
Profiles and Constraints

A variation profile is a set of plane curves and lines to which you can assign geometrical and dimensional constraints and which you can use to create solids or to add features to existing solids. Variation profiles are the basic components of ThinkDesign's parametric structure. You can use profiles to create solids and features by applying a linear sweep or revolving the profile. In the case of a feature, the profile is used either to add material to or remove material from a solid. Profiles can be appropriately constrained and when used to create a solid or feature, the dimensional constraints assigned to the profile become the parametric dimensions of the resulting solid or feature. By changing the values of the profile's constraints. You can therefore change the dimensions of the solid or feature. Note that constraints can be assigned to a profile either before you use it to create a solid or feature or after you have created the solid or feature. ThinkDesign also provides commands for automatically applying constraints. In ThinkDesign, 3D models can be built using different approaches. They are either Mechanical or Artistic. Designer's intent and the different steps to build the models dictate these approaches. In this short task, we will study these different approaches. The Mechanical approach emphasizes the 3D model being completely constrained or parametric. The Artistic approach is interested in free form shape, which later can be constrained. Let's see how these two approaches differ and how being aware of these differences matter.

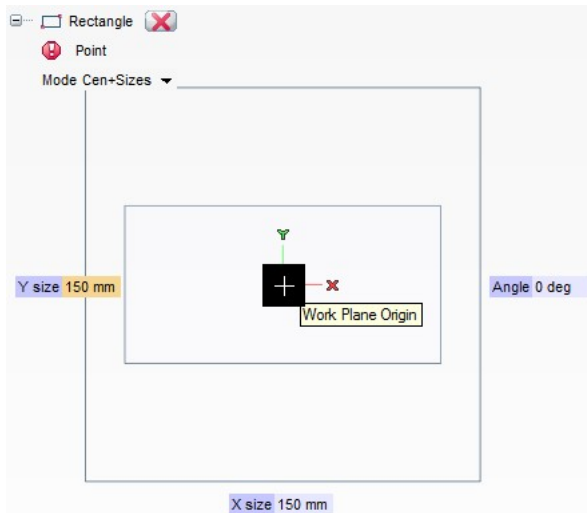
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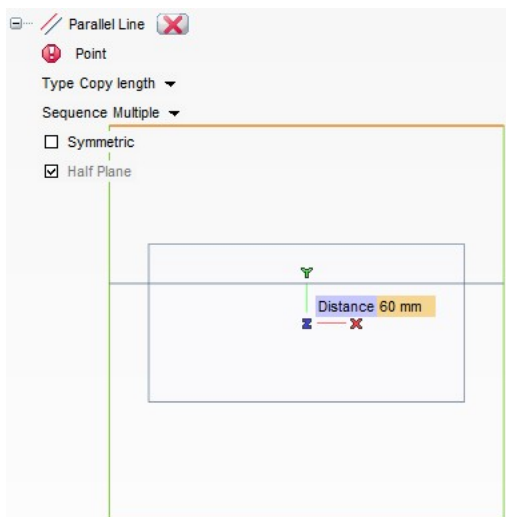
1. Step 1: Draw the basic shape curves

- Open a **New Model**.
- Right click in the graphics area and select **Options/Properties**
- Set the Measure units to Millimeter under Document Properties tab.

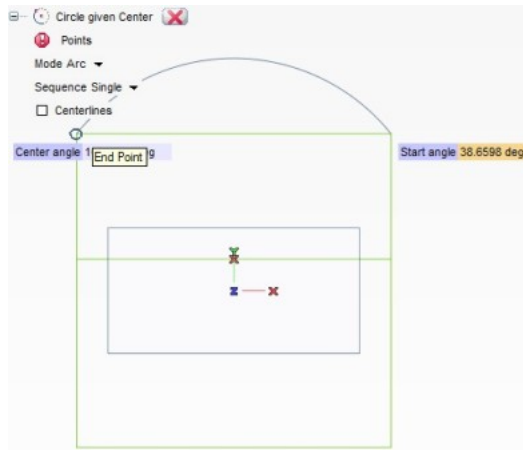
- Start the **Rectangle** command.
- Set the Mode to Cen+Sizes.
- Set X and Y values to 150 and snap the rectangle to the **Work Plane Origin**.



- Start the **Parallel Line** command.
- Type :Copy length and check for Half Plane option.
- Select one of the edges as shown below.
- Set the distance as 60.

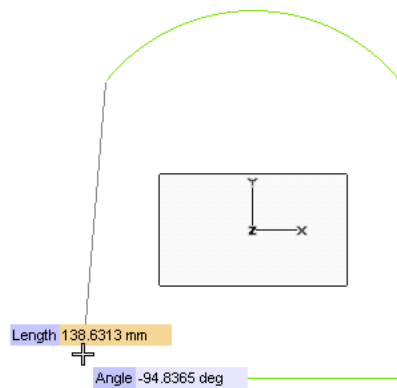


- Activate the **Center Circle** command.
- Change the Mode to Arc.
- Using the **Mid Point Snap**, select the mid point of the parallel line for the arc center.
- Select two corners of the rectangle as start and end points.
- Hit **Esc**.



- **Delete** the top horizontal and parallel lines.

This geometry has 4 single entities. We can modify the dimensions of any one of them. For example: Double click near the intersection point of the left vertical line and the bottom horizontal line and drag it. You can insert new values for the length and the angle parameters in their respective mini dialogs. But this modification will not be reflected on the entire shape.

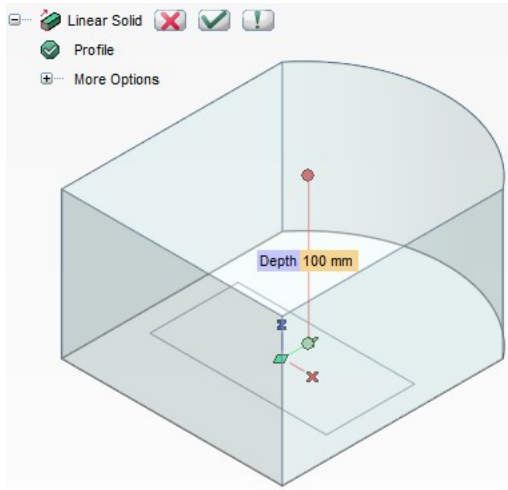


Press **Ctrl+Z** to Undo the last step.

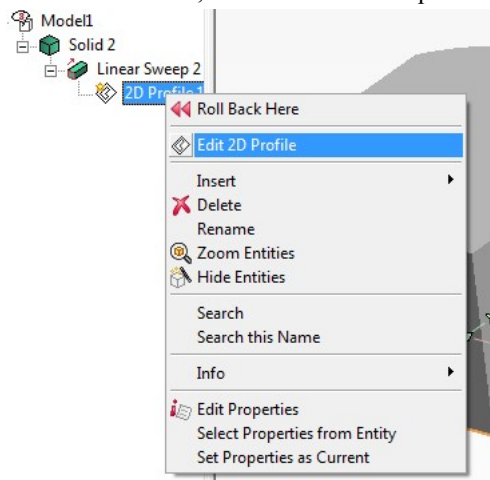
2. Step 2: Solid from curves

Select all the 4 curves.

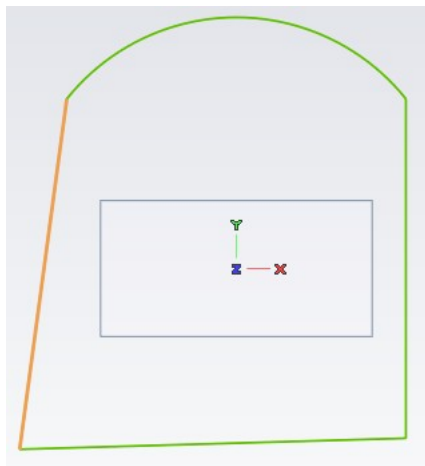
- Start the **Linear Solid** command.
- Set the Depth to 100.
- Click OK.



When the solid was created from the curves these are automatically turned into a profile. However we can modify only the depth now. We should be able to control the shape and the size of the base profile, with dimensions. In order to do this, we have to edit the profile to insert some constraints.



- Hit **F8** key to see the view in Plan view and use **Fit View** command to place the profile to middle of the screen.
- Change the display to **Wireframe View**.
- Double click on the bottom left corner and drag it. The entire shape of the profile changes.



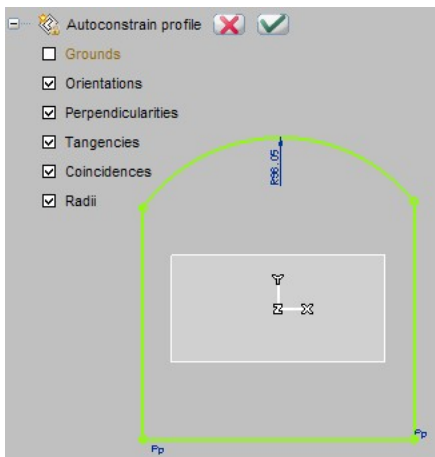
- Hit **Esc** to retain the profile to the original shape.

We can see that all the curves are linked and behave like an elastic one. But all the points are free to move in the desired direction. This happens as there are no Geometrical constraints attached to the profile. Let us see how can do this!!

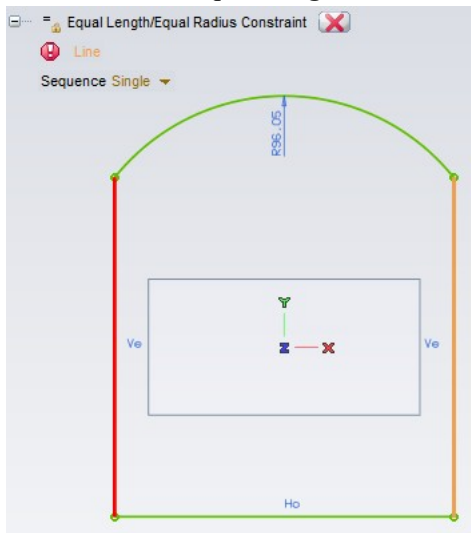
3. Step 3: Adding constraints

Let's constraint the profile now.

- Select **Insert** → **Profile** → **AutoConstraint**, it inserts some implicit geometrical constraints.

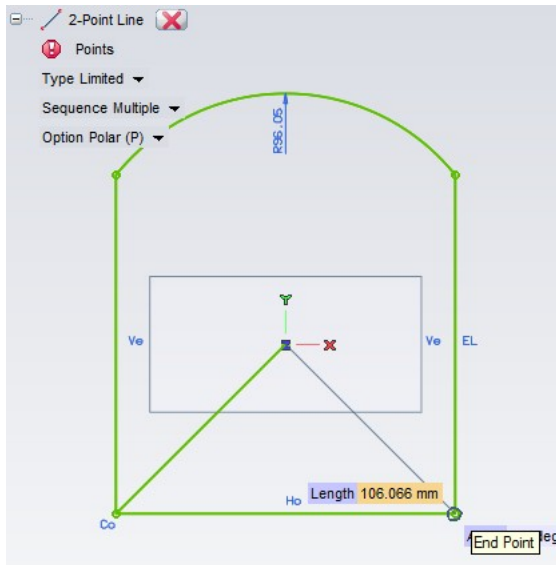


- Now add the **Equal Length/Radius Constraint** selecting the two verticle lines as shown.

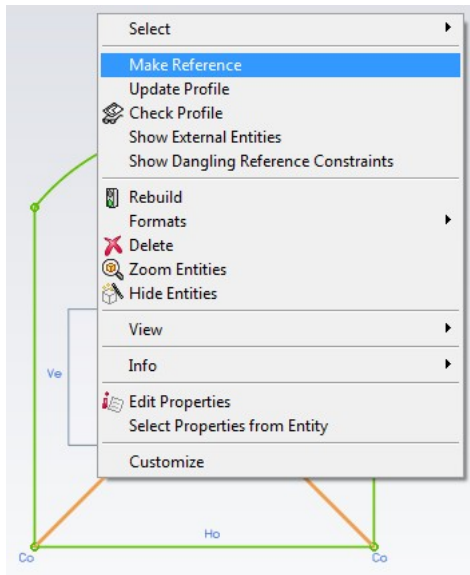


Now we want the length and the height to be symmetrical about **Work Plane Origin**.

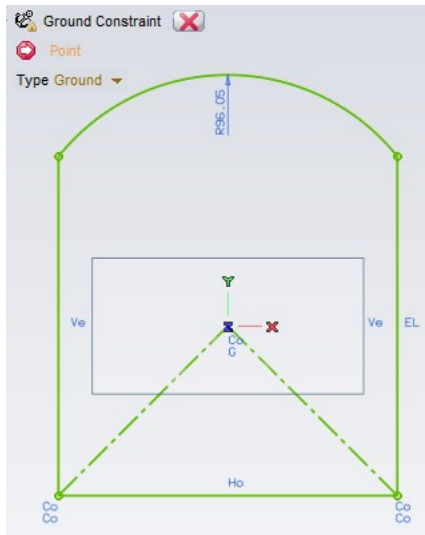
- Start the **Two-point Line** command.
- Select the bottom left corner of the profile and the **Work Plane Origin**.
- Similarly draw one more line from **Work Plane Origin** to bottom right corner, as shown below.



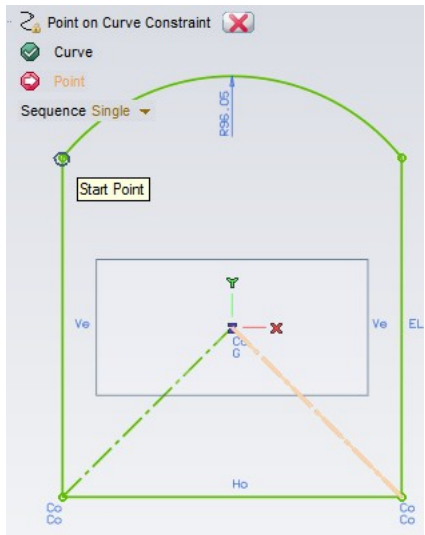
- Right click on these two curves and say Make Reference.



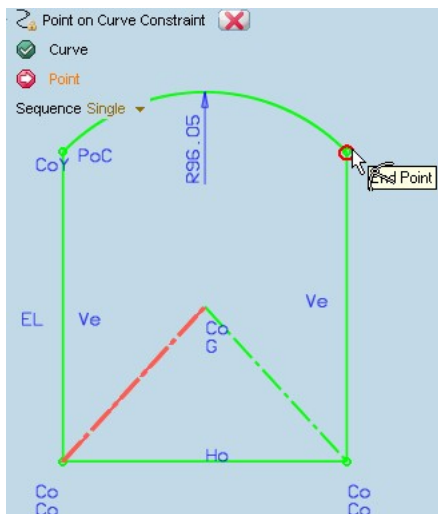
- Insert the **Coincident Constraint** to the two end points of the reference lines at **Work Plane Origin**.
- And also add the **Ground Constraint** to one of the end points of the reference lines.



- Now insert the **Point On Curve Constraint** by selecting the left corner point and right reference line.

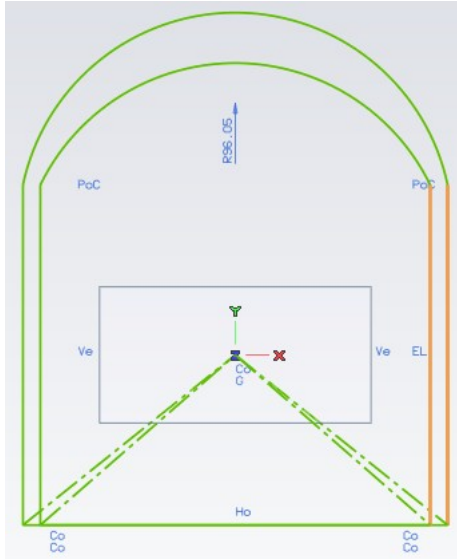


- And insert the **Point On Curve Constraint** by selecting the right corner point and left reference line.



Let's try if the constraint controls the shape correctly. Double click on the right vertical line and drag it. Remember to Press **[Esc]** in order to return to the previous shape. Try to drag the bottom horizontal line. You will not

be able to change the shape or the size of the profile.

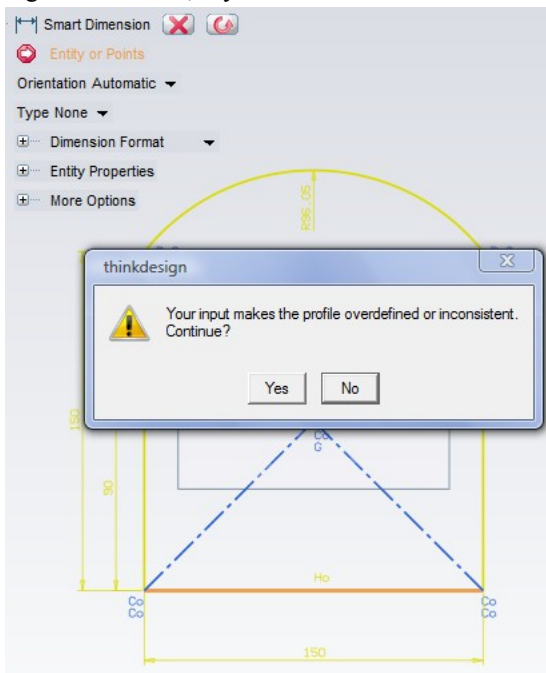


Good! this is what we needed.

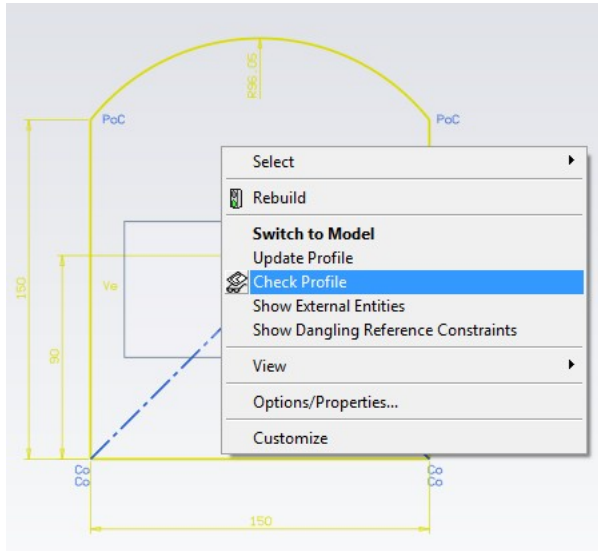
Now, let's add some dimensions to the profile to control the shape.

- Activate **Smart Dimension** and dimension it as shown.

While adding dimension 90, by selecting the bottom Horizontal line and the arc center on top, we may get warning-Over defined, say Yes.



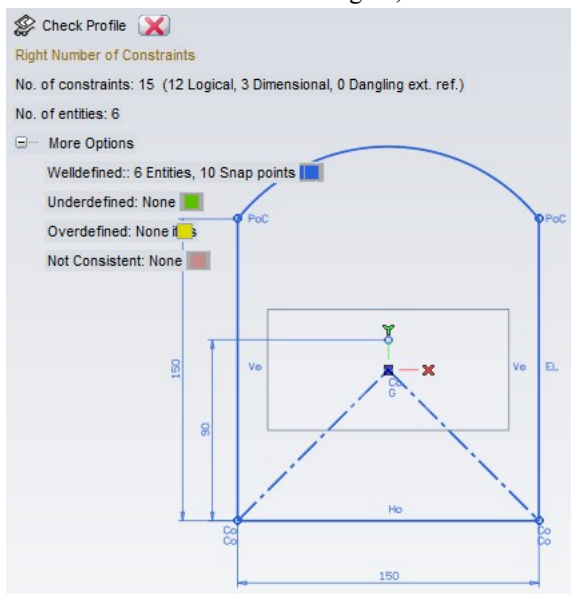
- Right click in the graphics area and say Check Profile.



The Selection list shows that the profile is Over defined and it also highlights the entities that are Overdefined, Underdefined and Welldefined.

- Cancel the Command.
- **Delete** the Dimension R96.05.
- **Rebuild Model** .

Start **Check Profile** command again, the Selection list shows that the profile is right constrained.

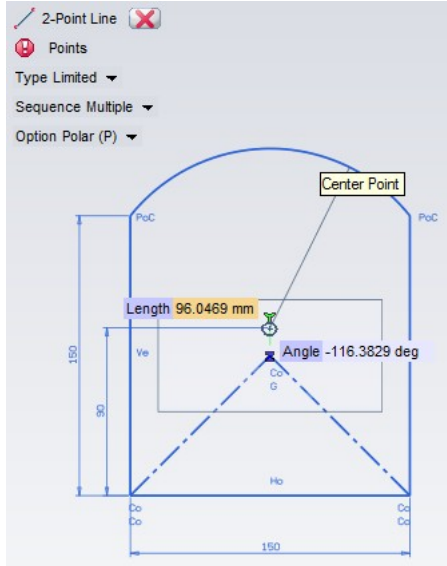


Remember that, in ThinkDesign it is not mandatory to create a fully constrained profile because we can create a solid from a partially constrained profile.

4. Step 4: Controlling the work plane position

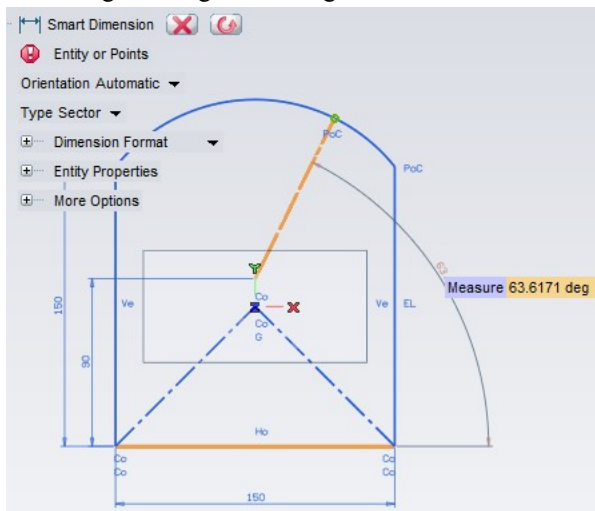
Now we have to create a KeyHouse, tangent to the circular surface but whose position is controlled by an angle. First we must position the work plane, such that it reflects the conditions demanded. This work plane must be linked therefore to an entity. A work plane can be linked to entities with history so we will use a profile and not curves.


- Click 2D Profile.
- Select the **Two-point Line** command.
- Select start point on the arc using **Point on Curve Snap** and select the arc centre using **Snap to Arc Center** as the second point.





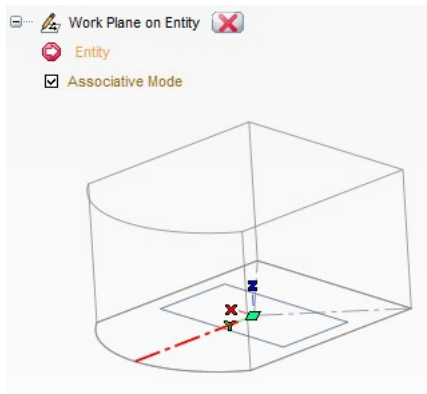
Two constraints are added automatically.

- Make this line as a Reference line.
- **Smart Dimension** the angle between the reference line and the bottom edge.
- Change the angle to 90 degrees.



Switch back to the modeling environment by clicking on  Model tab. This is the entity, we are going to link the work plane with.

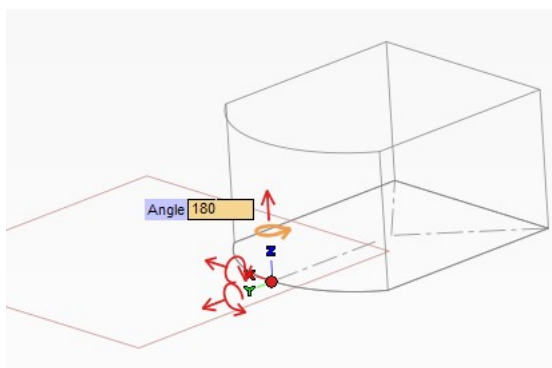
- Activate the command **Edit**  **Work Plane**  **On Curve**.
- Select the Reference line.

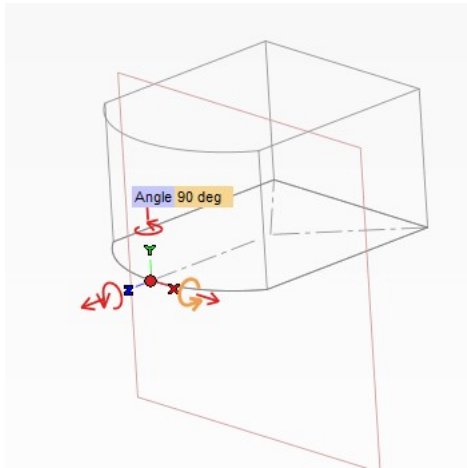


- And select the left side corner point as shown below.

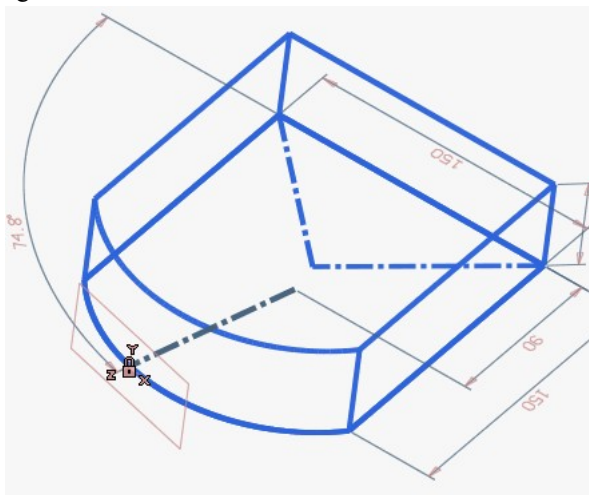


- Click on Work plane to Edit the Work Plane.
- Rotate the work plane by 180deg along Y axis and then rotate by 90deg on the X-axis.
- Hit OK.

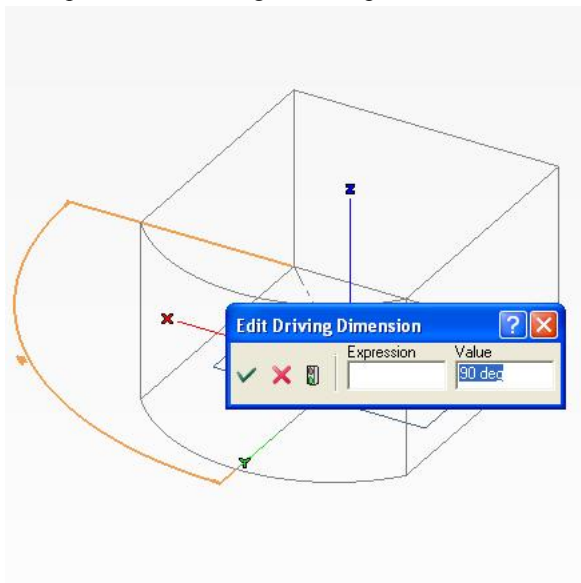




Now go to **Edit** → **Work Plane** → **Move** and select the end-point of the Vertical reference line as shown in the image below.

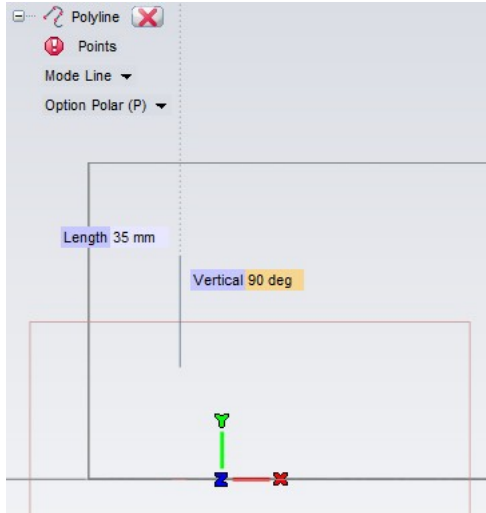


Let's try to modify the driven angle value. Double click on angle 74.8 and change it to 90. As you can see, the work plane moves along with the profile.

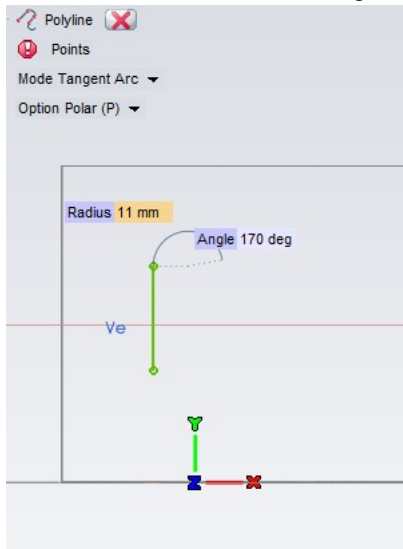


5. Step 5: Keyhouse in Profile mode

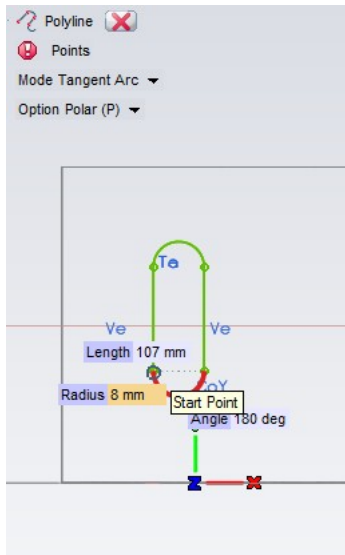
- Hit keys **F8** and **F** to have the work plane view.
- Click 2D Profile
- Select **Polyline** and draw a vertical line.



- Change the Mode to Tangent Arc.
- Whatever the radius, and keep the angle 180deg.

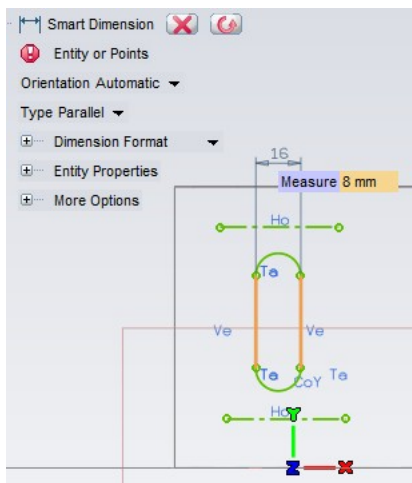


Continue with other vertical line and an tangent arc to close the profile.

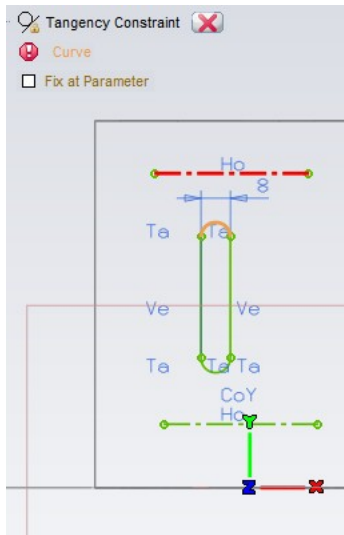


We must have 4 **Tangent Constraint**, if not we need to add it manually.

- Draw 2 **Two-point Line** one above and the other below the oblong profile.
- Select these 2 horizontal lines and Make Reference.
- Add **Smart Dimension** to the slot width (8mm).
- Now activate **Tangent Constraint** and select the upper horizontal line and the upper arc of the oblong profile.



- Similarly add **Tangent Constraint** to lower arc and line.
- The dimension should be 18mm between these 2 lines.

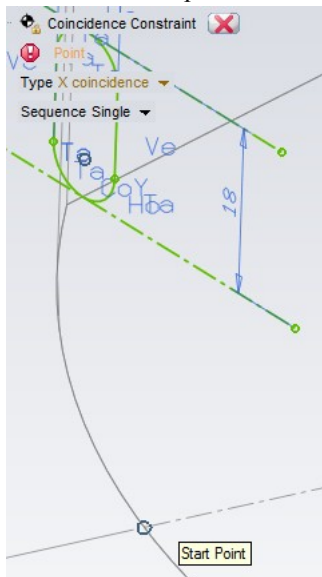


Now let's see how to constraint this profile to the external entity.

6. Step 6: Constraint to an external entity

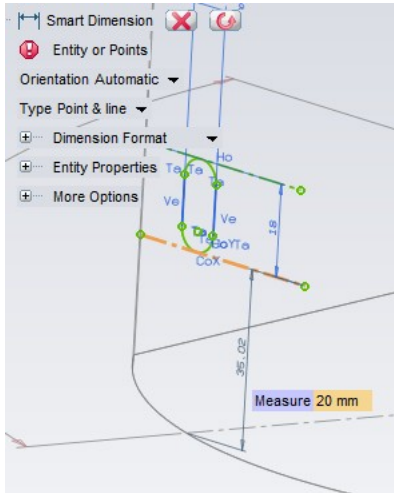
Let's fix the position of the profile on the XY plane. For this, we need to apply constraints to external entities.

- Rotate the view for clear visibility.
- Select **Coincident Constraint** and hit X Point button.
- Select the center point of the lower arc of the profile as first point using **Snap to Arc Center**.
- Pick the end point of the reference line as the second point using **End Point Snap**.

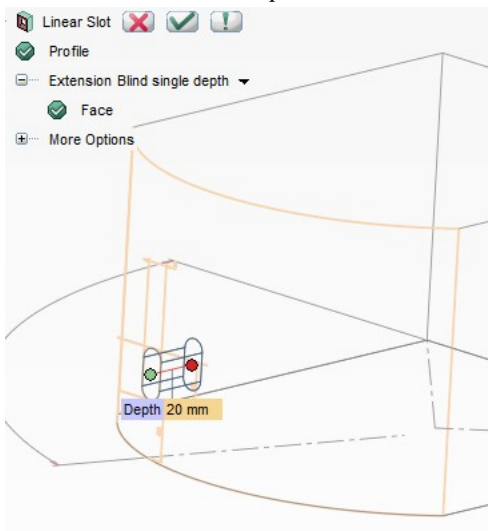


Now the profile can move only along Y-axis. You can cross check by selecting the profile and dragging. It moves only along Y-axis.

- Start the **Smart Dimension** command.
- The distance between the end point of the reference line to the horizontal line should be 20mm.



- Activate **Linear Slot**.
- Select the arc surface for Face.
- Set the Extension Depth to 20mm.



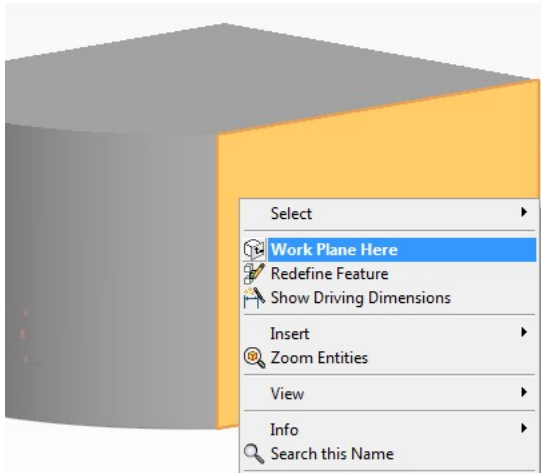
- Hit OK.

Now if you change the inclination of the work plane from the bottom edge to 110 or 120 deg, the position of the slot is also altered.

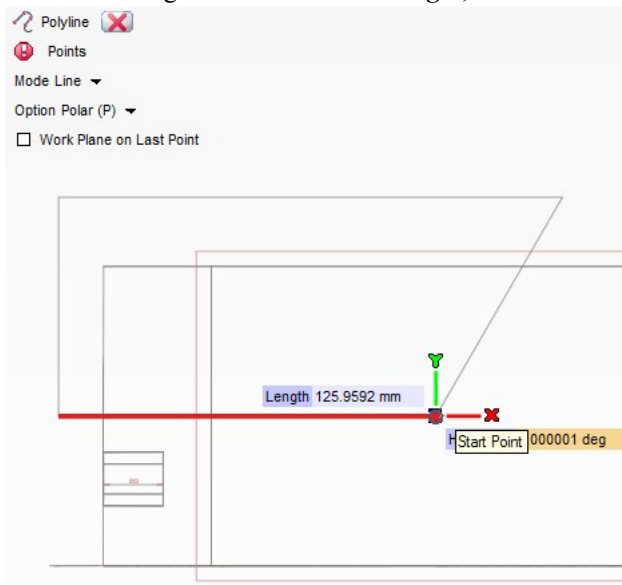
7. Step 7: Multi island profile

Now we are going to create multiple complex. A complex is a closed loop optionally containing multiple disjoint closed loops (island).

- Select **Shaded View**.
- Right click on the right face and say Work Plane Here.

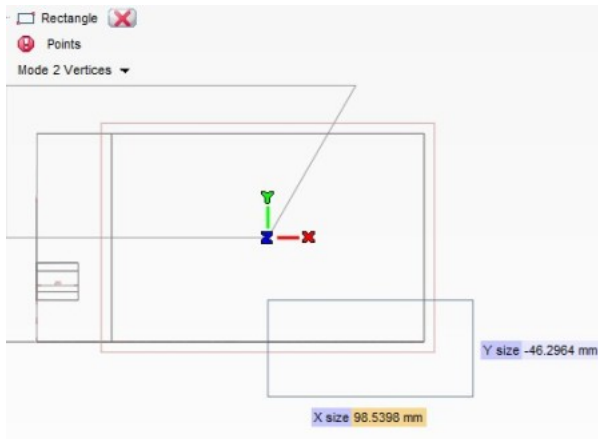


- Switch back to **Wireframe View**.
- Hit keys **F8** and **F** to have the work plane view.
- Start **Polyline**.
- And starting from **Work Plane Origin**, draw a line angled 60deg and close it as shown.



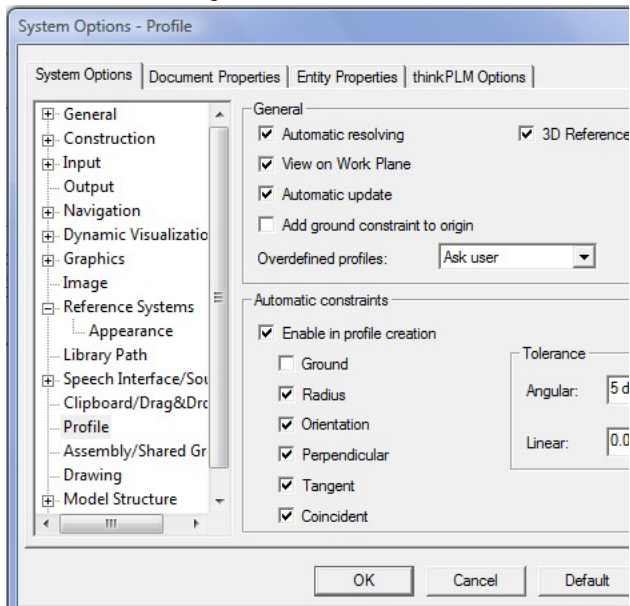
- Now activate the **Rectangle** command.
- Specify the first point of the rectangle using **Enable Point Coordinates**.
- Give X,Y,Z values as 0,-30,0.

Hit OK, and pick the diagonal end point of the rectangle outside the model.



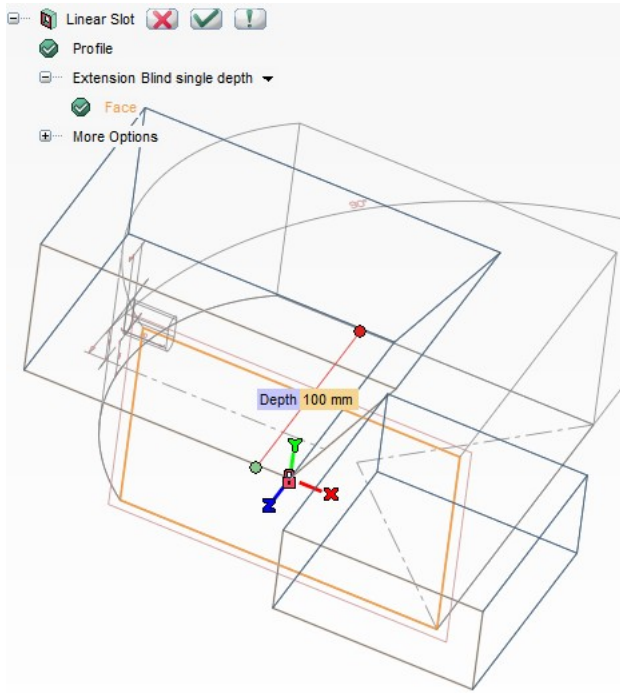
We have now two closed planar shapes composed of 8 curves and they are not still profiles.

- Go to **Options/Properties**.
- Check for the option **Enable in Profile creation** under the **Profile** category of the **Document Properties** tab.



We will see the reason for checking this option later.

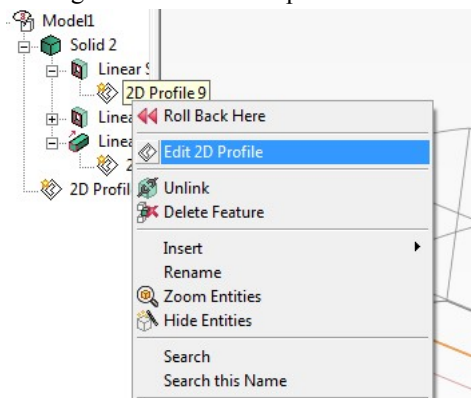
- Start **Linear Slot**.
- Select all the 8 curves and face of the solid.
- Set the depth to 100mm.
- Say **OK**.



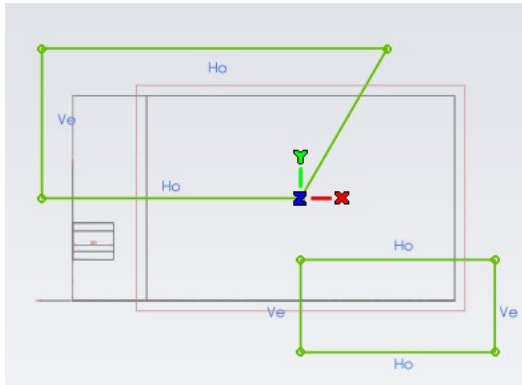
When a slot/protrusion features are created from curves, these curves are automatically turned into profile.

Here, we can insert a single slot feature i.e., two pockets using one profile containing multiple disjoint closed loops (island).

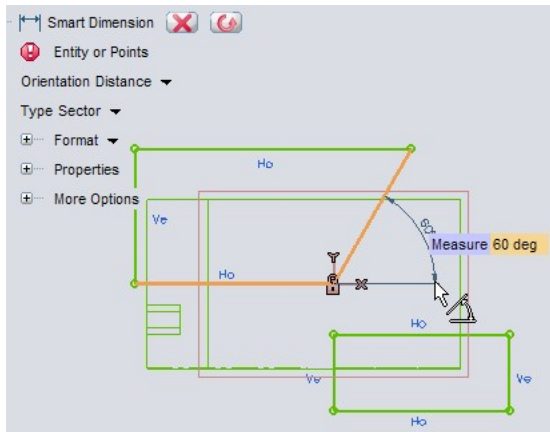
- Right click on the Model icon in History tree and say Hide Dimensions.
- Right click on the last profile created and say Edit Profile.




Notice that constraints are added automatically. This is because we have activated the option Enable automatic constraints in profile creation then.



- Insert an angular dimension 60deg using **Smart Dimension**.

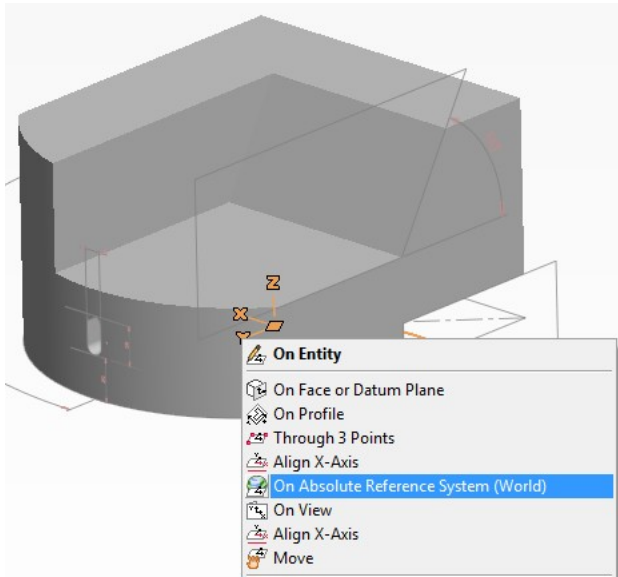


Click  Model tab to switch back to modeling environment.

8. Step 8: Link between profile and work plane

Profiles and Work Planes are like Siamese twins; They follow each other always. We should be aware of the current Work Plane position before sketching a profile.

- Change the display to **Shaded View**.
- Right click on the work plane and select Set to World.

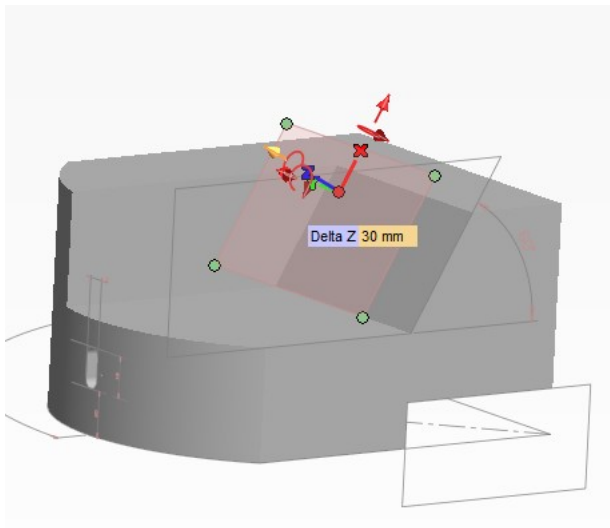


Let's place the Work Plane on the slanted face.

- Right click on the slanted face and say Work Plane Here.

Once it has been set onto that face, Work Plane gets linked to it .See the “locks symbol”. Let's move this work plane to another linked position.

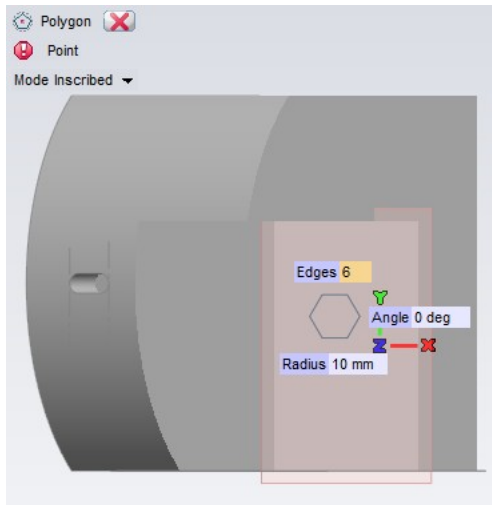
- **Edit** ⇨ **Work Plane** ⇨ **Edit** .
- Translate the work plane 30mm in Z direction.
- Confirm it with OK.



Let's sketch a profile on Work plane. This means that all entities created on Work Plane will also be related to that face.

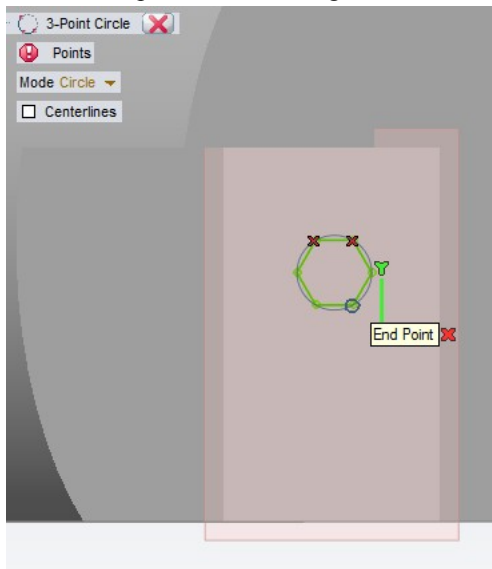
- Hit **F8** and **F**.
- Click 2D Profile

- Activate **Polygon**.
- Set No of edges and Radius to 6 and 10 respectively and place it approximately on center of the face.



Now we need to fix the position of the center of the hexagon, but the hexagon does not have a center point, to snap for or add dimension on it. We shall use reference circle.

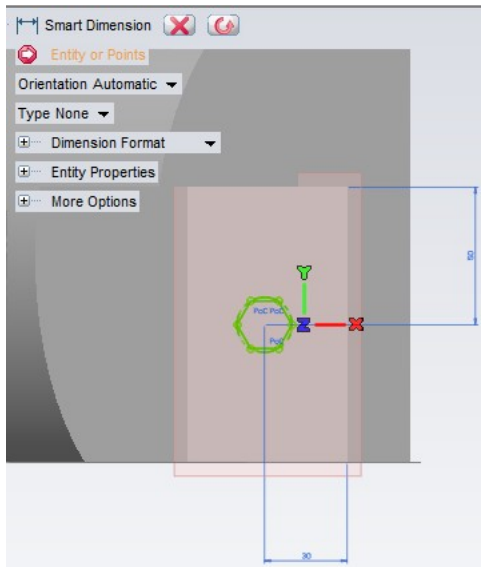
- Start **Three-Point Circle**.
- Select 3 points of the hexagon.



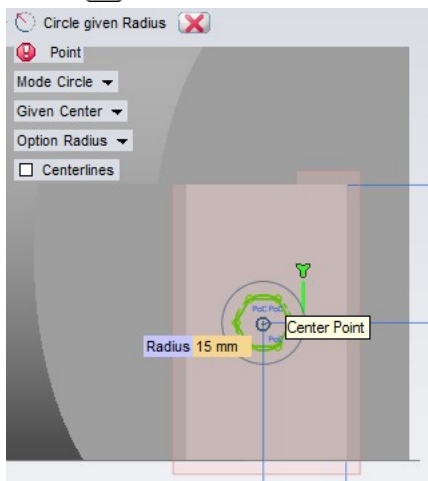
- Select the circle and Make Reference using the command **Make/Unmake Reference**.

Now add 3 Point on Curve constraints manually. Remember, we do not want to have a hexagon, which is completely parametric but only manage its position. So, these 3 constraints will serve our purpose

- Start **Smart Dimension**.
- Dimension as shown in the image.



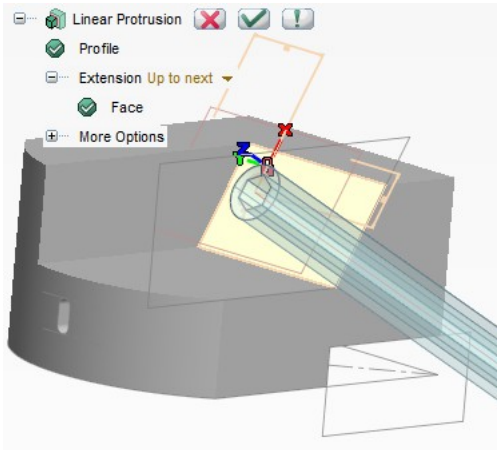
- Activate **Radius Circle**.
- Using **Snap to Arc Center**, draw a circle at the center of the reference circle of radius 15 mm.
- Hit **[Esc]**.



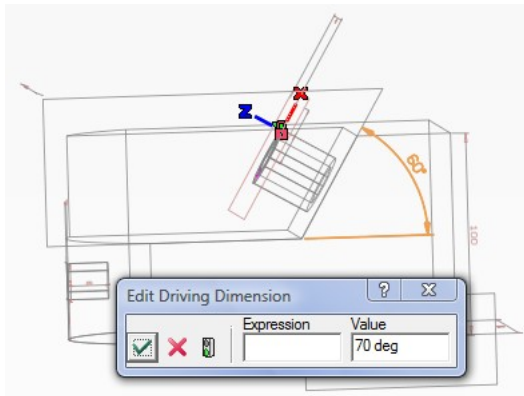
Now we have a profile of Joined island.

Now, let's extrude it up to the slant face.

- Select **Linear Protrusion**.
- Select the profile and change the Extension option to Up to next.
- Select the slanted face.
- Double click on the arrow to invert the direction if necessary.
- Hit **[OK]**.



- Change the display to **Wireframe View**.
- Set the view to **Left View**.
- Change the angle to 70deg.



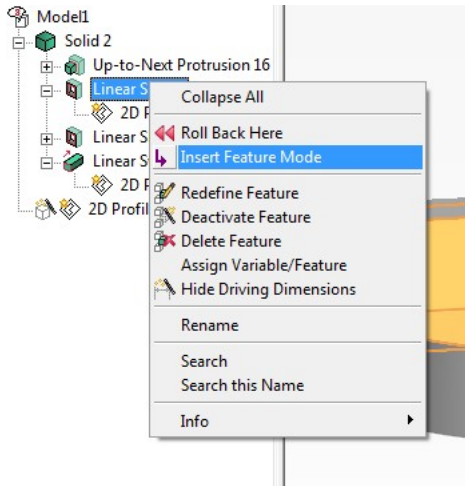
Hit Rebuild.

Notice that the profile of the linear protrusion changes its orientation. This is because its work plane is linked to the solid face.

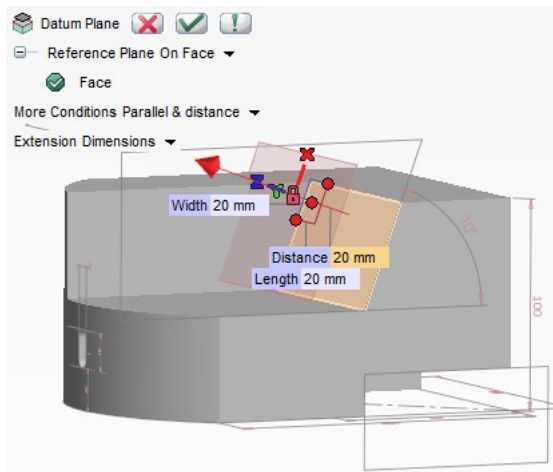
Well, but how can we change the length of the protrusion? We can do that by changing the distance of the profile from the slanted face, but we do not have any dimensional constraint that controls the distance of the work-plane where the profile lies, from the slanted face.

We can use an entity from the history tree called “datum plane”. And the profile must be linked to this plane. So we must insert the datum plane before creation of the profile. Can we do this without creating the the profile and the protrusion again? Answer is YES.

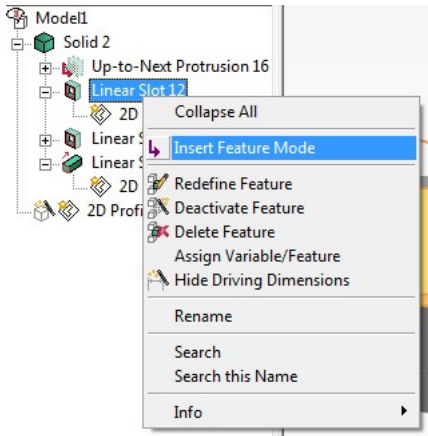
- From the history tree select the slot feature just before the protrusion feature, then right-click and say insert feature mode.



- Start **Datum Plane**.
- Let the reference plane be On Face.
- And change the more conditions to Parallel_distance.
- Set the distance value to 20 mm.
- Confirm with OK.

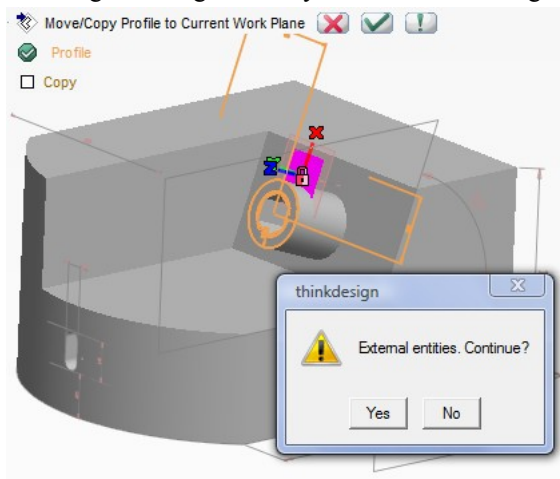


- Change the color of the datum plane to Blue if needed.
- Right click on the datum plane and say Work Plane Here.
- From the history tree select the same slot feature, just before the insert mode sign, right-click and say insert feature mode command again.
- **Rebuild Model** .



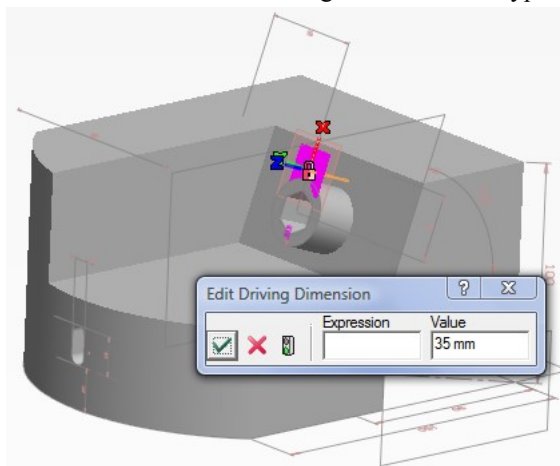
Now we are moving the profile on to the work plane.

- Select **Move/Copy to Current Work Plane**.
- Select the hexagon profile.
- We get messages and say Yes for both messages.



As you can see the profile has moved to this new work-plane but this work-plane is linked to the datum plane. Let's try to change the distance of the datum plane.

- Double-click on its driving dimension and type 35 mm and rebuild.

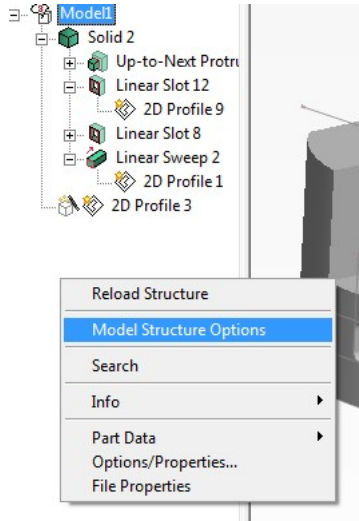


Good, so we can modify the length of the linear protrusion.

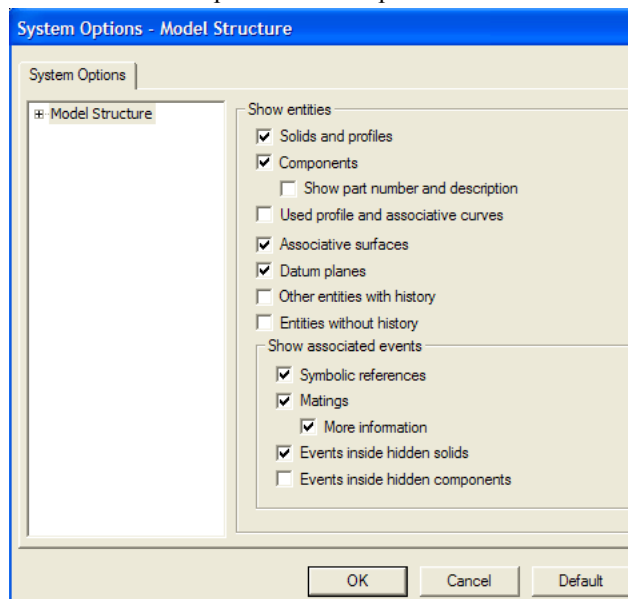
Consider the datum plane to be 100mm from the flat back side surface. Then to achieve this, we need to change the reference face of the datum plane.

Datum planes are entities with history. Hence we can see them in the history tree.

- Right click on the history tree and say History Tree Options from the context menu.

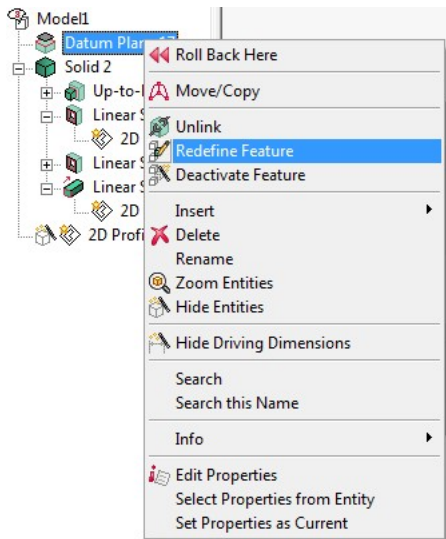


- Check for the option Datum planes.

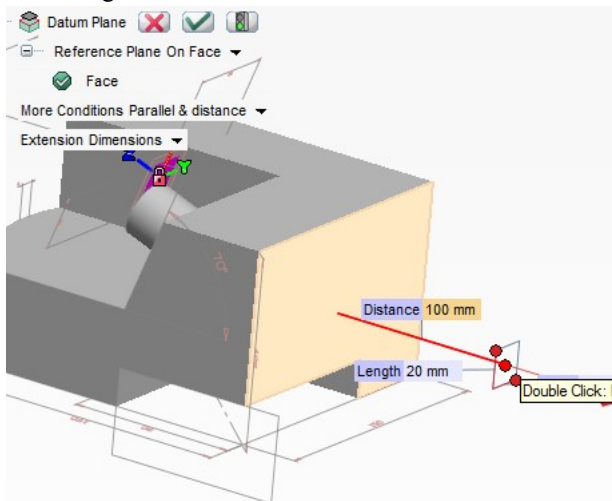


Notice the history tree for the datum plane.

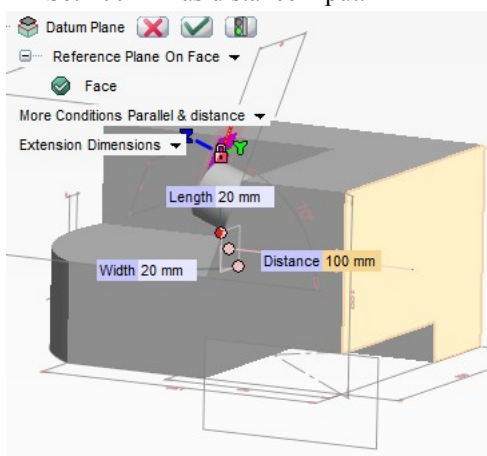
- Right click and say Redefine Feature.



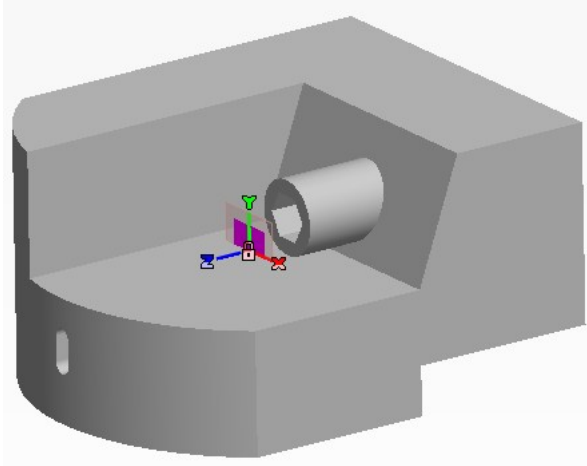
- Select the back face of the model and double click on the arrow to invert the normal direction.
- Change the more condition to Parallel and distance if necessary.



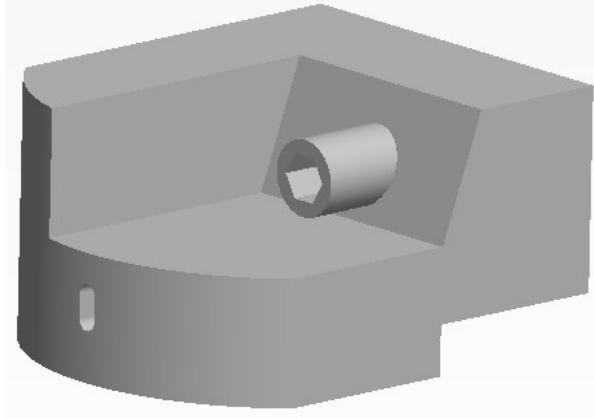
- Set 100 mm as distance input.



- **Rebuild Model .**



The model has been updated to reflect the new position of the datum plane. The linear protrusion has also changed its position since it is linked to the datum plane.



Hide all the dimensions and profiles created.

Good work. Congratulations!!!