


➢ On top of the cylinder that has a diameter of 0.387 inches, insert another cylinder with the following dimensions.

Diameter = **0.387** inches

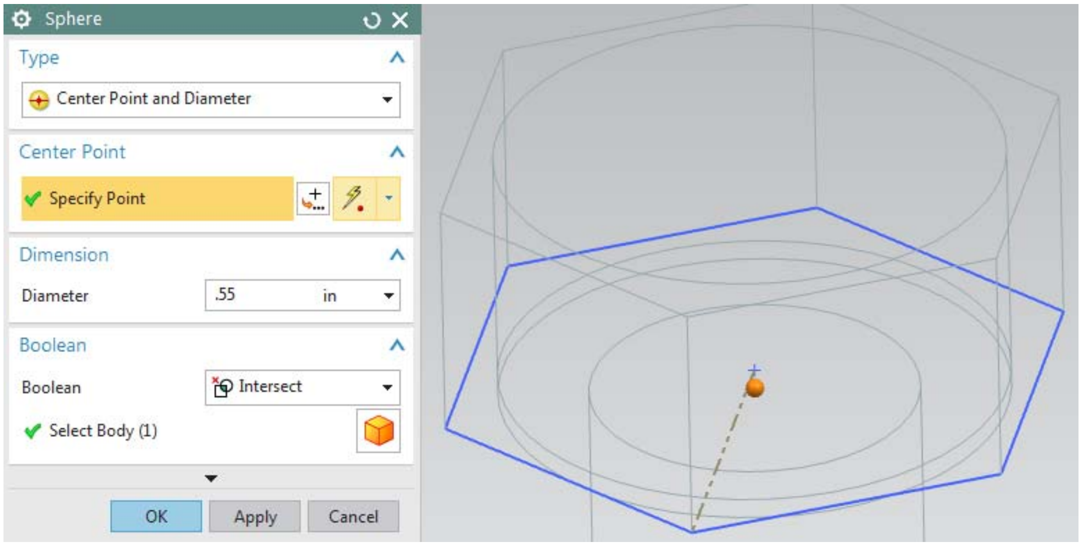
Height = **0.1875** inches

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



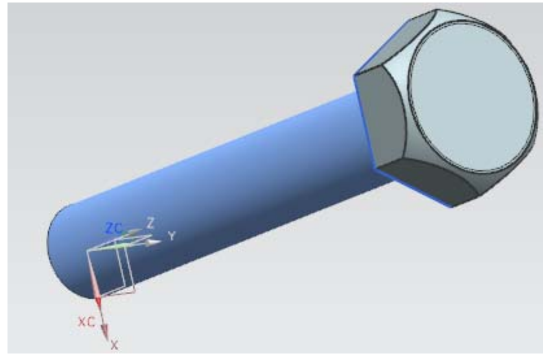
We will now use the feature operation *Intersect*.

- 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*



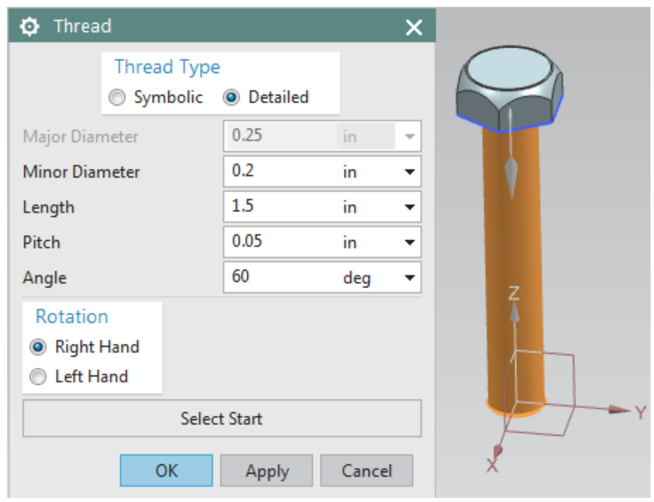
- Choose the hexagonal head
- Click **OK**

This will give you the hexagonal bolt as shown.

Note: This blend feature on the bolt hat can be created also by revolving cut with a section about its axis, you can try it out.

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

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

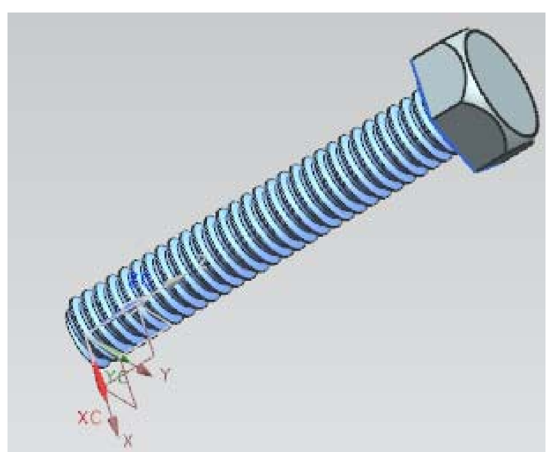


- Click on the **Detailed** radio button
- Keep the **Rotation** to be **Right Hand**
- Click on the bolt shaft (the long cylinder below the hexagon head)

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 figure on the right. Save the model.

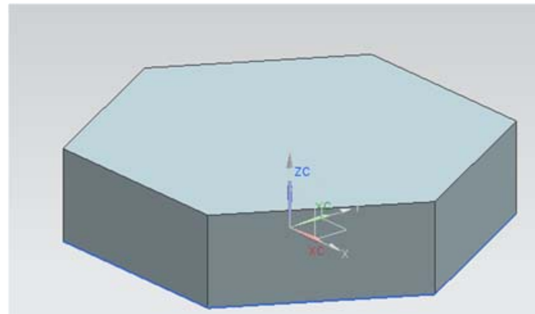


4.7.2 Hexagonal Nut

- Create a new file and save it as **Impeller_hexa-nut.prt**
- Choose **Insert** →**Curve** →**Polygon**
- Input **Number of Dides** 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.

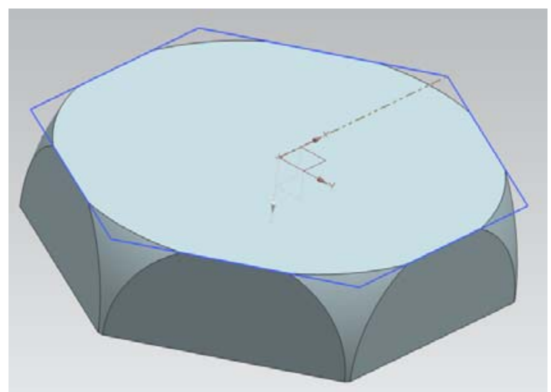
➢ Choose **Insert** →**Design Feature** →**Sphere**

➢ 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**



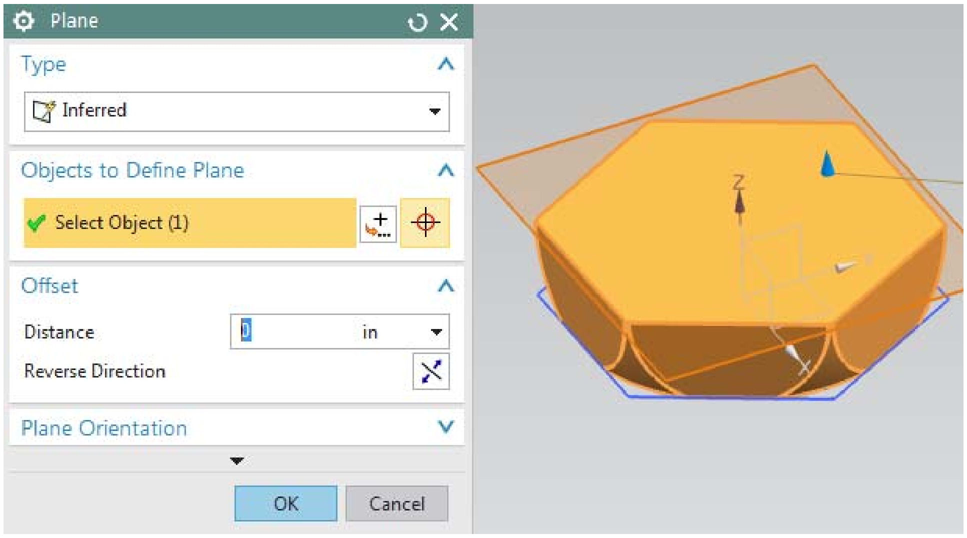
The model will look like the following. We will now use a *Mirror* command to create the other side of the *Nut.*

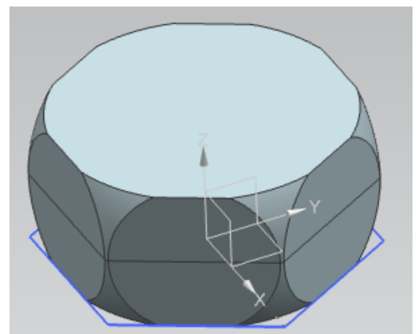
➢ Choose **Edit** →**Transform**

➢ 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





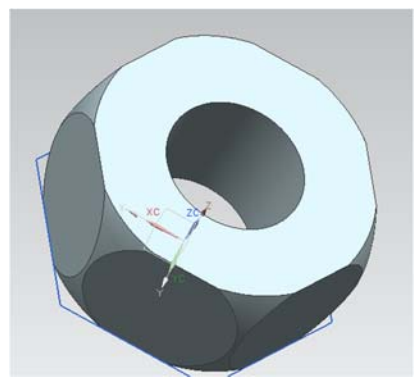
- Click on **OK**
- Click on **Copy**
- Click **Cancel**

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

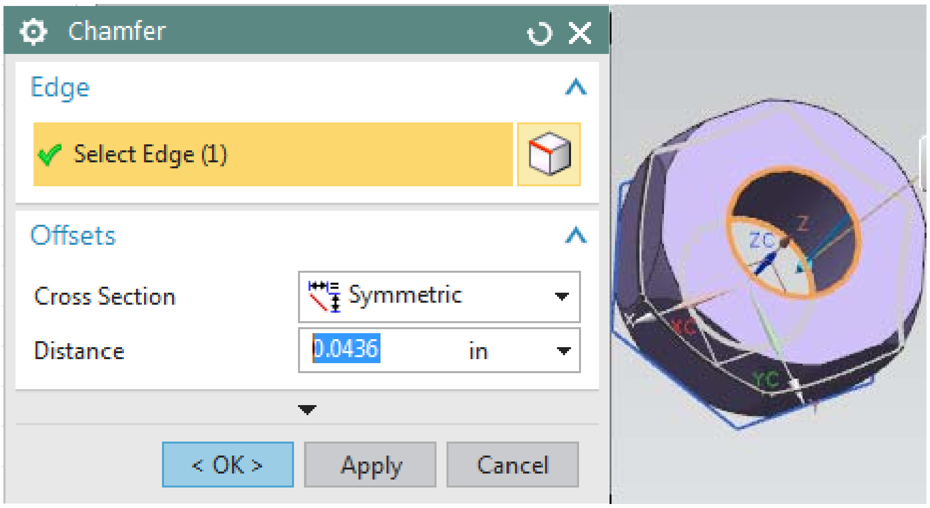
  Height = **1** inch



- 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

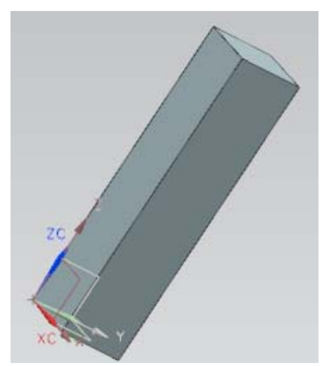
Here, we will make use of some *Primitives* and *Feature Operations* such as *Edge Blend, Chamfer*, and *Subtract*. It should be noted that the same model can be more efficiently created by *2D Sketching* and *Extruding*, but *Primitives* are used here to familiarize the users with these features.

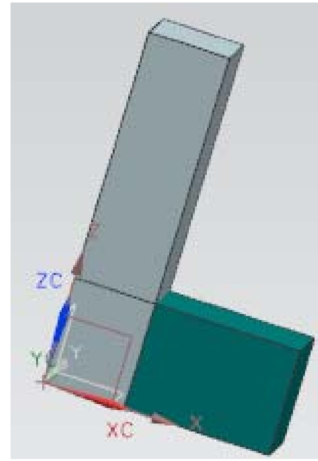
- Create a new file and save it as **Arborpress_L-bar**
- Choose **Insert** →**Design Feature** →**Block**
- Create a **Block** with the following dimensions:

  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:

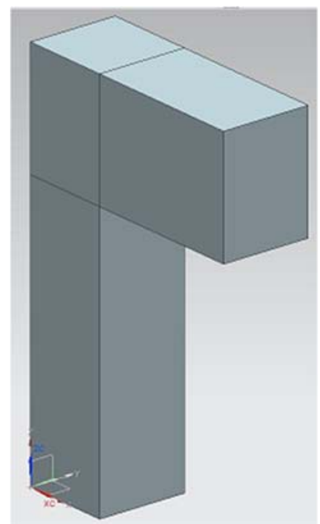
  Length = **182** inches

  Width = **65** inches

  Height = **85** inches

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

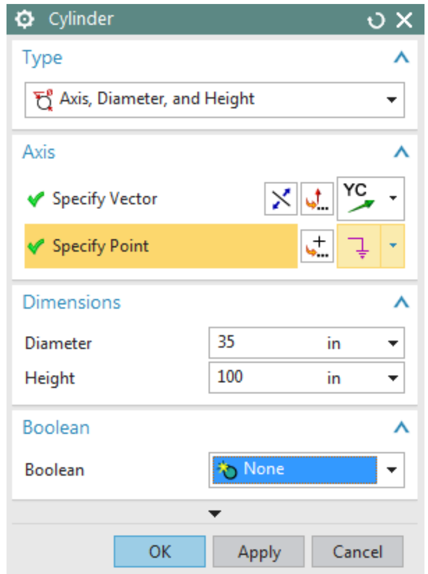
- Click **Edit** →**Move Object**

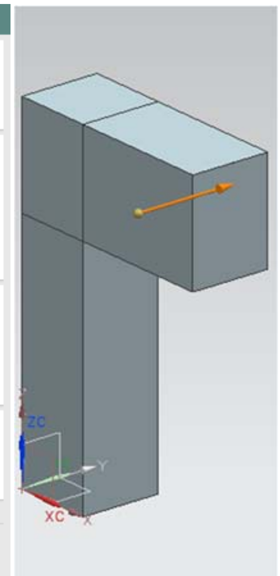


- Select the second block (green) and click **OK**
- Choose the **Motion** as **Distance**
- Select the positive **ZC** in the **Specify Vector** dialog
- Enter **200** as the **Distance** value
- Make sure that **Move Original** button is checked and click **OK**
- Click **Move** and then **Cancel** 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**





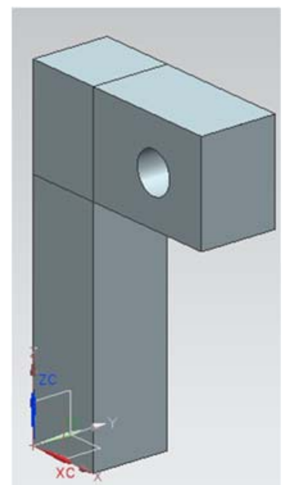
- ➢ On the **Specify Vector**, select the **YC Axis** icon
- ➢ In the **Specify Point**, enter the following values:

  XC = 130
  YC = -5
  ZC = 242

- ➢ The cylinder should have the following dimensions:

  Diameter = **35** inches

  Height = **100** inches



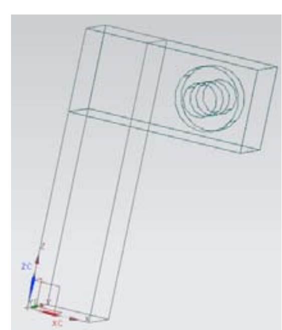
- ➢ Under the **Boolean** drop-down window, choose **Subtract**
- ➢ Select the horizontal block at the top

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.

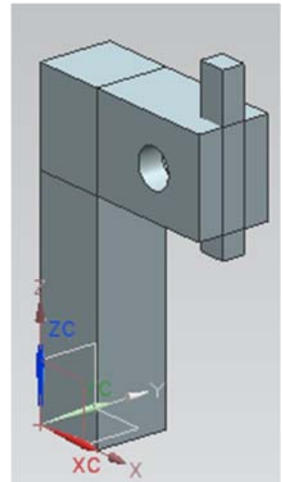


- ➢ Choose **Insert** →**Design Feature** →**Block**
- ➢ Create a block with the following dimensions:

  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**



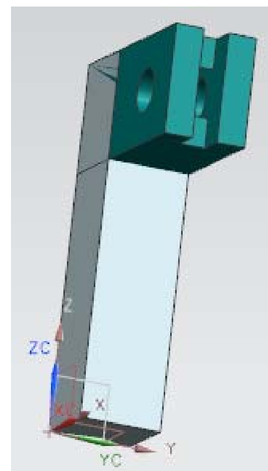
The model will look like the figure on the right. Now we will subtract this block from the block with the hole.

➢ 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**

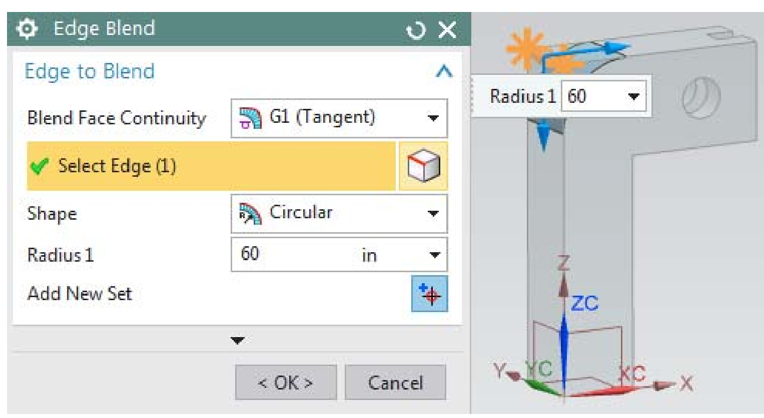
The two blocks are now combined into one solid model.

➢ 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

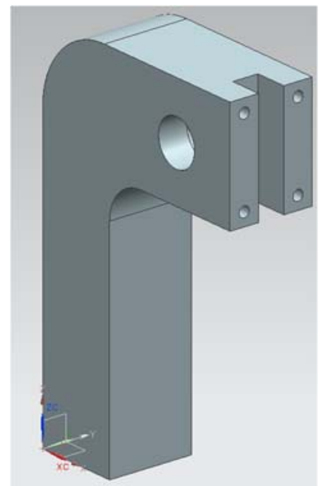
- Construct them in the **XC**-direction at the following point coordinates:

  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.

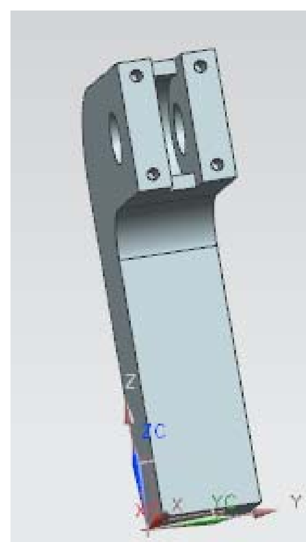
- Create a **Block** with the following dimensions:

  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***.**

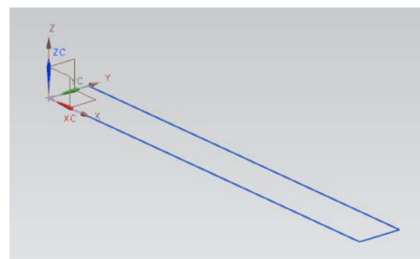
For *Corner Point* 1,

➢ 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**.

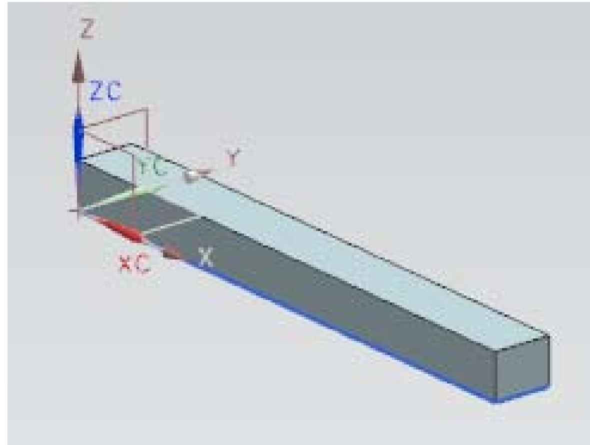
➢ Choose the default **Positive ZC**-direction as the **Direction**

➢ In the **Limits** window, type in the following values:

Start = **0**
End = **20**

➢ Click **OK**

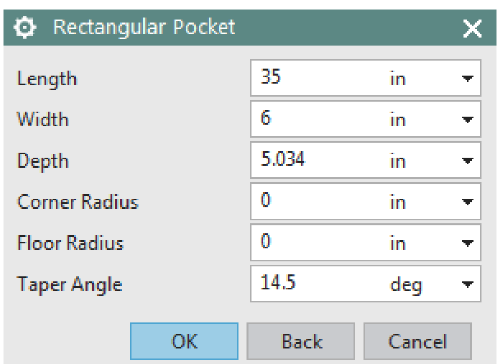
The extruded body will appear as shown on the right.

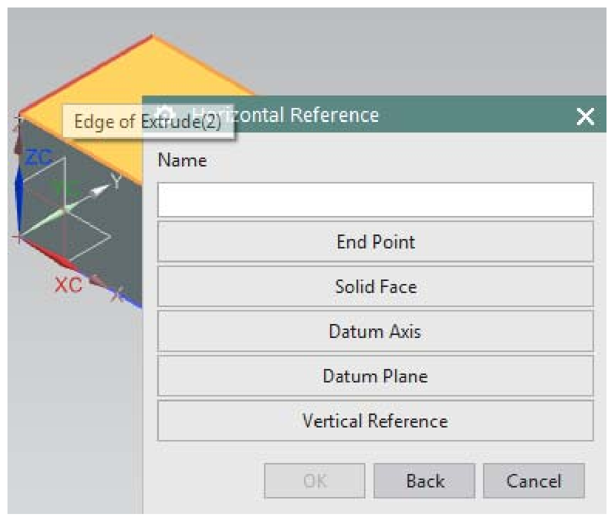


- Choose **Insert** →**Design Feature** → **Pocket**
- Choose **Rectangular** in the pop up window
- Click on the top surface of the rack
- Click on the edge as shown in the figure for the ***Horizontal Reference***

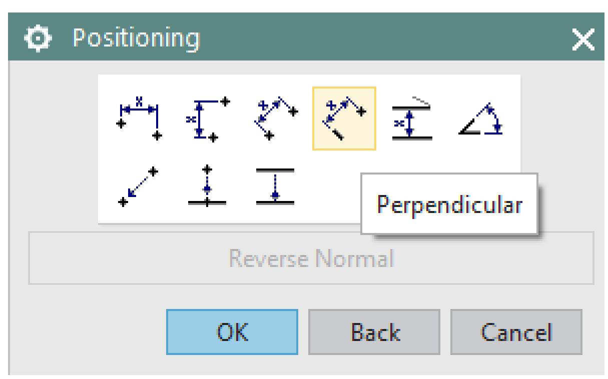
This will pop up the parameters window.

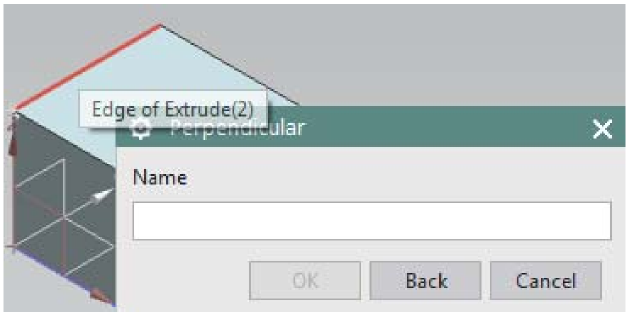
- Enter the values of parameters as shown in the figure and choose **OK**

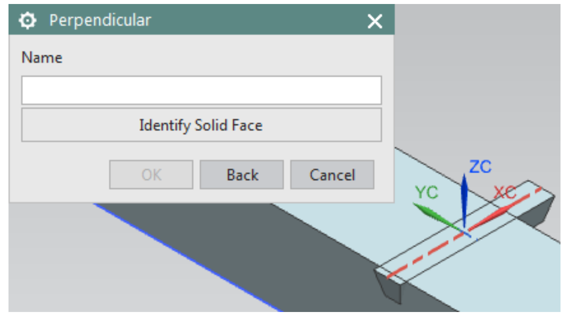




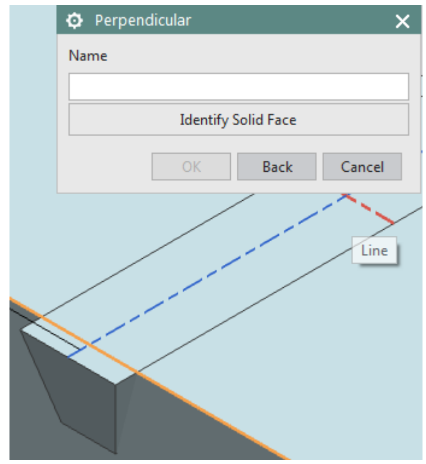
- When the **Positioning** window pops up, choose the **PERPENDICULAR** option
- Click on the edge of the solid and then click on the blue dotted line as shown below





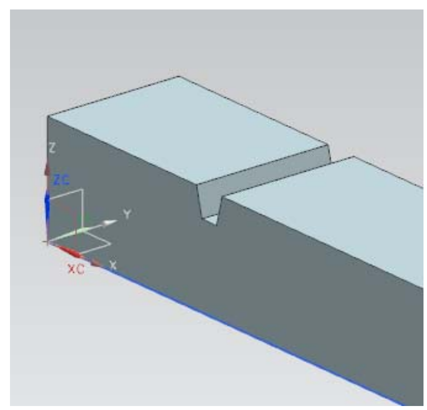


- ➢ Enter the **Expression** value as **37.8** and Choose **OK**
- ➢ Once again pick the **Perpendicular** option and then choose the other set of the edges along the Y-Axis, as shown on the right (the one perpendicular to the last blue line selected)



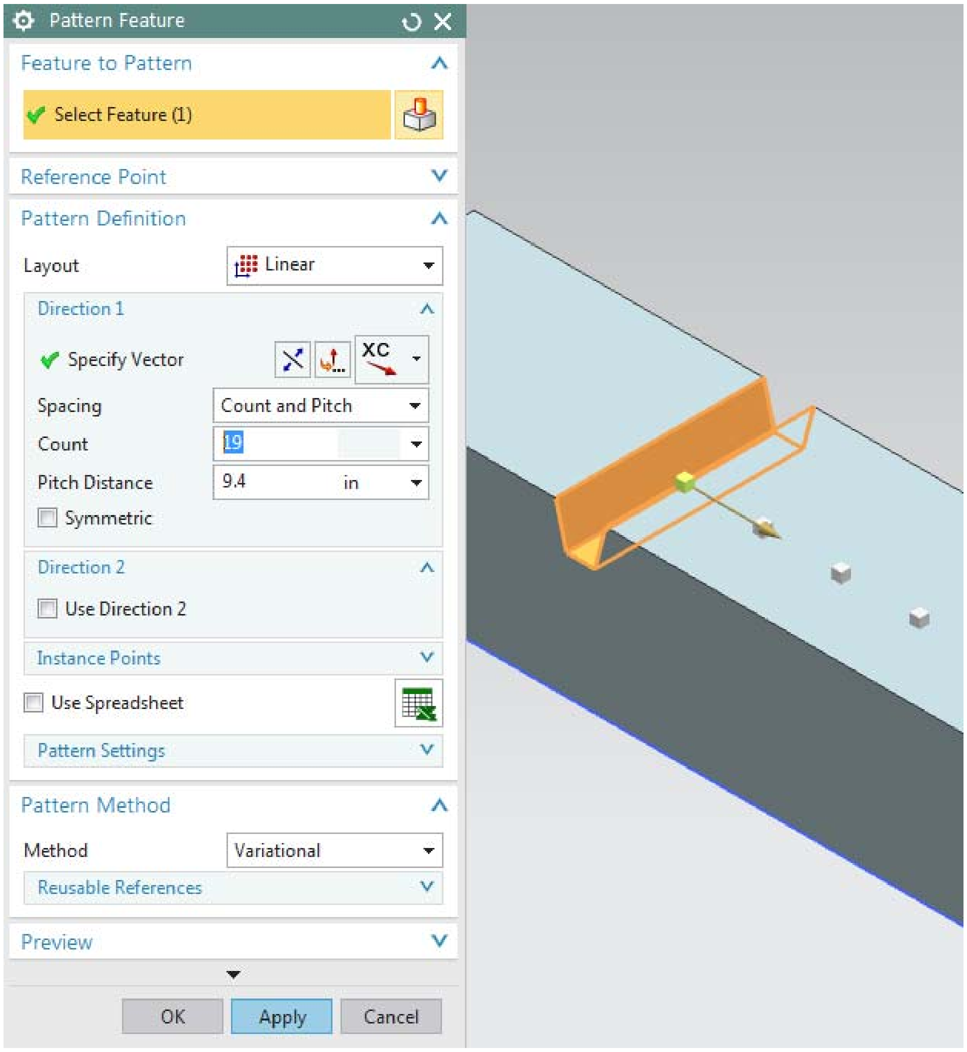
- ➢ Enter the expression value as **10** and click **OK**
- ➢ Click **OK** and then **Cancel**

The model will now look as follows.

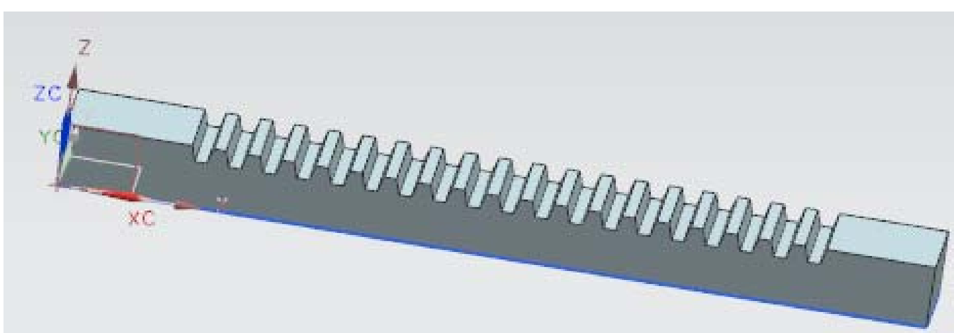


Let us create the instances of the slot as the teeth of the *Rack* to be meshed with *Pinion*.

- ➢ 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**

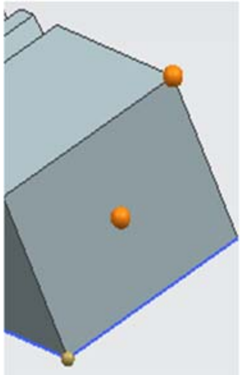


The model of the *Rack* will look as the one shown in the figure.



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**

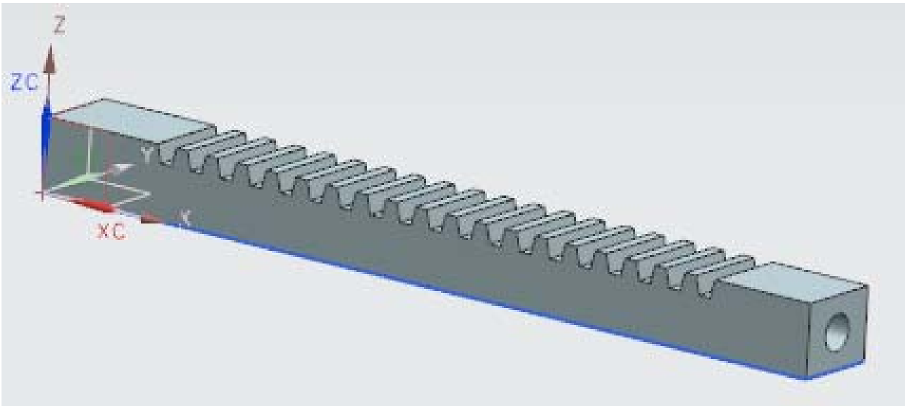


- Choose **–XC**-Direction in the **Specify Vector** dialog box
- Click on the **Point Dialog**
- In the **Points** dialog box select **Between Two Points** option and select the points as shown in the figure on the right (diagonally opposite points). The option selects the midpoint of the face for us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

  Diameter = **10** inches

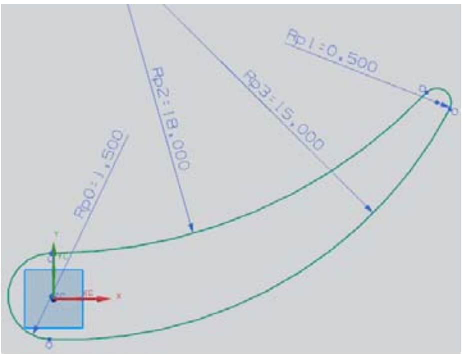
  Height = **20** inches
- Choose **Subtract** in the **Boolean** dialog box

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

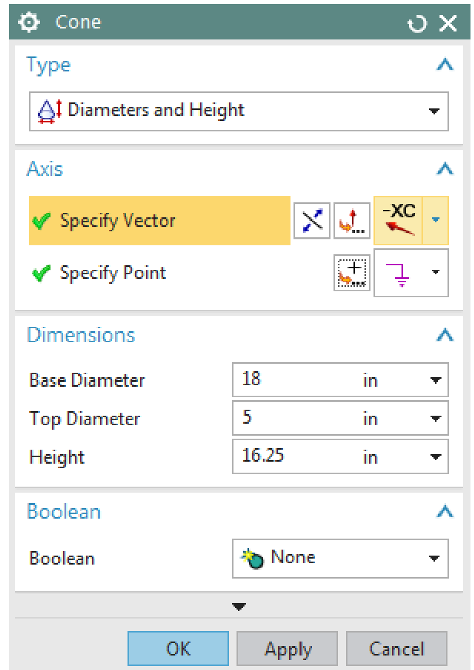


4.7.5 Impeller

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



Now let us model a cone first.



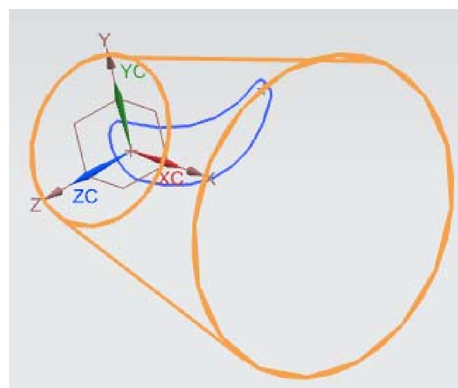
- Choose **Insert** →**Design Feature** →**Cone**
- Select **Diameters and Height**
- Select the **–XC**-Direction in the **Specify Vector** dialog box
- In the **Point Dialog**, enter the coordinates **(14, 0, 0)**.
- Enter the following dimensions:

  Base Diameter = **15** inches

  Top Diameter = **8** inches

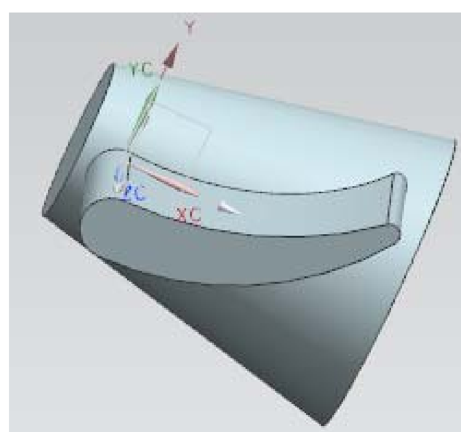
  Height = **16.25** inches

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



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

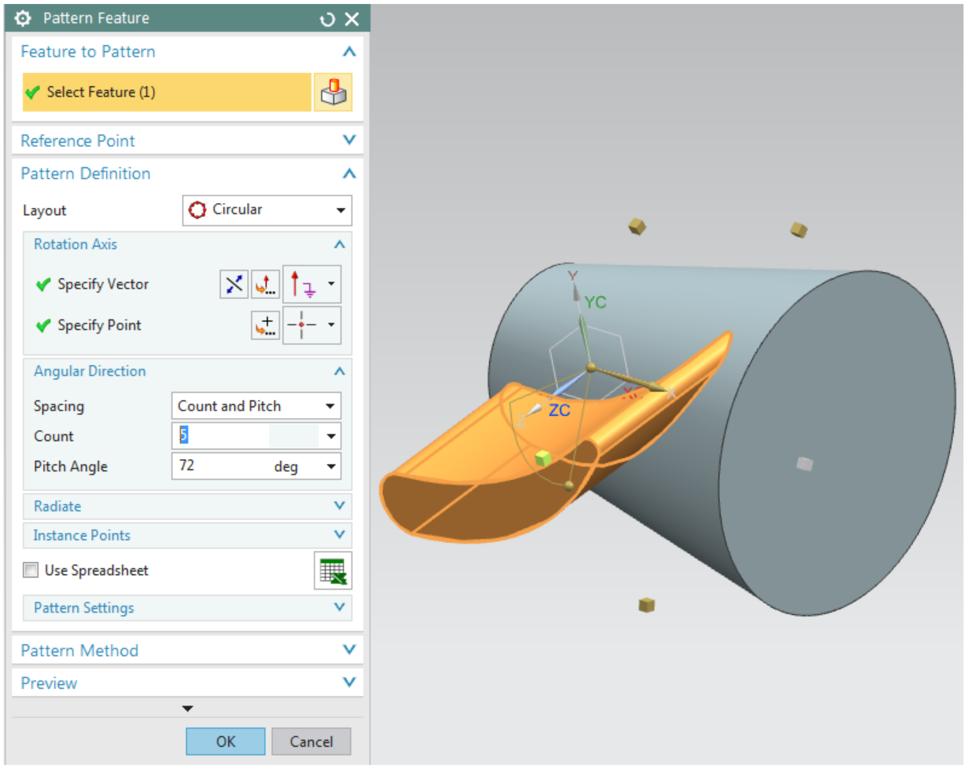
The model will be as follows.



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

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

➢ Click **OK**

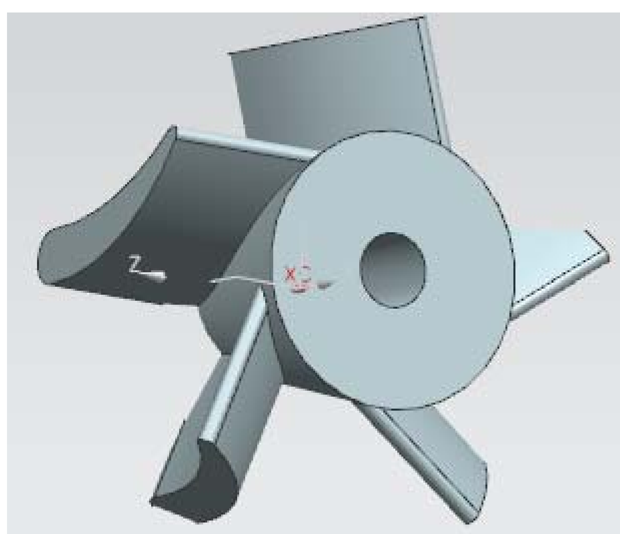


Now, let us create two holes in the cone for the shaft and the locking pin. Note that these holes can also be created by *Hole* menu option.

➢ **Subtract** a cylinder with a **Diameter** of **4** inches and a **Height** of **16** inches from the side of the cone with the larger diameter

➢ **Subtract** another cylinder with a **Diameter** of **0.275** inches and a **Height** of **0.25** inches from the side of the cone with the smaller diameter

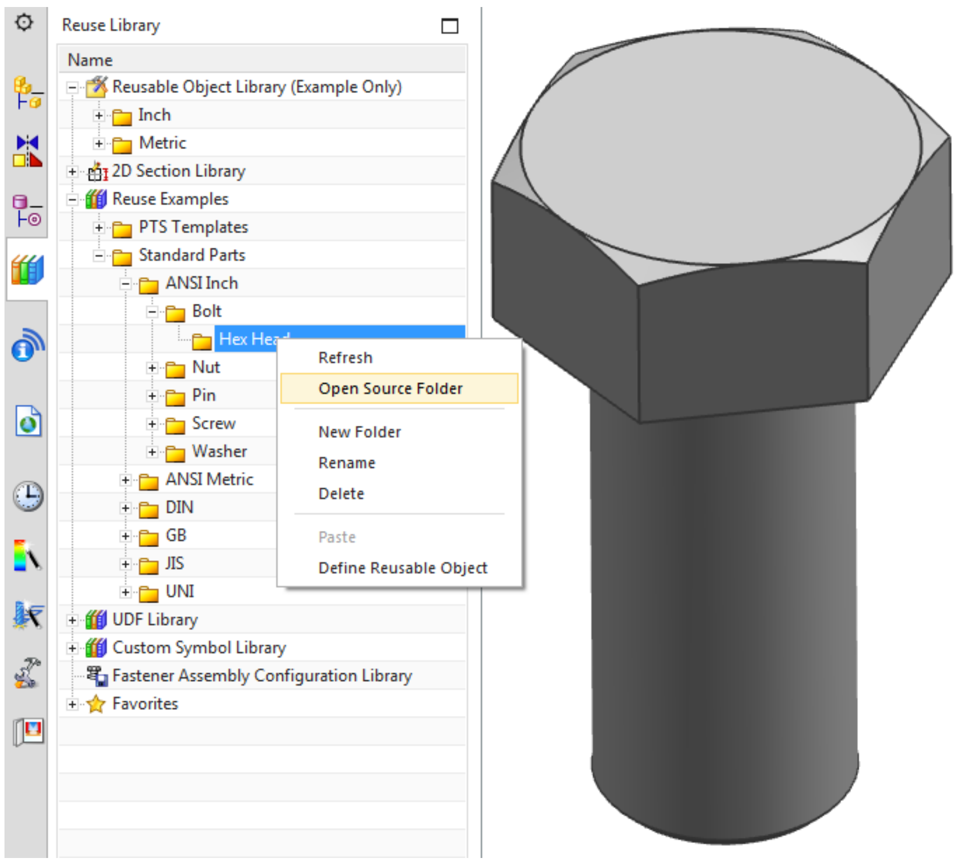
The final model will look like the figure on the right. Save and close your work.



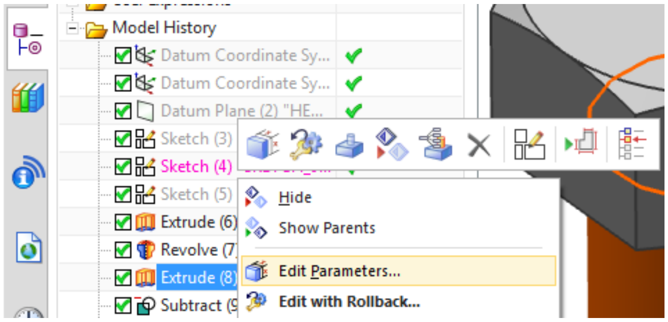
4.8 STANDARD PARTS LIBRARY

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

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



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



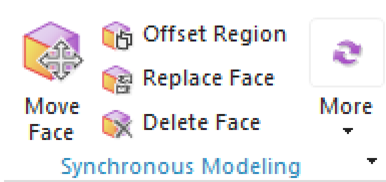
4.9 SYNCHRONOUS TECHNOLOGY

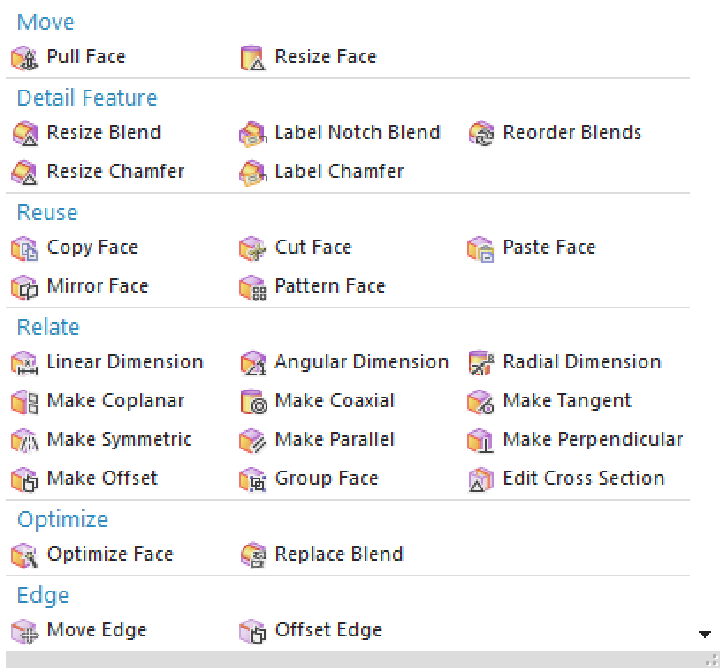
One of the important and unique features which NX offers apart from *Design Features* and *Freeform Modeling* is *Synchronous Technology*. With the options available in *Synchronous Modeling* group in the ribbon bar in the *Modeling Application* tab, the user can modify complex 3D models without the model history tree and without knowing the feature relationships and dependencies. The "push-and-pull" options can be used to modify the 3D model using faces, edges and cross-sections. NX 12 supports the *Synchronous Modeling* to work with 3D models from CATIA, Pro/ENGINEER®, SolidWorks®, and Autodesk Inventor®, apart from the standard formats including IGES, ISO/STEP and JT.

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

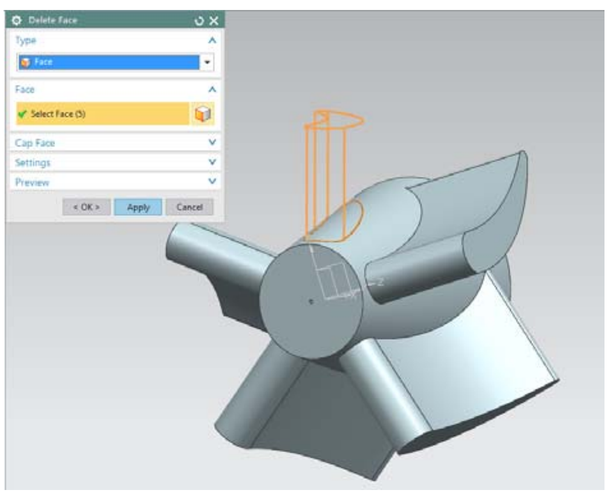
- Open a new file in NX
- Choose **File →Import →Impeller_impeller.stp**

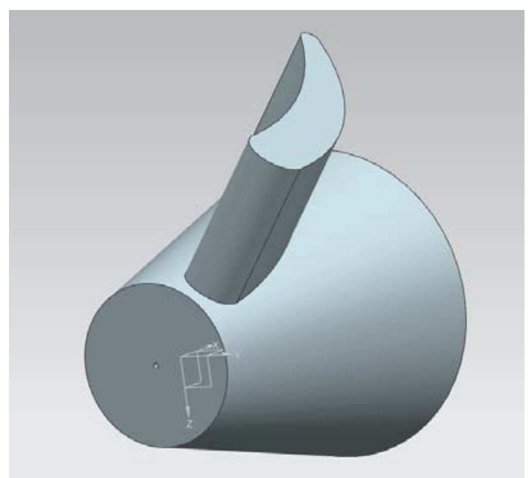
Observe here that the .stp file would not have any model history. We will explore some of the options available in the *Synchronous Modeling* group in the ribbon bar. Click *More* to view a comprehensive list of options available in synchronous modeling.





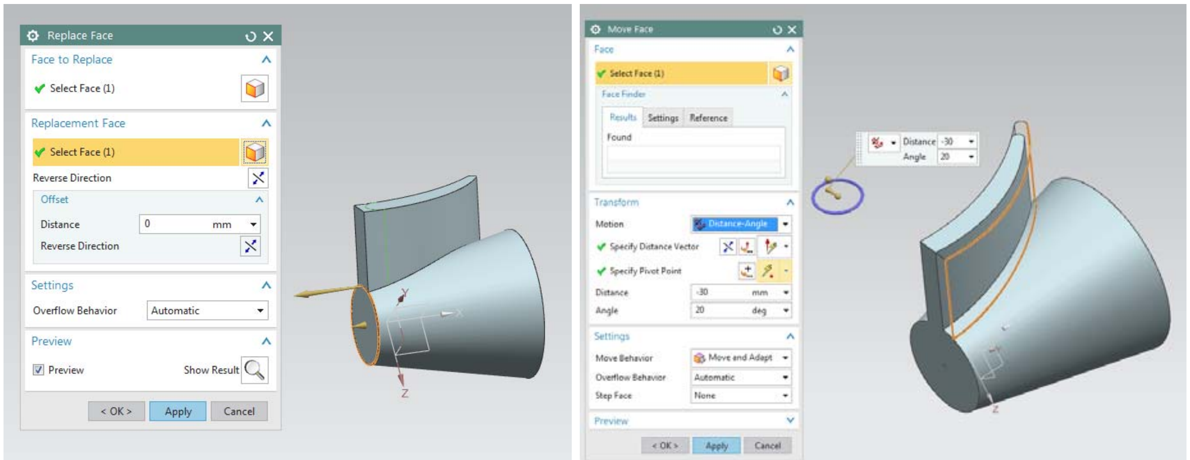
- Click **Delete Face** and select the faces of the blade to delete the blade
- Repeat the process and delete all except one blade. The part should look as shown below.



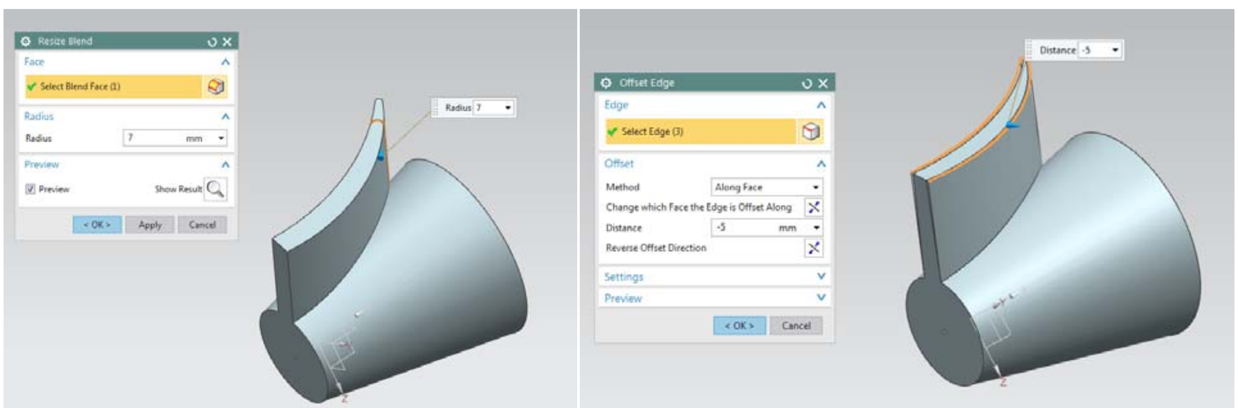


- Click **Replace Face** and select the end face of the blade with large blend radius as **Face to Replace** and select the flat surface of the cone with smaller diameter as the **Replacement Face** to delete the blade.

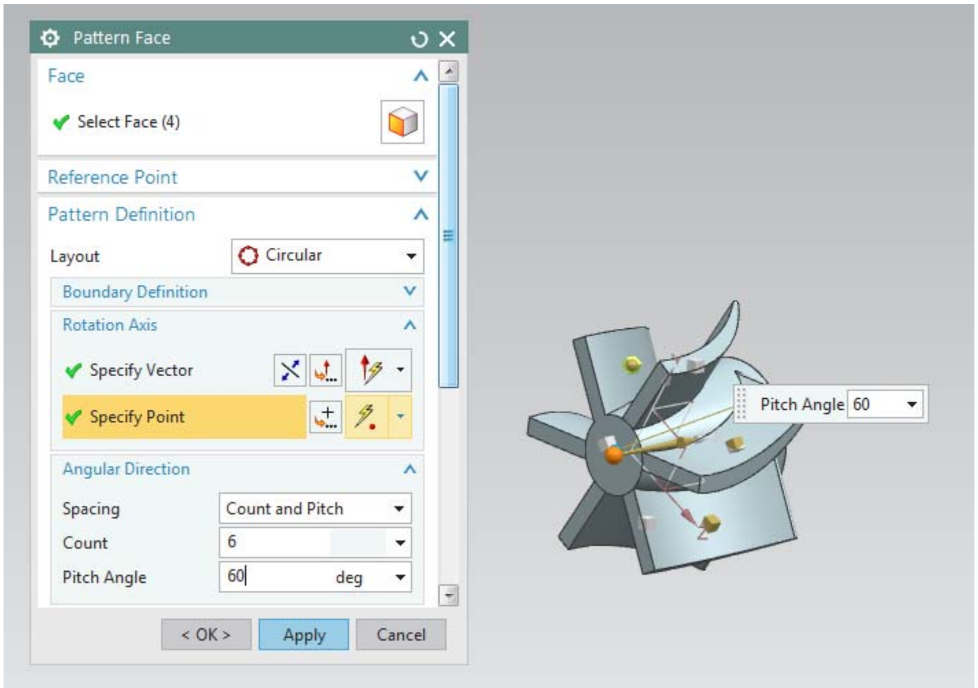
The part should look as shown below.



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



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

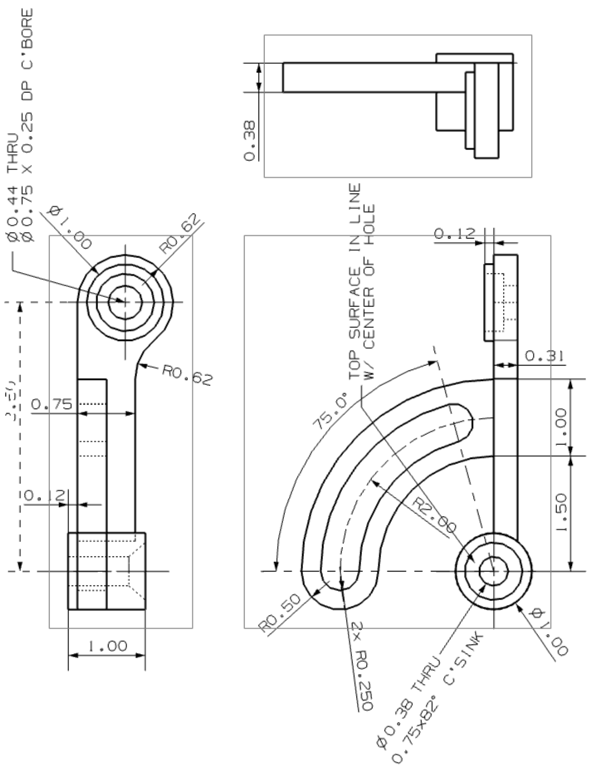


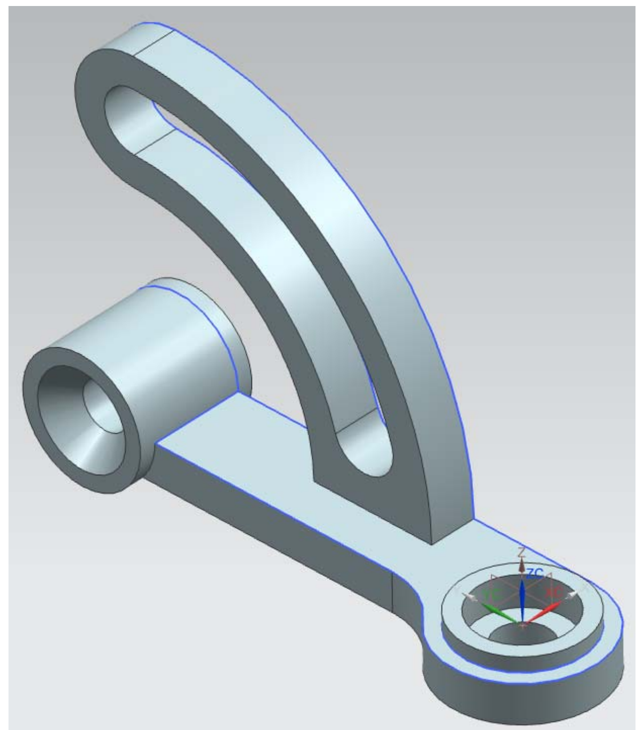
Therefore, it can be observed that a standard .stp file has been modified by increasing the number of blades and changing the blade profile. Similarly, the user can either modify any supported 3D model depending on the design need or create a new 3D model with synchronous modeling "push and pull" tools.

4.10 EXERCISES

4.10.1 Rocker Arm

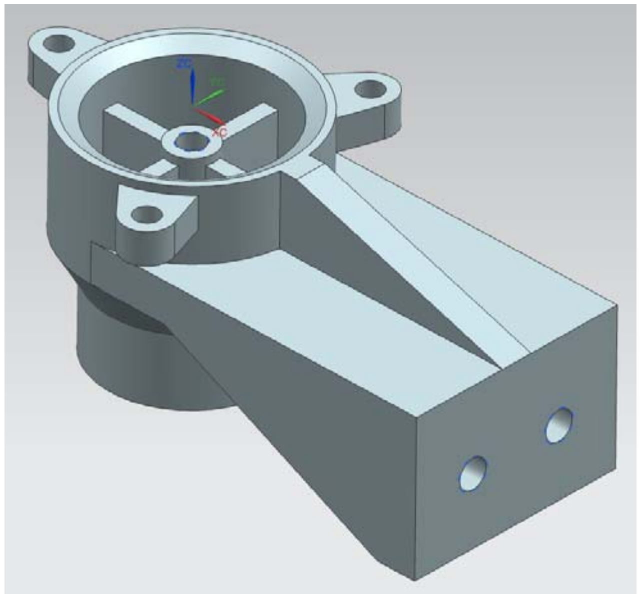
A rocker arm is an oscillating lever conveying radial movement from the cam lobe to linear movement at the poppet valve. Use the following drawings to model a rocker arm (dimensions are in inches).

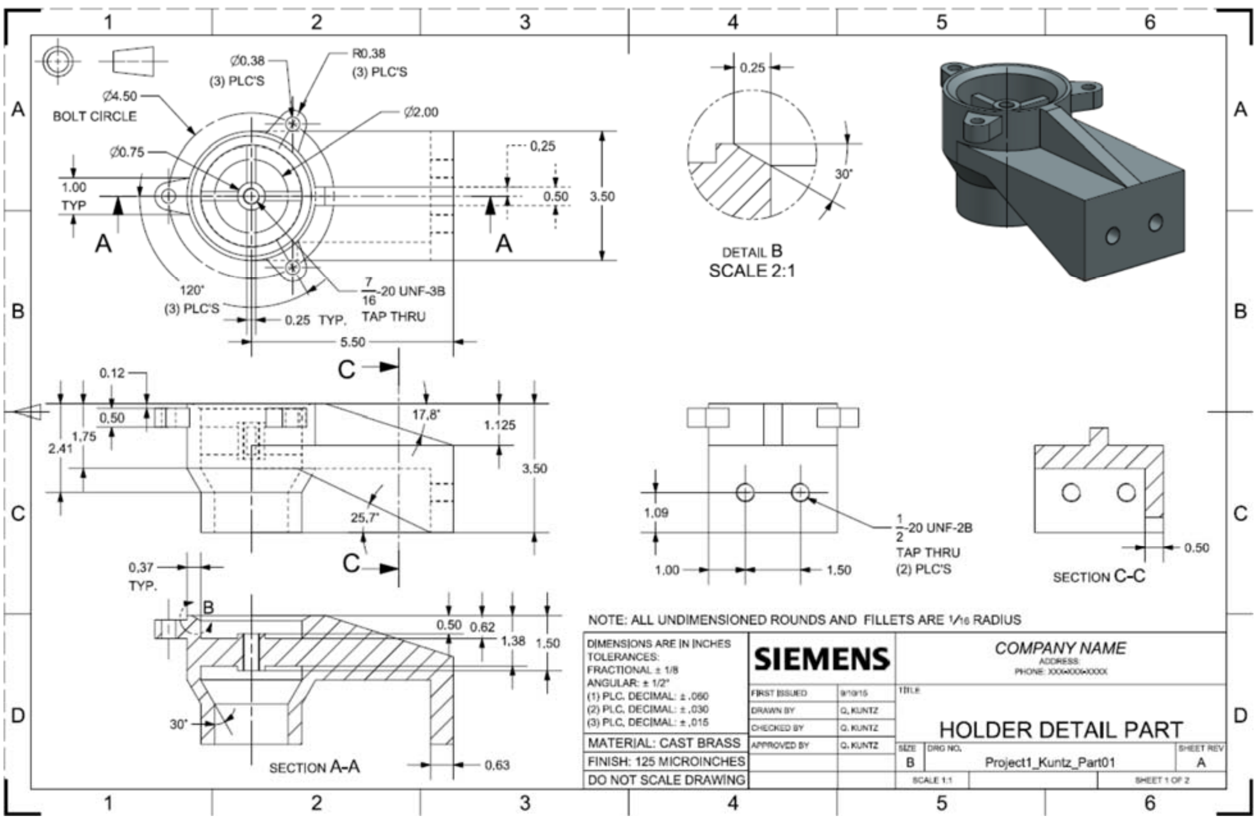




4.10.2 Holder

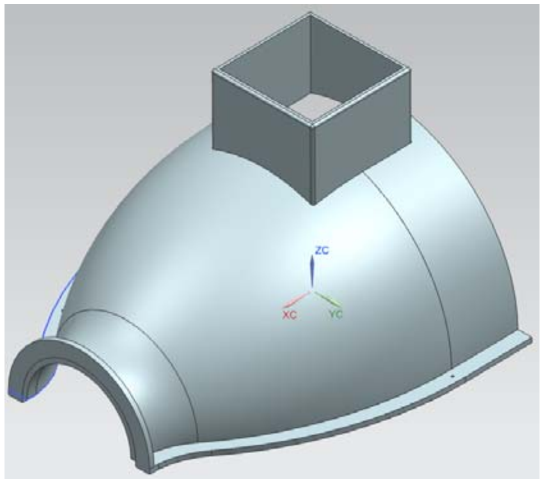
You have started to model a holder in chapter 3 by modeling a 2D sketching. Open your saved file and continue modeling this holder. Use the detailed drawing to replicate the part.





4.10.3 Impeller Upper Casing

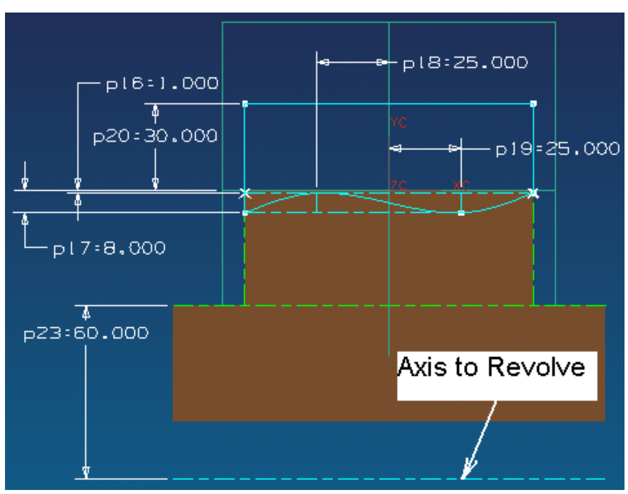
Model the upper casing of the *Impeller* as shown below. The dimensions of the upper casing are the same as for the lower casing, which is described in Section 4.4 in detail. The dimensions for the manhole should be such that impeller blades can be seen and a hand can fit inside to clean the impeller.



4.10.4 Die-Cavity

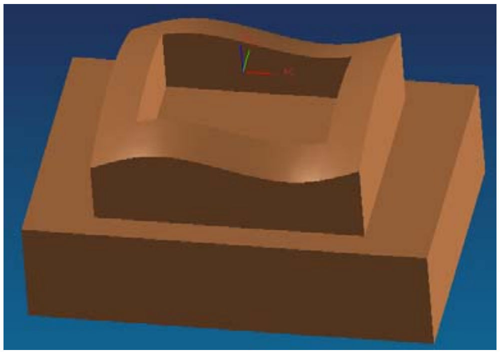
Model the following part to be used for the *Chapter 8 Manufacturing Module*. Create a new file *Die_cavity.prt* with **units in *mm***. Create a rectangular *Block* of *150, 100, 40* along *X, Y* and *Z,* respectively with the point construction value of *(-75,-50,-80)* about *XC, YC and ZC*. Create 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.

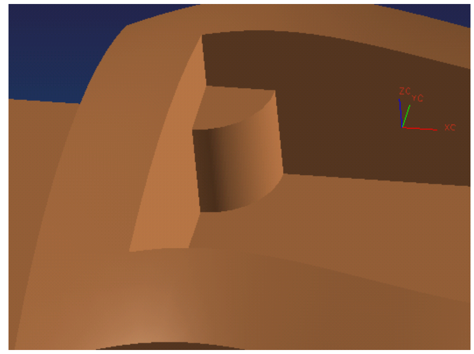
Create a sketch as shown below including the spline curve and add an Axis line. Dotted lines are reference lines. While sketching, create them as normal curves. Then right click on the curves and click convert to reference. Give all the constraints and dimensions as shown in the figure below.



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.