

CHAPTER 3 – TWO DIMENSIONAL SKETCHING

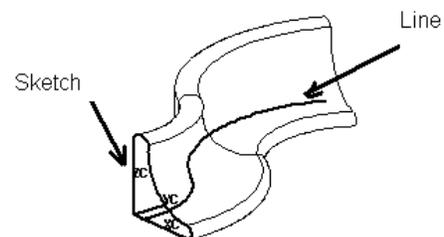
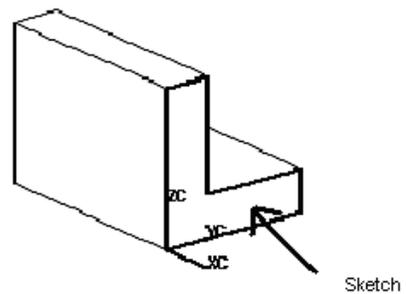
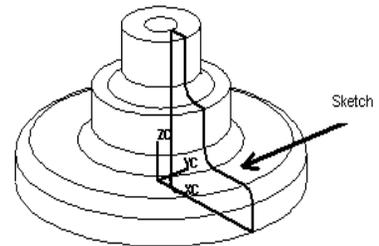
In this chapter, you will learn how to create and edit sketches in NX 12. You can directly create a sketch on a *Plane* in *Modeling* application. In most cases, *Modeling* starts from a 2D sketch and then *Extrude*, *Revolve* or *Sweep* the sketch to create solids. Many complex shapes that are otherwise 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 12 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 12 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**

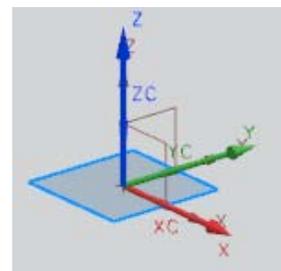
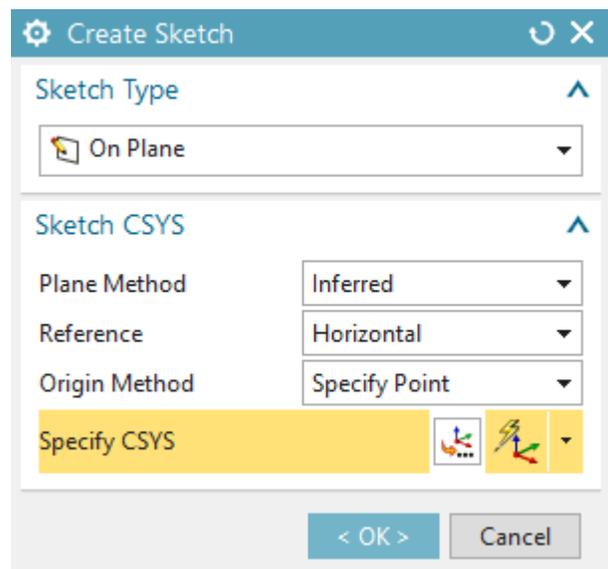
In the other method you can create Sketch using

- 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.

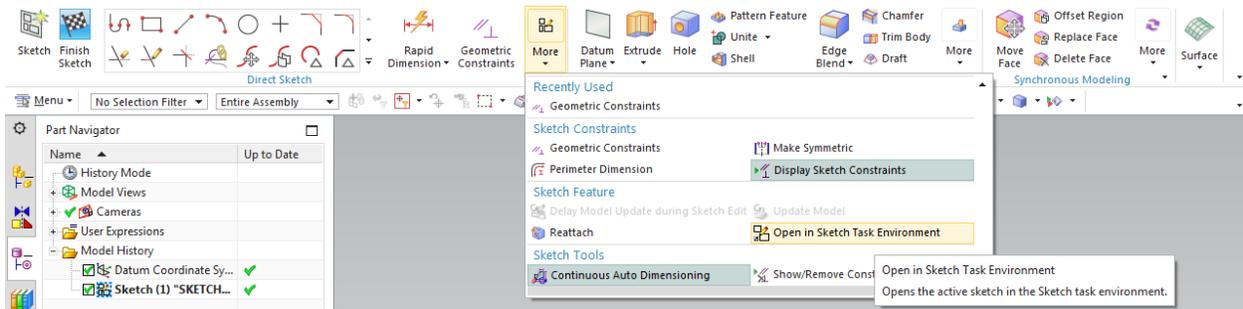
- Choose the **XC-YC** plane and click **OK**

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 12 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



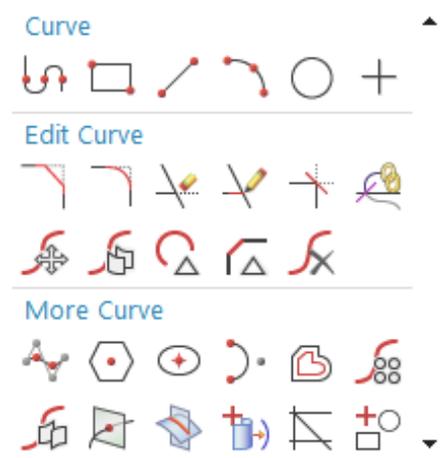
There are three useful options next to the *Finish Flag*. You can change the name of the sketch in the box. The next one is *Orient to Sketch* which orients the view to the plane of the sketch. If the model file is rotated during the process of sketching, click on this icon to view the sketch on a plane 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.

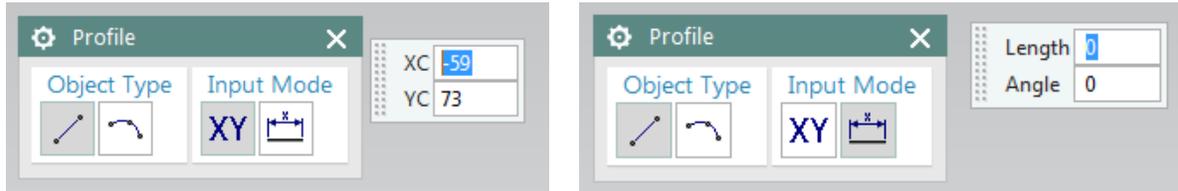
3.3 SKETCH CURVE TOOLBAR

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.

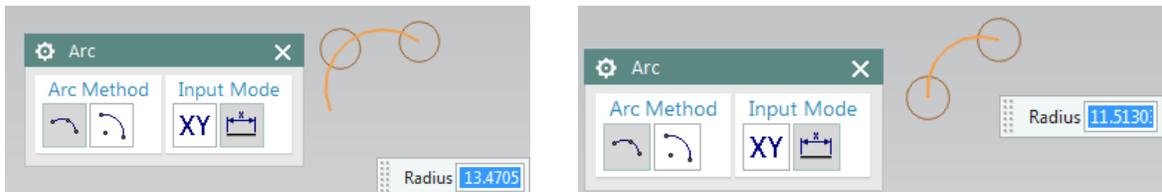


Line

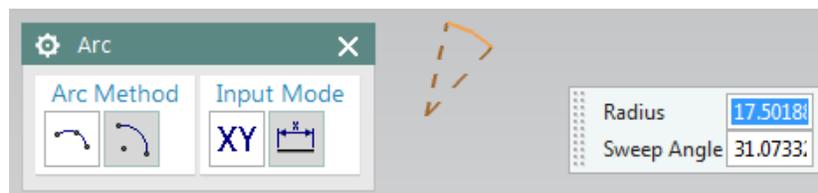
This option will selectively create only straight lines.

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.



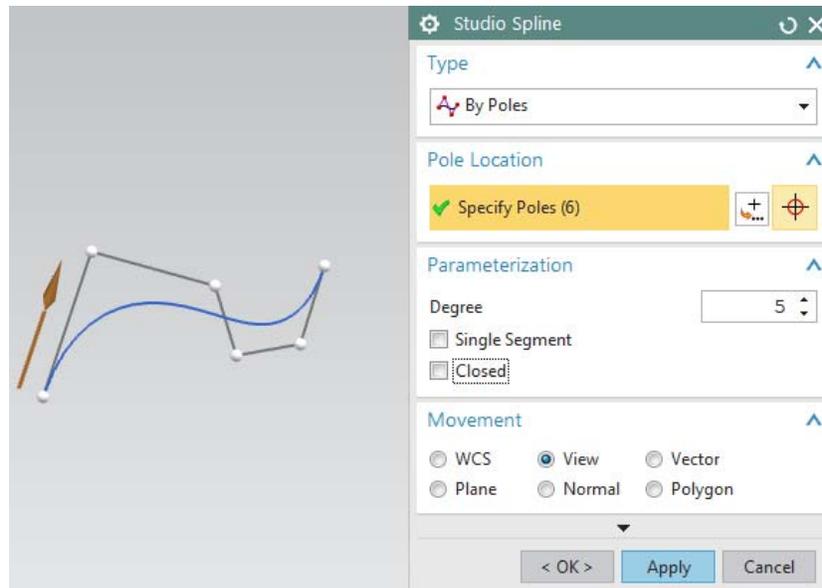
Quick Trim

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

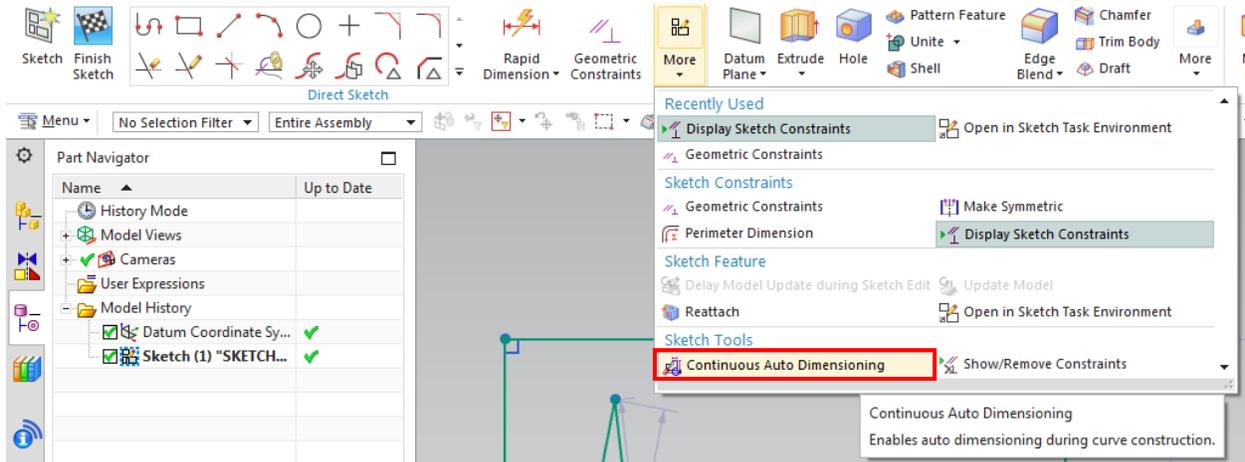
You can create basic spline curves (B-spline and Bezier) with poles or through points with the desired degree of the curve. The spline will be discussed in detail in the seventh chapter (Freeform Features).



3.4 CONSTRAINTS TOOLBAR

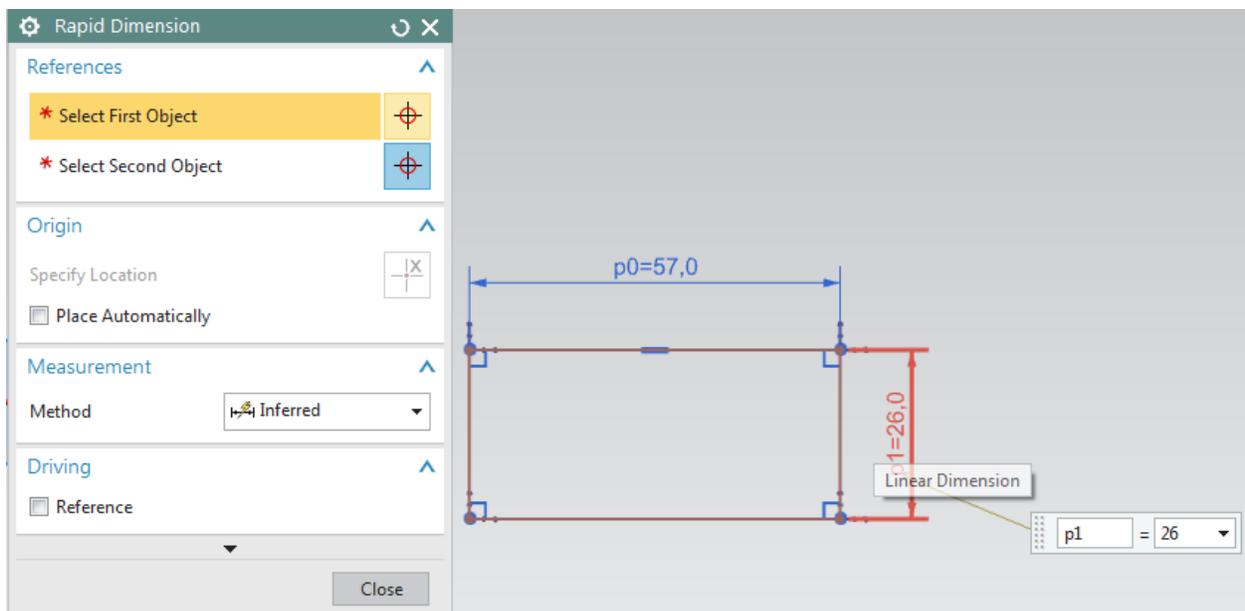
All the curves are created by picking points. For example, a straight line is created with two points. In a 2D environment, any point has two degrees of freedom, one along X and another along Y axis. The number of points depends on the type of curve being created. Therefore, a curve entity has twice 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 12 smart constraints are applied automatically, i.e. automatic dimensions or geometrical constraints are interpreted by NX 12. You can turn this option off by clicking on *Continuous Auto Dimensioning* as shown below. The following paragraphs show how to manually apply constraints.



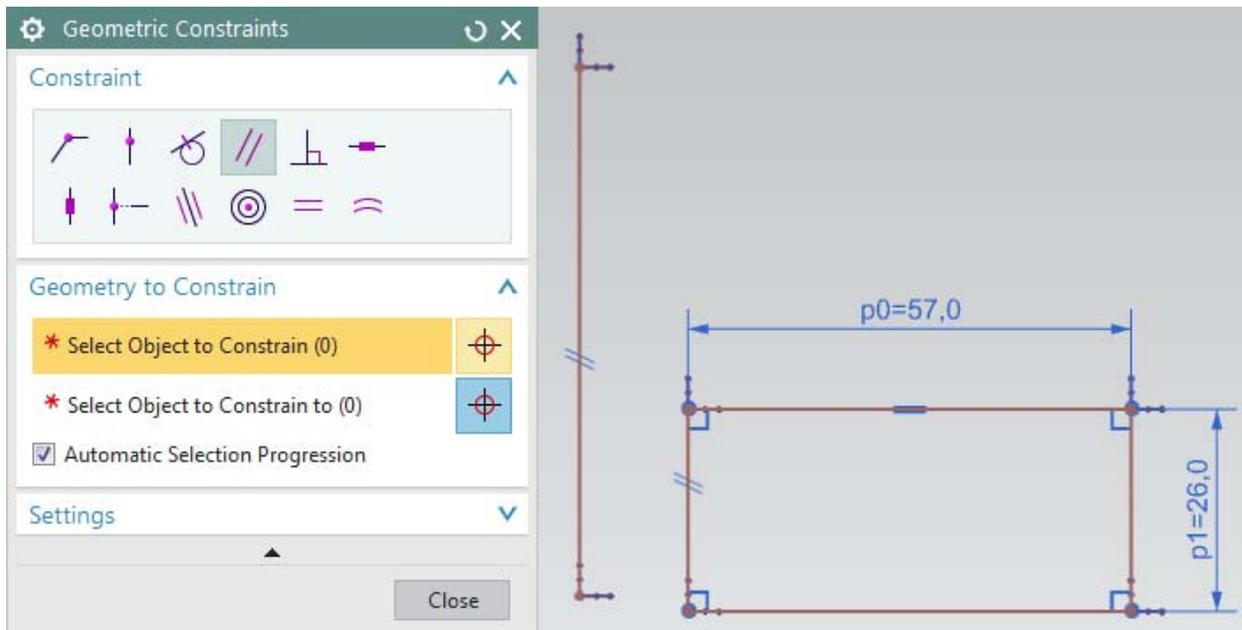
Dimensional Constraints

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

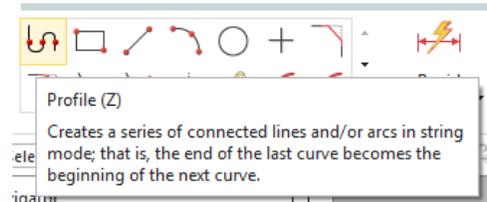
This window lists all the constraints and types of constraints pertaining to any entity selected. You can delete any of the listed constraints or change the sequence of the constraints.

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.

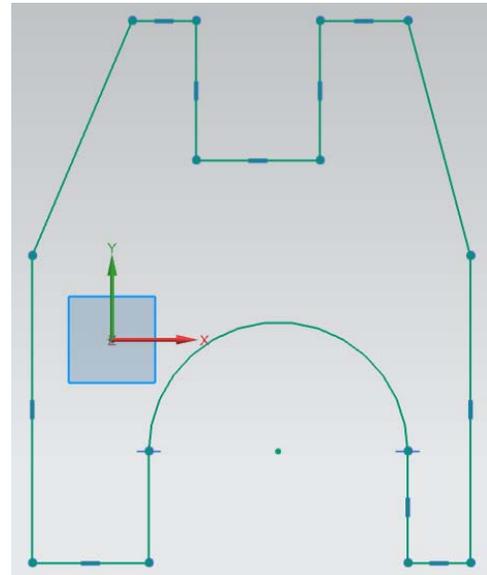
3.5 EXAMPLES

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.

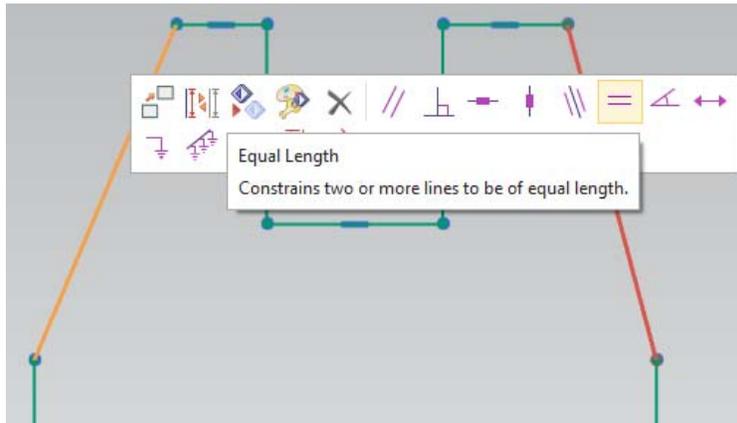
- Choose **Insert** → **Geometric Constraints** or click on the **Constraints** icon in the side toolbar 

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.

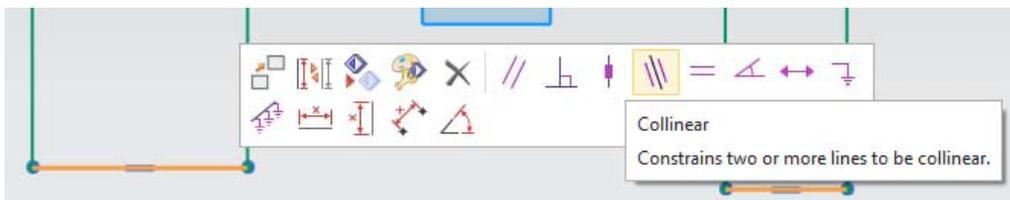
- Select **Point on Curve**  constraint
- Select the Y-axis and then the center of the arc
- Repeat the same procedure to place the center of the arc on the X-axis

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.

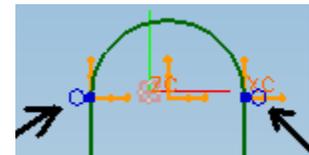
- Select the two slanted lines and make them **Equal Length**



- 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.



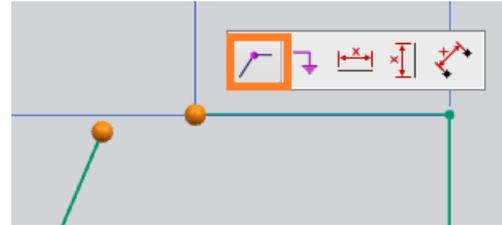
- Select the circular arc and one of the two vertical lines connected to its endpoints
- Select the **Tangent** icon 

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.

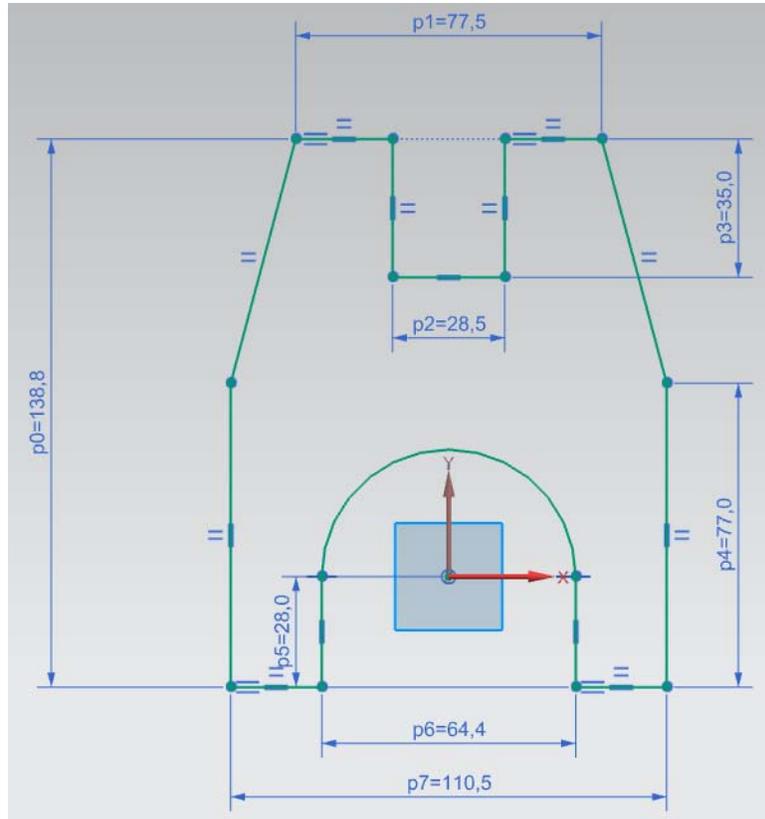
- Select the two vertical lines and make them **Equal**
- Similarly select the two small horizontal lines at the top of the profile and make them **Collinear** and **Equal**
- Similarly select the two vertical lines and make them **Equal**

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



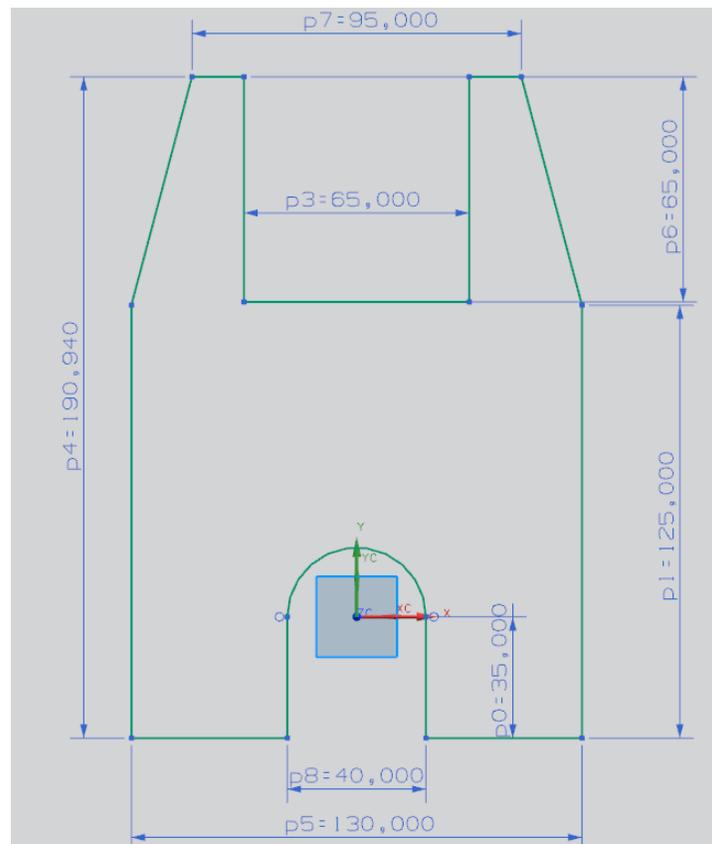
For example, to create a dimension for the top two corners,

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

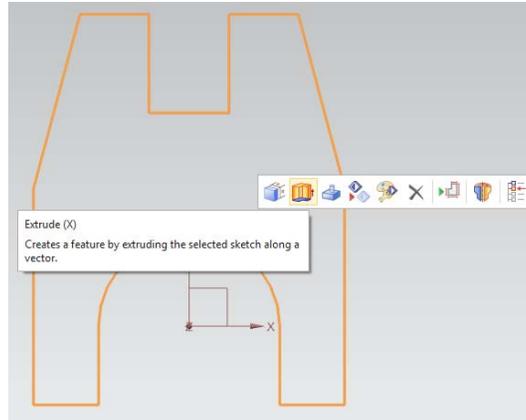
While dimensioning, if you find the dimensions illegible, but do not worry about editing the dimensions now.

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.

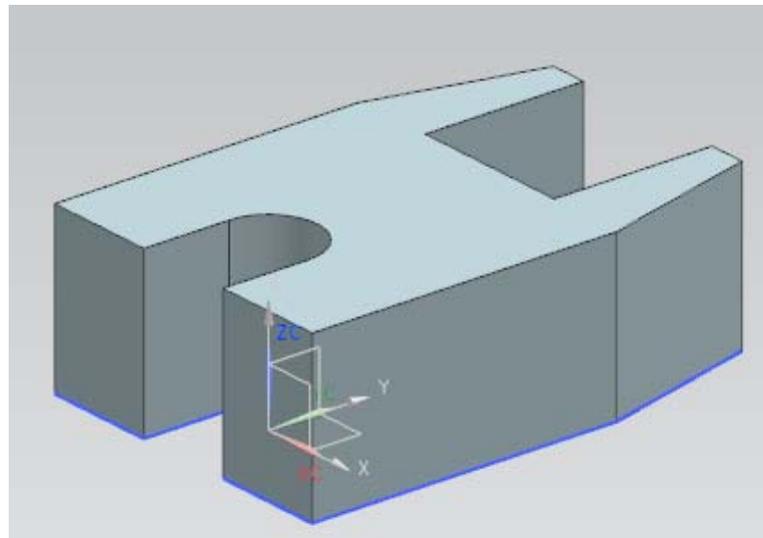
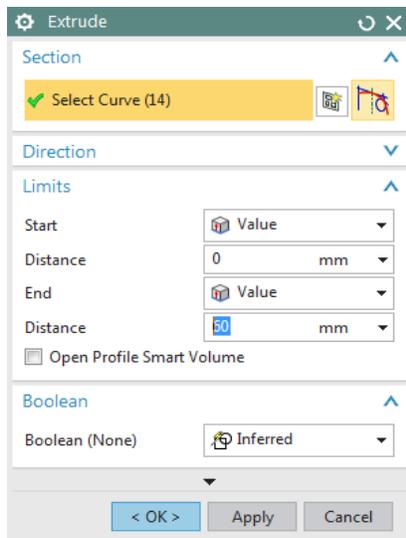
- Edit the values as shown in the figure below. Double click on each dimension to change the values to the values as shown in figure below



- 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)



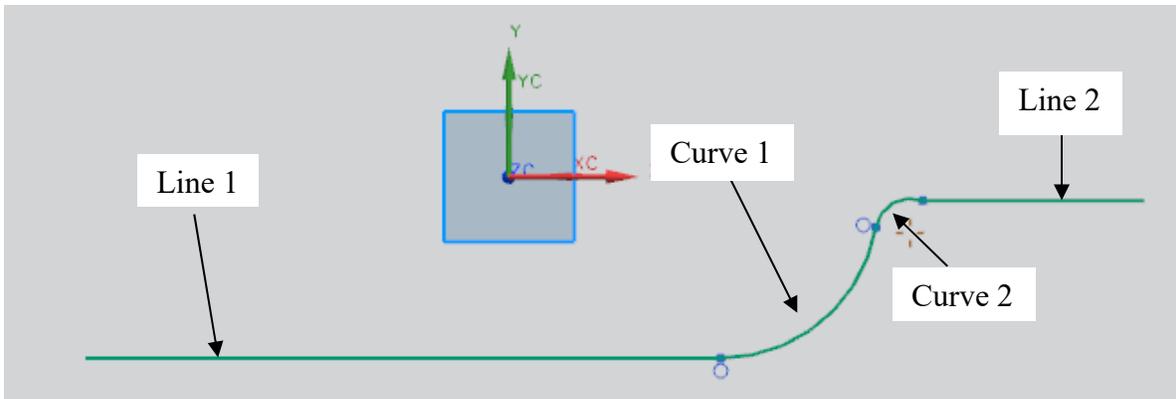
- Extrude this sketch in the Z-direction by **60 mm**



- Save and close the file

3.5.2 Impeller Lower Casing

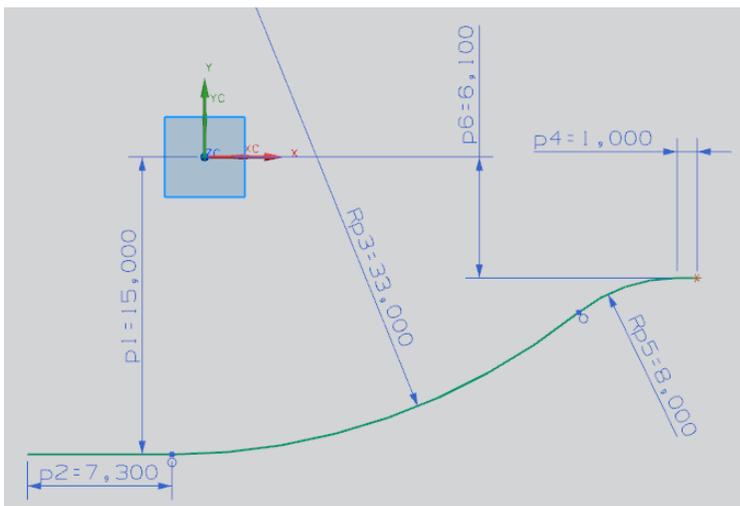
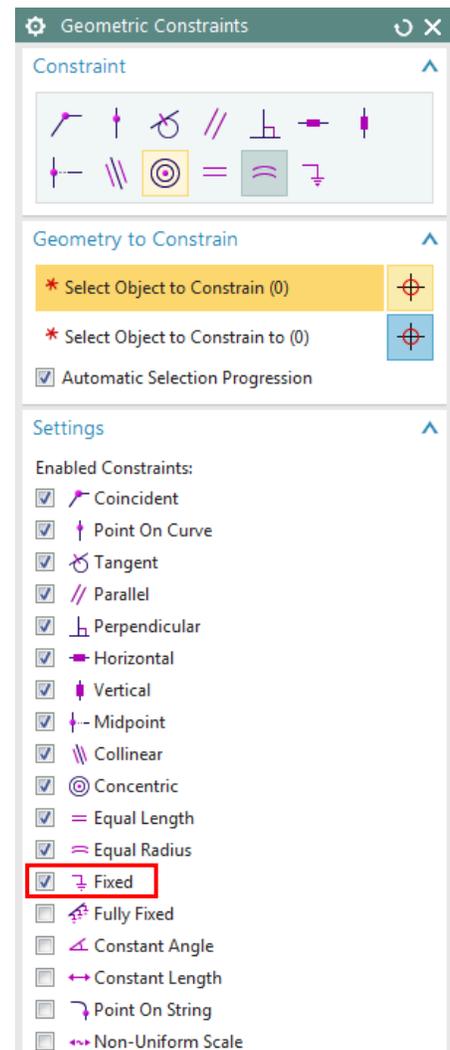
- Create a new file in **inches** and save it as **Impeller_lower_casing.prt**
- Click **Menu** → **Insert** → **Sketch In Task Environment** or click **Sketch In Task Environment** icon  from the ribbon bar
- Set the sketching plane as the **XC-YC** plane
- Make sure the **Profile** window is showing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



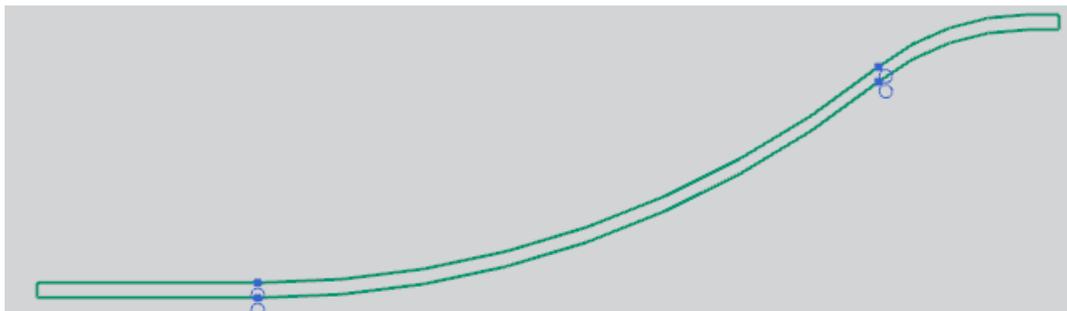
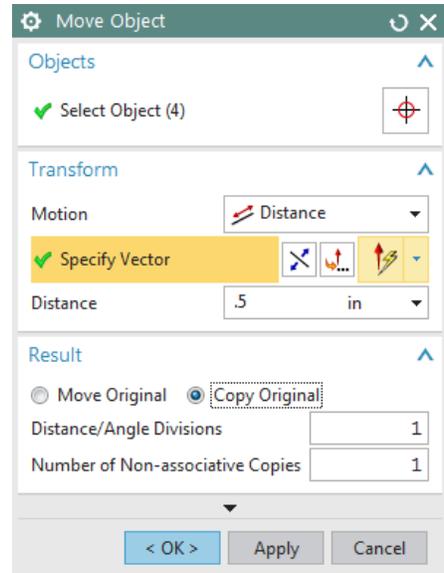
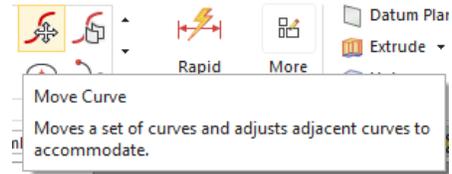
- Create a point at the origin (0, 0, 0) by clicking the plus sign in the **Direct Sketch** group

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 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



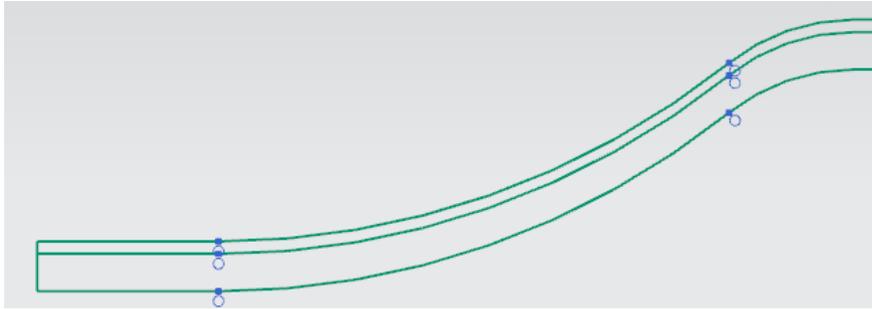
The sketch is ready.

- 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.

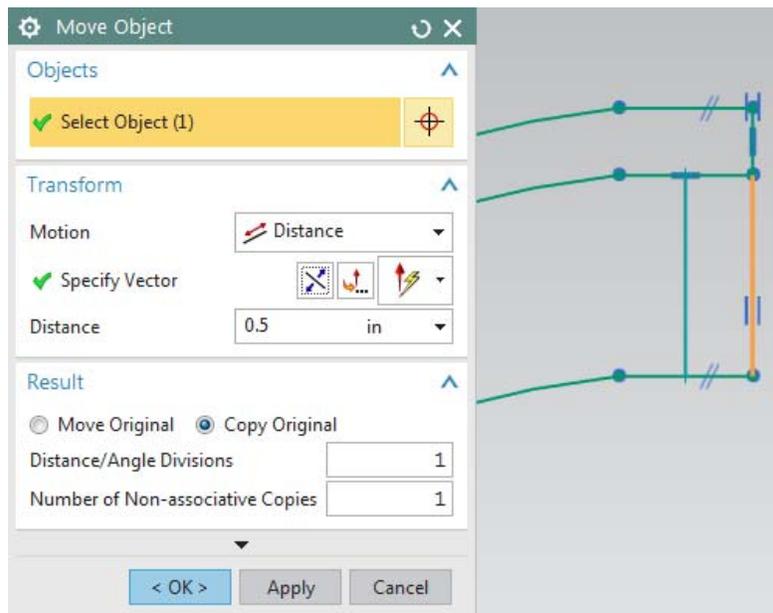
- Using **straight lines** join this curve with the inside curve of the casing

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

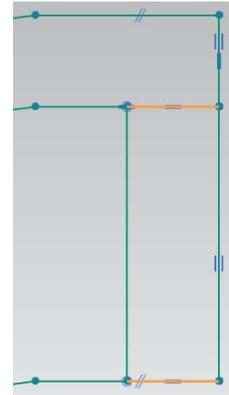


- **Move** the lower curve in the **XC**-direction by **-0.5** inches. This is the same as translating it in the **negative XC-direction** by **0.5** inches

- 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.

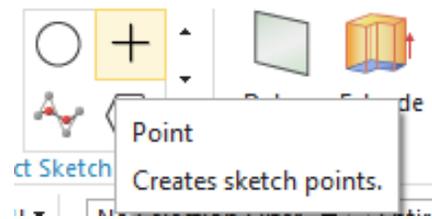
3.5.3 Impeller

- 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)**

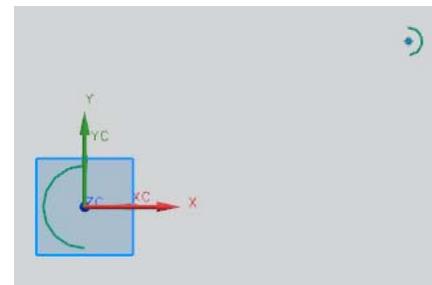
- Click on the **Arc** icon on the side toolbar and click on the **Arc by Center and Endpoints**

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

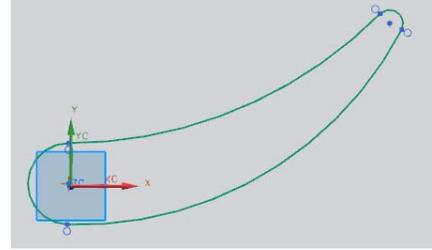
- Click on the point at (11.75, 6, 0) and create an arc with a radius of **0.5**

- 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

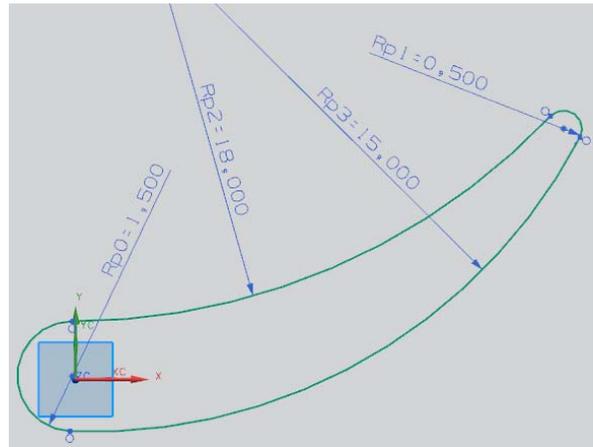
- Click on the **Constraints** icon in the side toolbar and make sure that all the arcs are **Tangent** to one another at their endpoints



- 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** 

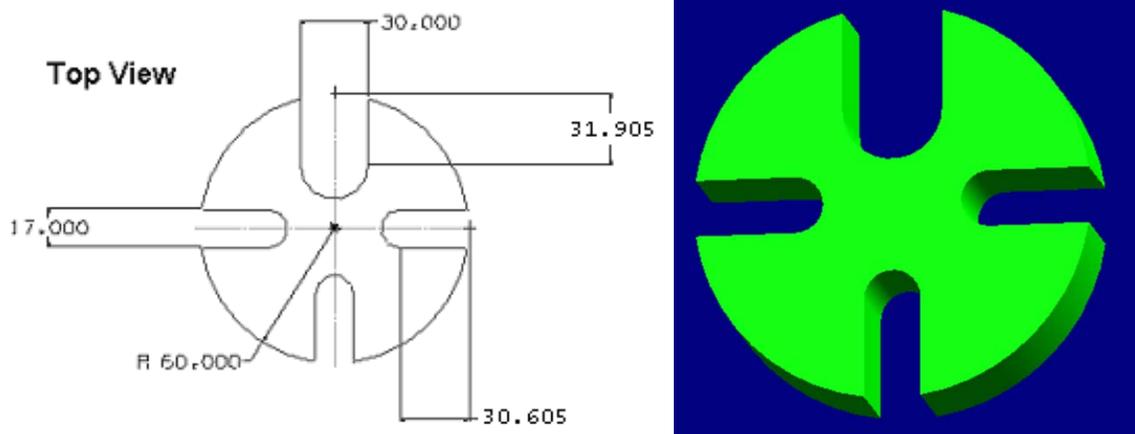
- Save and Close the file.

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

3.6 EXERCISES

3.6.1 Circular Base

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



3.6.2 Sketching of a Holder

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

