Basic Geometric Constraints Tutorial

Using Cobalt™

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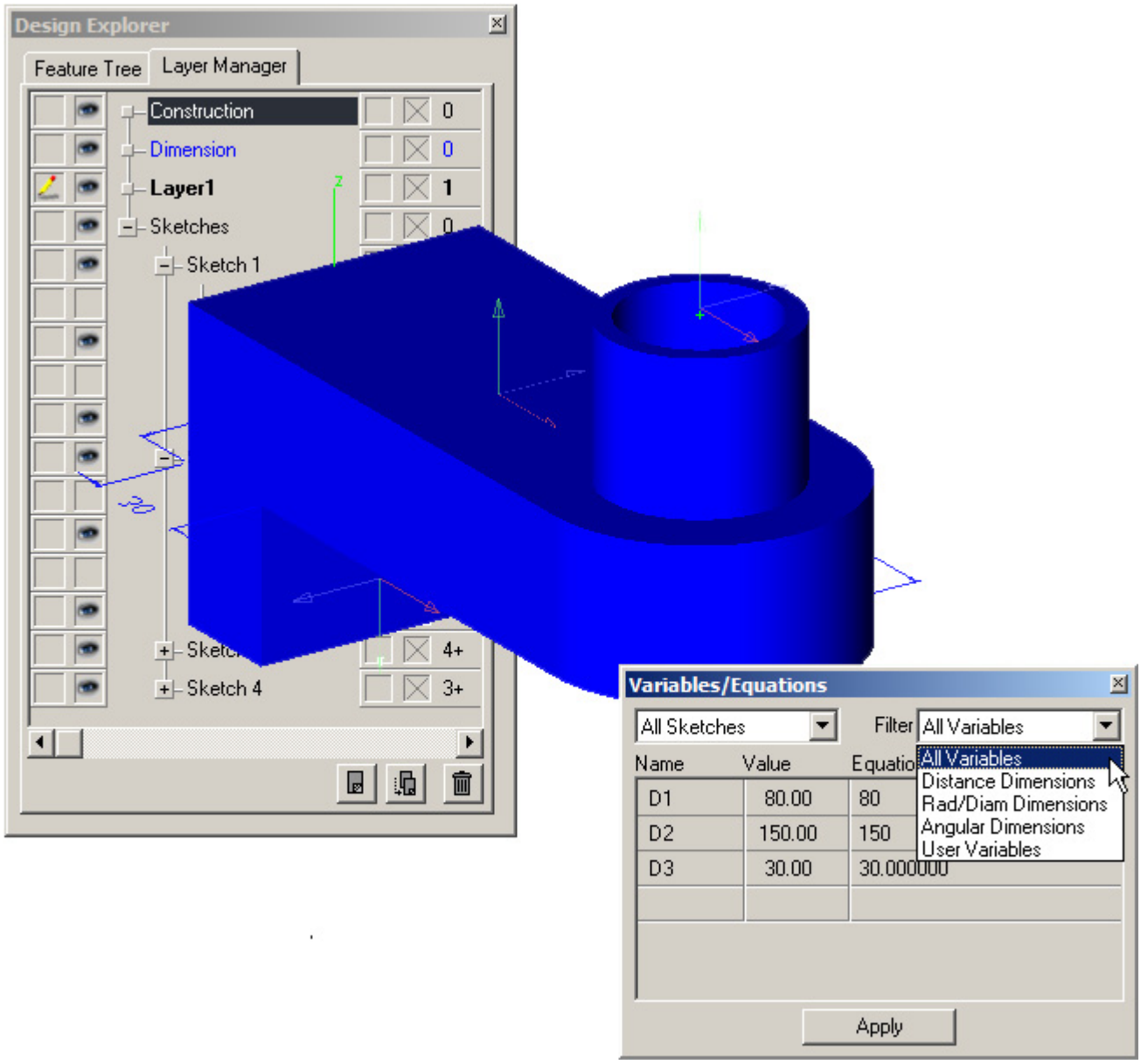
3DCONST0903

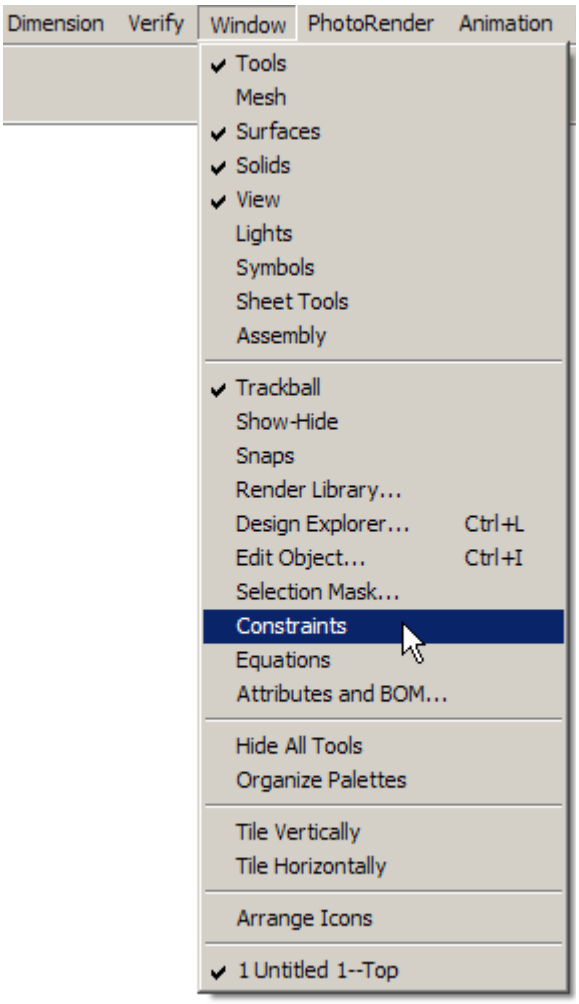
Basic Geometric Constraints Tutorial

This tutorial demonstrates how to set up geometric constraints and equations to drive geometry in Cobalt's Digital Constraint Manager.



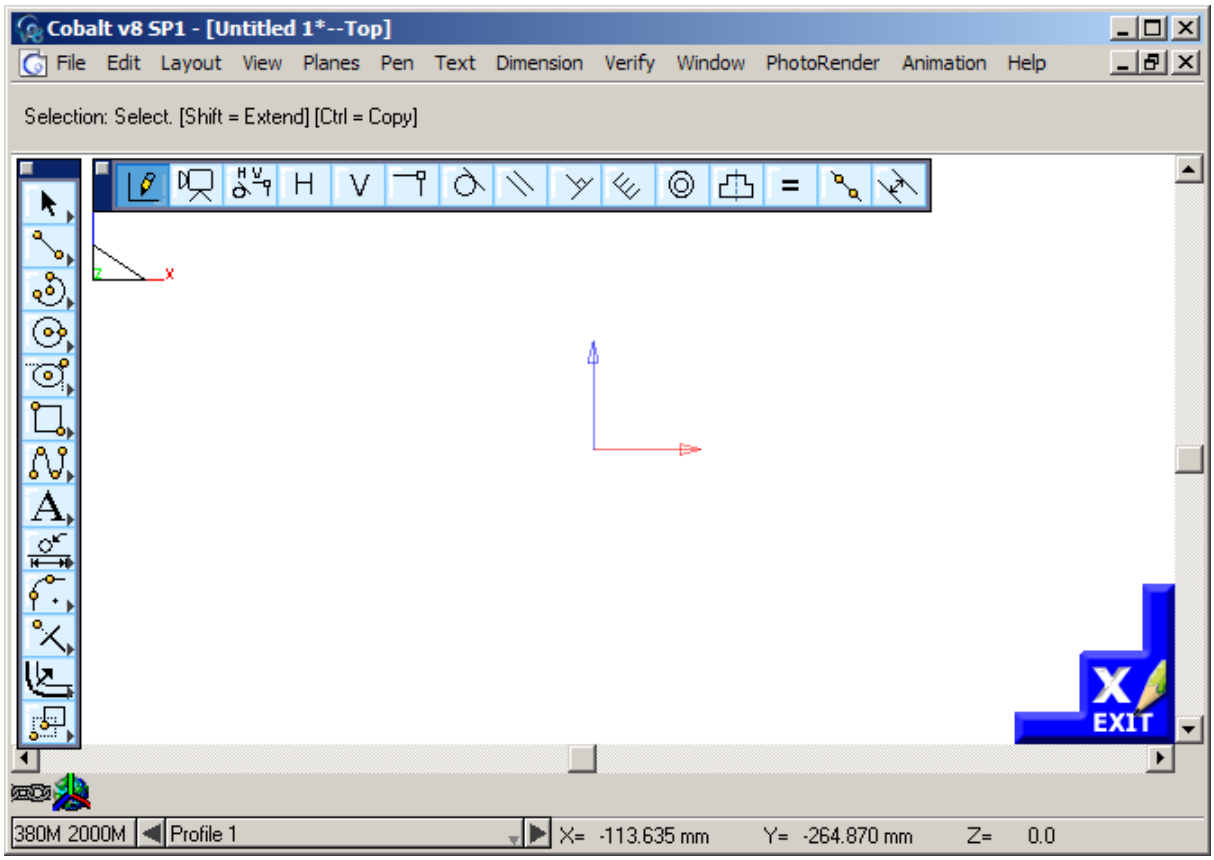
Creating the Base Extrusion

1. After launching Cobalt, select *Constraints* from the Window menu to bring up the **Constraints** palette.

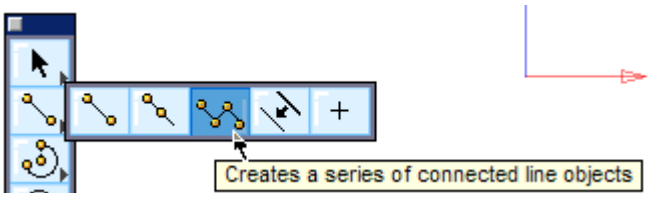


2. Select **Sketch Mode** from the **Constraints** palette then click anywhere in the drawing window to create a new sketch.

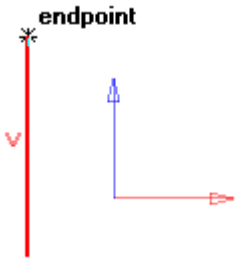
Notice that a new Exit Sketch Mode button appears in the drawing window along with a sketch mode axis. The palettes which can be used within the sketch mode are displayed.



3. Select the **Connected Line** tool from the **Tool** palette.

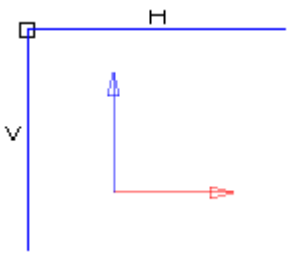


4. Start drawing a vertical line somewhere below and to the left of the Sketch Mode origin which ends somewhere above and to the left. As soon as you click to generate the endpoint of this line, a little V symbol appears at the midpoint to signify that a vertical constraint has been automatically added to the line.



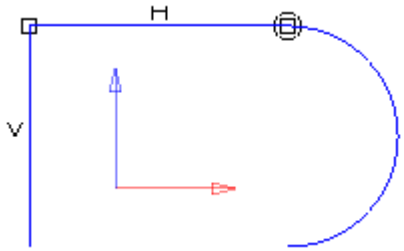
5. Next draw a Horizontal line which starts at the endpoint of the previous line and extends above and to the right of the sketch mode origin. Click to generate the endpoint of this line.

   A little H symbol appears at the midpoint to signify that a horizontal constraint has been automatically added to the line. A little box symbol appears where the two lines meet. This shows that a coincident constraint has been applied between the endpoints of the two lines.



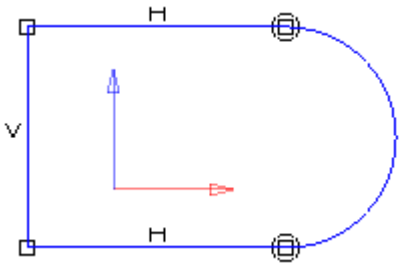
6. Holding down the ALT key (Macintosh) or CTRL key (Windows), draw a tangent arc which begins at the line just created and extends vertically down until the Drafting Assistant displays an *intersection* with the starting point of the first created line.

   As soon as you click to generate the endpoint of the tangent arc, a little box symbol appears where the line and the arc meet. Also a little circle symbol appears at the same point, signifying that a tangent constraint has also been applied between the line and the arc.

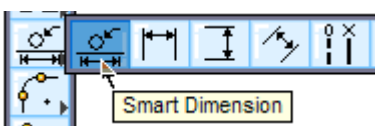


7. Complete the profile with a line which begins at the endpoint at the arc just created and extends to the starting point of the first line. Do not forget to double click the last point to close the profile.

   After double clicking to generate the endpoint of the line, a little box symbol appears where the line and the arc meet to show that a coincident constraint has been applied between the endpoints of the arc and the line. A little circle symbol appears at the same point, showing that a tangent constraint has been applied between the arc and the line. In addition, a little box symbol appears where the line meets with the first line created to indicate that a coincident constraint has been applied between the endpoints of the two lines.



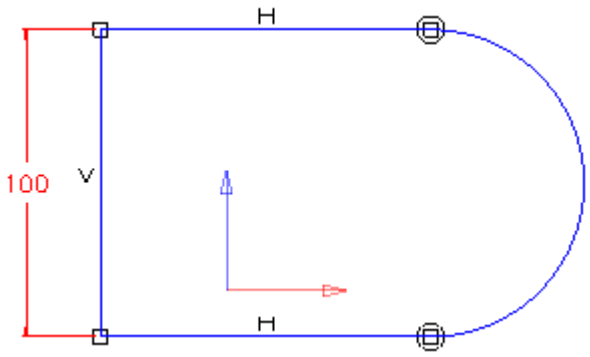
8. Select the **Smart Dimension** tool from the **Tools** palette.



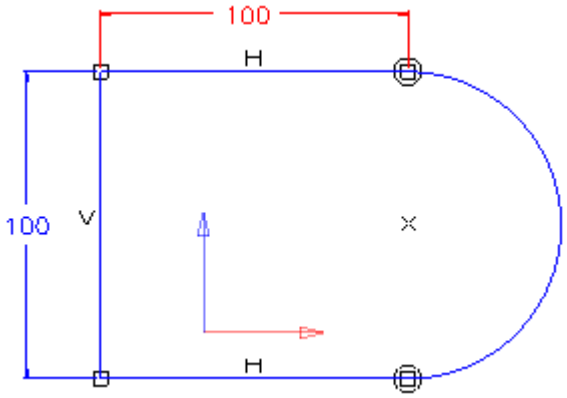
9. Dimension the vertical line by clicking once on the line itself to indicate what to dimension and then once to the left of the vertical line to identify where the dimension text should be placed. With the dimension still highlighted, type 100 mm in the status line and press the ENTER or RETURN key.

   The dimension value in the drawing window is modified and the line's length is resolved to 100mm.

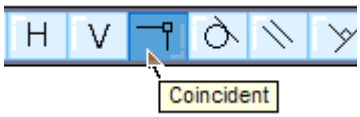
   Although the line's length has been changed, the profile remains intact, so that all endpoints remain connected and tangency is maintained.



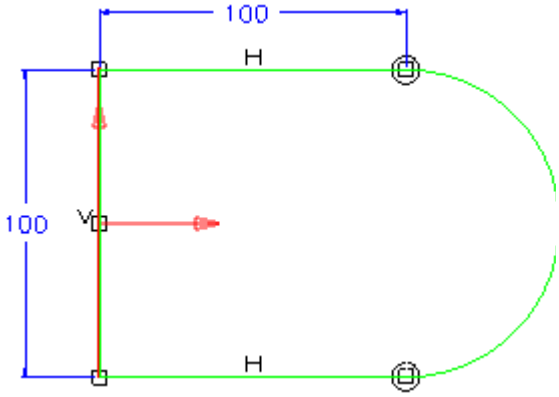
10. Dimension the uppermost horizontal line by clicking once on the line itself to indicate what to dimension and then once above the line to indicate where the dimension text is to be placed. Once again, with the dimension still highlighted, type 100 in the status line and press the ENTER (Windows) or RETURN (Macintosh) key. As before, the dimension value is modified and the line's length is resolved to 100 mm.



11. Select the **Coincident** tool from the **Constraints** palette.



12. Apply a coincident constraint between the midpoint of the vertical line and the origin of the sketch by clicking once on the vertical line and then once on its midpoint to indicate which element to constrain and at which point the constraint should be applied. Then click twice on the sketch origin to indicate which element to constrain the vertical line to and at which point. When the sketch origin is clicked for the second time the whole profile moves to take account of the coincident relationship which has just been added.

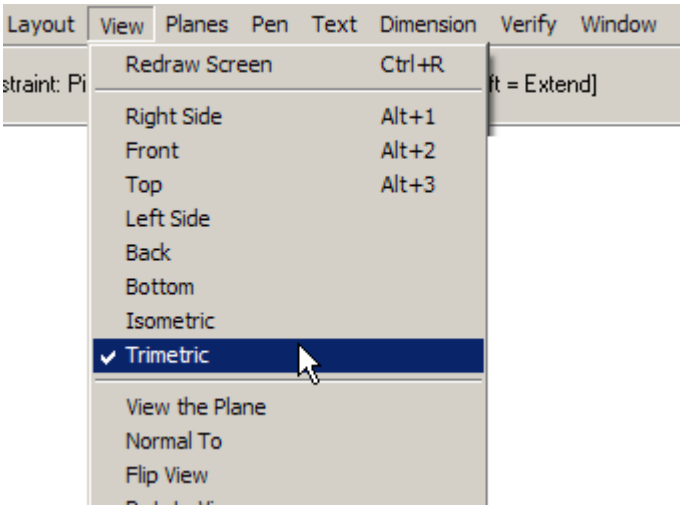


Although the profile has moved to resolve the new coincident constraint, the profile remains intact, so that all endpoints remain connected, tangency is maintained and dimensions remain unchanged. The color of the profile changes from blue to green, indicating that the sketch is now fully constrained so that all the lines and curves in the sketch, and their positions, are described by dimensions or relations, or both.

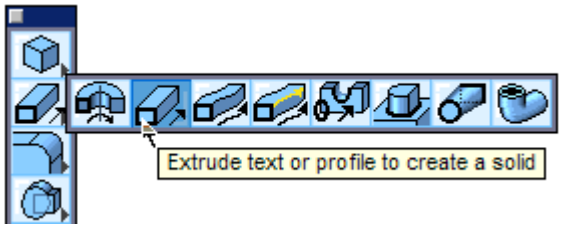
With Cobalt it is not necessary to fully dimension or define sketches before using them to create features.

13. Exit the sketch mode by clicking on the Exit Sketch Mode button in the drawing window.

14. Change the orientation of the view to trimetric by selecting *Trimetric* from the View menu.

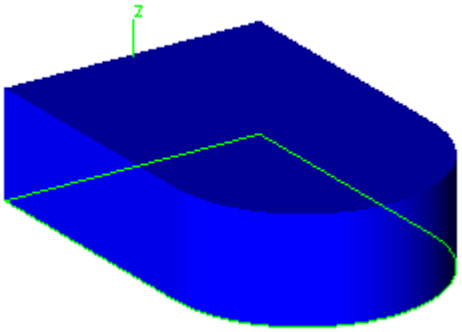


15. Select the **Extrude Solid** tool from the **Solids** palette.



16. Extrude the sketch to form a solid by first selecting the sketch profile then selecting two points in the z axis. With the extrusion still highlighted, type 40mm in the status line Distance field and press the ENTER (Windows) or RETURN (Macintosh) key.

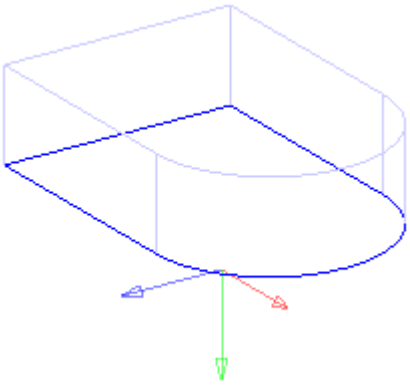
    The base extrusion is now complete.



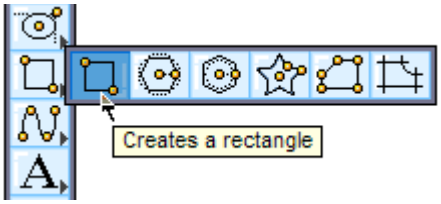
Creating the Rectangular Boss

1. To sketch the profile of the rectangular boss, select the **Sketch Mode** tool from the **Constraints** palette.
2. Instead of selecting anywhere in the drawing window, as before, select the bottom face off the base extrusion.

   Once again, the Exit Sketch Mode button appears in the drawing window along with a sketch mode axis. Only the palettes which can be used within the sketch mode are displayed. Any geometry which does not lie on the sketch plane is dimmed and geometry which does lie on the sketch plane is displayed as normal.

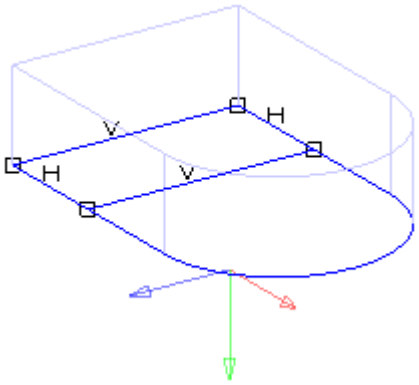


3. Select the **Rectangle** tool from the **Tool** palette



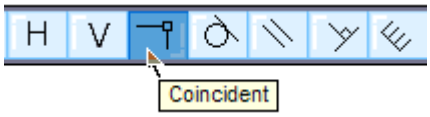
4. Construct a rectangle from the bottom left-hand corner of the base extrusion to some point along the right-hand edge of the extrusion.

   Click on the endpoint of one of the lines which make up the left hand corner of the base extrusion and then click somewhere along the length of the line which makes up the right hand edge of the base extrusion. Little H, V and Box symbols appear to denote that horizontal, vertical and coincident constraints have been applied to the rectangle.



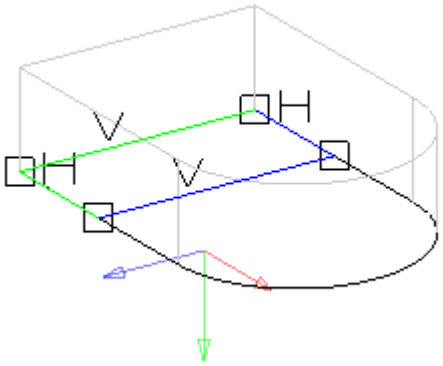
At this point constraints have not been applied between the geometry which makes up the rectangle and the edges of the base extrusion.

5. Select the **Coincident** tool from the **Constraints** palette.

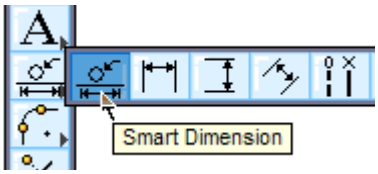


6. Add a coincident constraint between the endpoint of the long edge of the rectangle and the endpoint of the rear edge of the base extrusion.

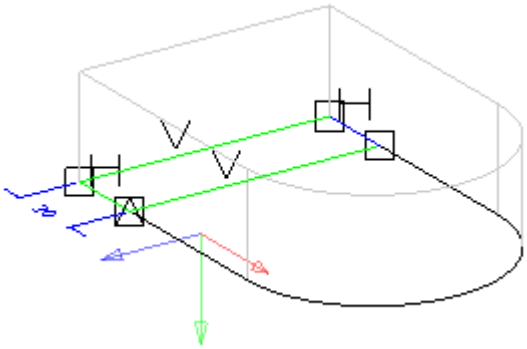
   Do this by clicking once on the line which makes up the long edge of the rectangle and then once on its endpoint to indicate which element to constrain and at which point the constraint should be applied. Then click once on the rear edge of the base extrusion and once on its endpoint to indicate which element to constrain the line to and at which point.



7. Repeat this procedure for the opposite endpoint of the long edge of the rectangle and the opposite endpoint of the rear edge of the base extrusion.
8. Select the **Smart Dimension** tool from the **Tool** palette.



9. Using the **Smart Dimension** tool, first select the line which makes up the left-hand short edge of the rectangle at it's midpoint and then click once to the left of the line to indicate where the dimension text to be placed. With the dimension still highlighted type 30mm in the status line and press the ENTER (Windows) or RETURN (Macintosh) key. The dimension value is modified in the drawing window and the line's length is resolved to 30mm.

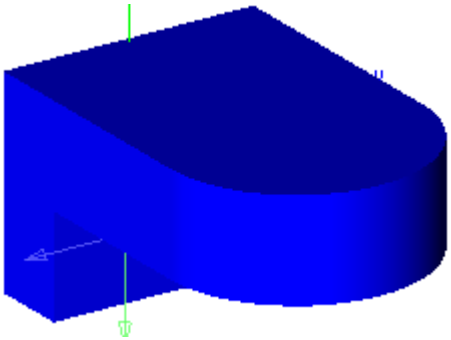


10. As the sketch is now complete and fully defined, exit the sketch mode by clicking on the Exit Sketch Mode button in the drawing window.

11. Select the **Protruded Feature** tool from the **Solids** palette.

12. Extrude the sketch to form a solid by selecting the base extrusion solid, then the sketch profile and finally the two points in the z axis. With the extrusion still highlighted type 40mm in the status line distance field and press the ENTER (Windows) or RETURN (Macitosh) key.

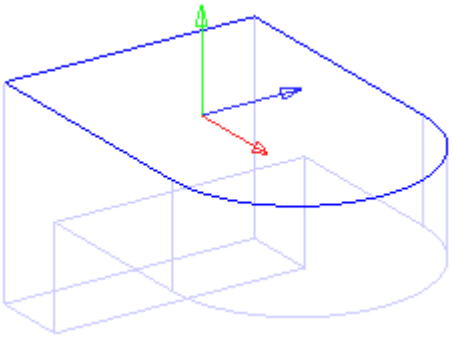
    The rectangular boss is now complete.



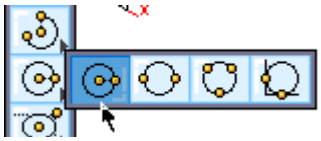
Creating the Cylindrical Boss

1. To sketch the profile of the cylindrical boss, select the **Sketch Mode** tool from the **Constraints** palette and select the top face of the base extrusion. Once again the Exit Sketch Mode button appears in the Drawing window along with a set of sketch mode axis.

   Only the palettes which can be used within the sketch mode are displayed. Any geometry which does not lie on the sketch plane is dimmed and geometry which does lie on the sketch plane is displayed as normal.

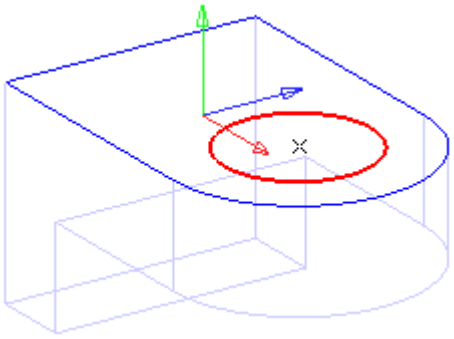


2. Select the **Center Point Circle** tool from the **Tool** palette

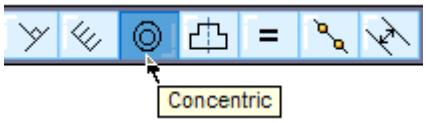


3. Construct a circle with its center positioned on the center point of the arc which defined the cylindrical face of the base extrusion. This is accomplished by first moving the cursor over the edge of the cylindrical face to highlight the center point of the defining arc. Then click once on the highlighted center point and once again to define the radius of the circle.

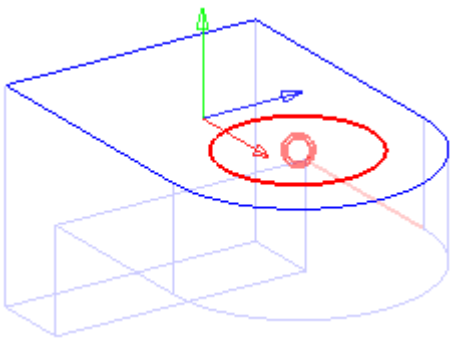
   At this point constraints have not been applied between the circle and the edges of the base extrusion.



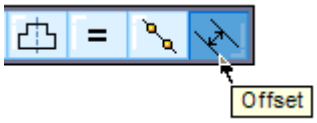
4. Select the **Concentric** tool from the **Constraints** palette.



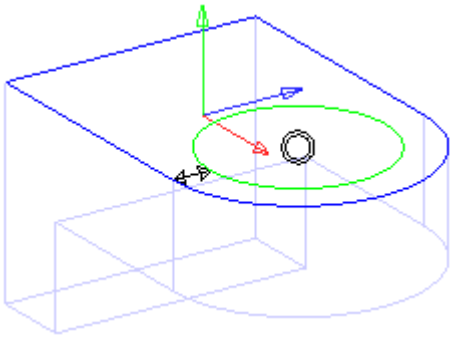
5. Add a concentric constraint between the circle and the edge of the cylindrical face of the base extrusion by clicking once on the circle and then once on the edge of the cylindrical face to indicate which element to constrain and to what.



6. Instead of using a **Smart Dimension** to define the diameter of the circle, this time an **Offset Constraint** between the circle and the edge of the cylindrical face of the base extrusion.
7. Select the **Offset** tool from the **Constraints** palette.

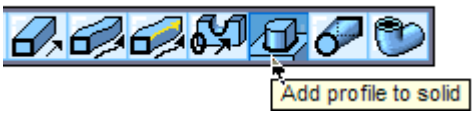


8. Add the offset constraint by clicking once on the circle and then once on the edge of the cylindrical face to indicate which element to constrain and to what. With the offset still highlighted type 15mm in the status line Offset field and press the ENTER (Windows) or RETURN (Macintosh) key.



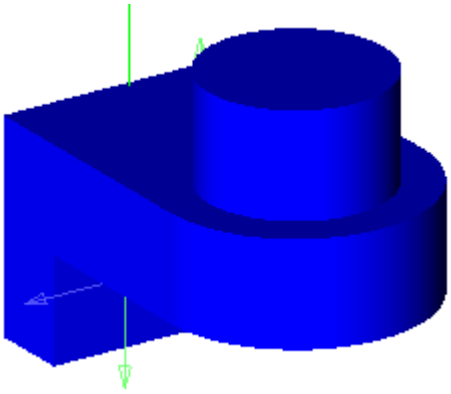
9. As the sketch is now complete and fully defined exit the sketch mode by clicking on the Exit Sketch Mode button in the drawing window.

10. Select the **Protruded Feature** tool from the **Solids** Palette.



11. Extrude the sketch to form a solid by first selecting the base extrusion solid to which to add to, then select the sketch profile and finally select two points in the z axis. With the extrusion still highlighted type 40mm in the status line Distance field and press the ENTER (Windows) or RETURN (Macintosh) key.

    The cylindrical boss is now complete.

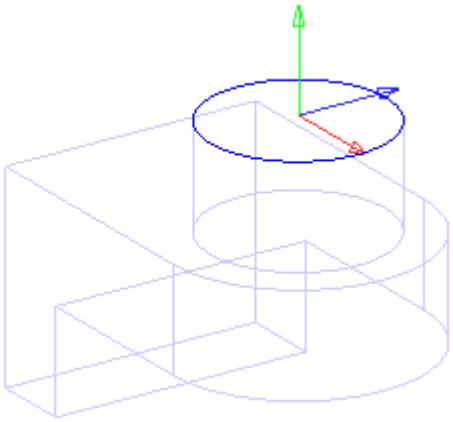


Creating the Counter Bore Hole

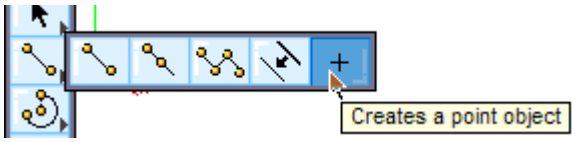
The creation of the counter bore hole is a two step process, first the center of the hole needs to be defined using a constrained sketch and then the hole feature itself needs to be created.

1. Select the sketch mode tool from the **Constraints** palette and select the top face of the cylindrical boss.

   Once again, the Exit Sketch Mode Button appears in the drawing window along with a set of sketch mode axis and only the palettes which can be used within the sketch mode are displayed. Any geometry which does not lie on the sketch plane is dimmed and geometry which does lie on the sketch plane is displayed as normal.

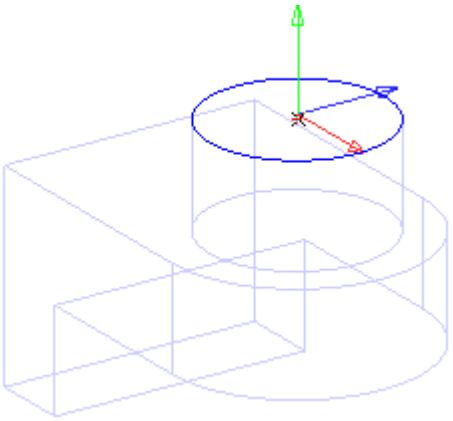


2. Select the **Point** tool from the **Tool** palette.

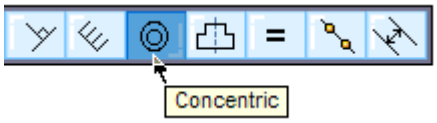


3. Click anywhere on the top face of the cylindrical boss to place the point.

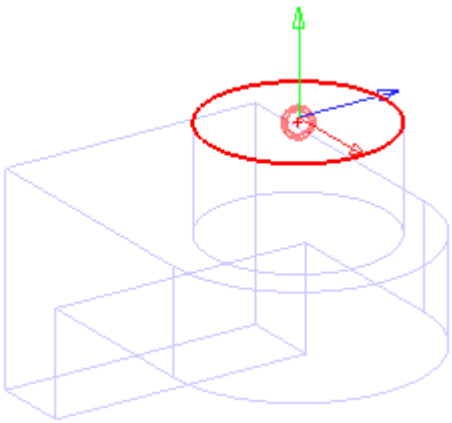
   At this point a constraint has not been applied between the point and the edge of the cylindrical boss.



4. Select the **Concentric** tool from the **Constraints** palette.

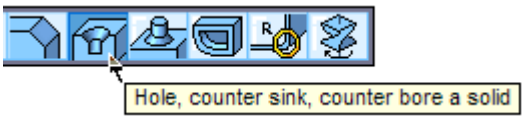


5. Add a concentric constraint between the point and the edge of the cylindrical boss by clicking once on the point and then once on the edge of the cylindrical boss to indicate which element to constrain and to what.

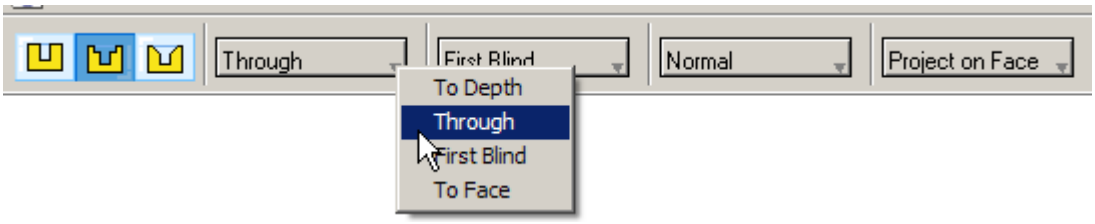


6. As the sketch is now complete and fully defined, exit the sketch mode by clicking on the Exit Sketch Mode button in the drawing window.

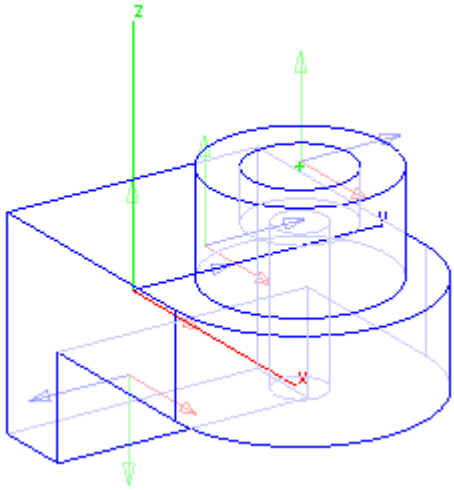
7. Select the **Hole** tool from the **Solids** palette.



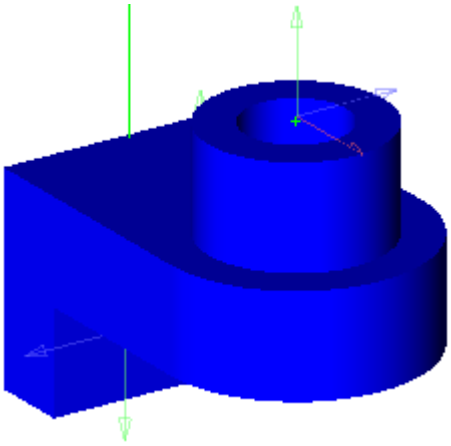
8. Ensure that the hole type is set to a **Counter Bore** and the options in the message line are specified as in the picture below.



9. Using the **Hole** tool first select the top face of the cylindrical boss and then select the point just created. With the counterbore hole still highlighted, type 20mm, 20mm and 40mm in the status line Diameter, Bore Depth and Bore Diameter fields respectively and press the ENTER (Windows) or RETURN (Macintosh) key.

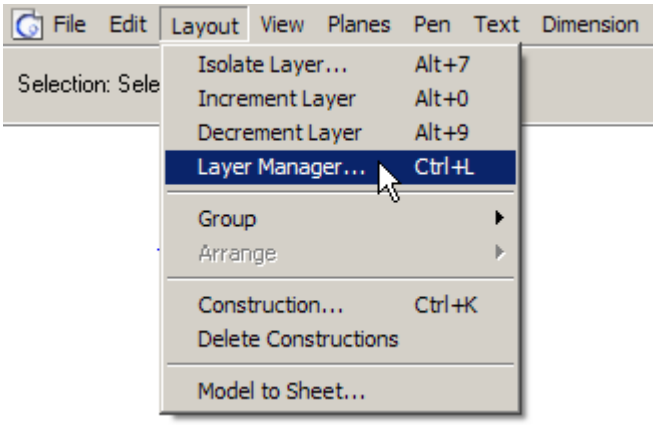


The part is now complete, however, to show the true power of constraints it is necessary to go back and make some modifications.

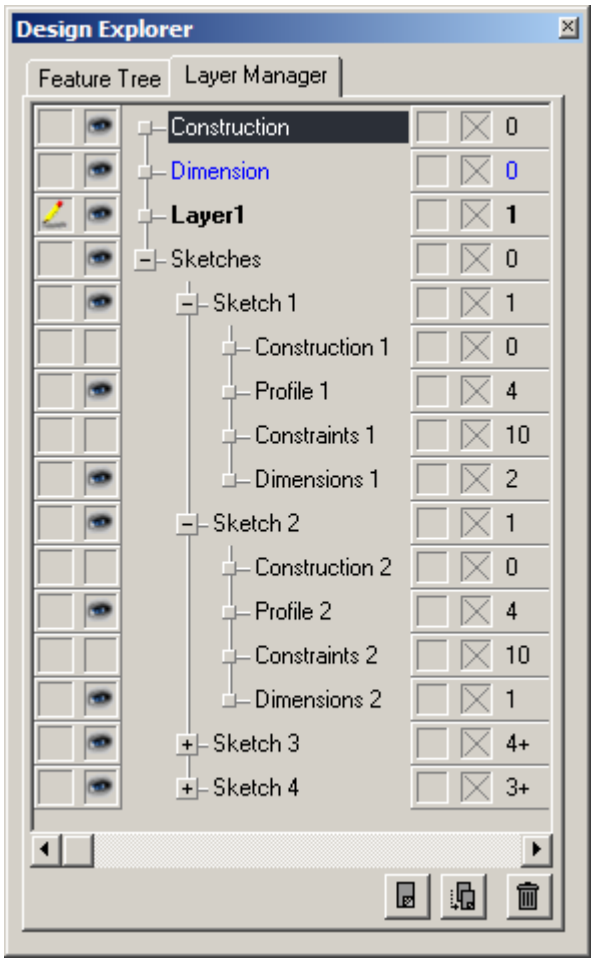


Modifying the Part

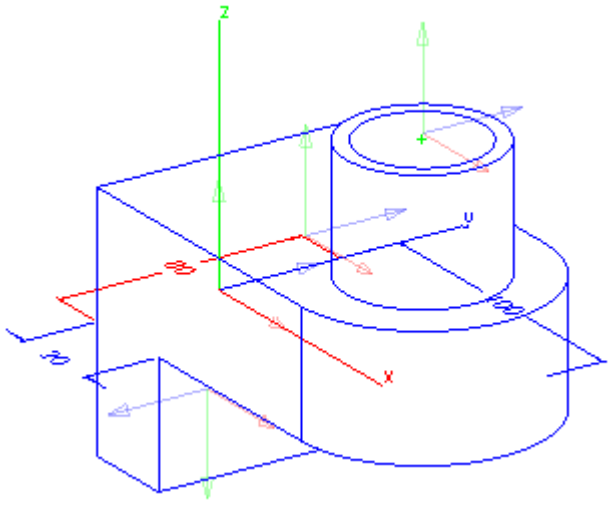
1. Select *Layer Manager* from the Layout Menu.



2. Open the Sketches branch by clicking on the plus button next to the word Sketches, then open the branches for Sketch 1 and Sketch 2. Finally, click on the empty box to the left of the layer names for Dimensions 1 and Dimensions 2 to display those layers.

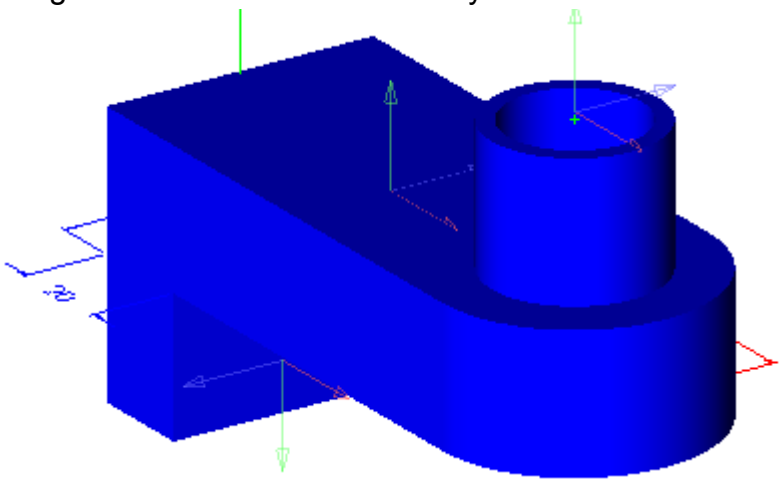


3. Select the dimension at the rear of the component which currently reads 100mm. Change this value to 80mm by typing 80mm in the status line and pressing the ENTER (Windows) or RETURN Macintosh) key.



The base extrusion resolves to be 80mm wide, as does the rectangular boss which is driven by the coincident constraints applied between the long edge of the sketch rectangle that defines the boss and the edge of the base extrusion. The diameter of the cylindrical boss also decreases as it is driven by the offset constraint applied between the circle that defines the boss and the edge of the cylindrical face of the base extrusion.

4. Select the dimension on the right hand side of the component which currently reads 100mm. Change this value to 150mm by typing 150mm in the status line and pressing the ENTER or RETURN key.



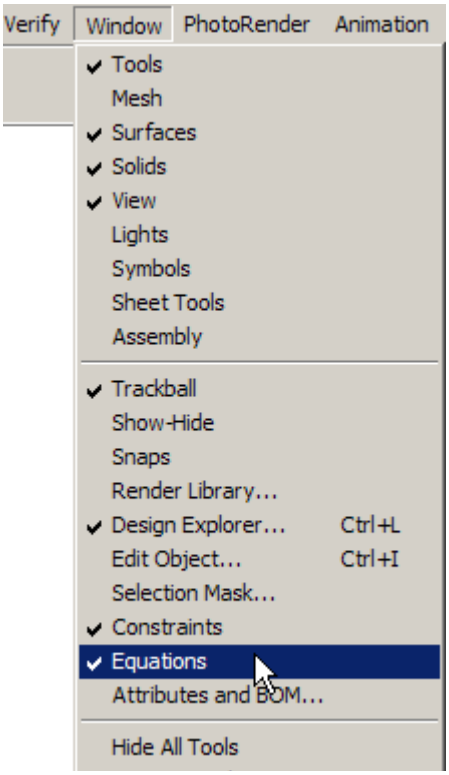
The base extrusion resolves to be 150mm in length. The cylindrical boss and counter bore hole also remain concentric with the cylindrical face of the base extrusion as a result of the concentric constraints applied to both the circle which defines the cylindrical boss and the point on the top face of the cylindrical boss which defines the center of the counter bore hole.

Modifying The Part Using An Equation

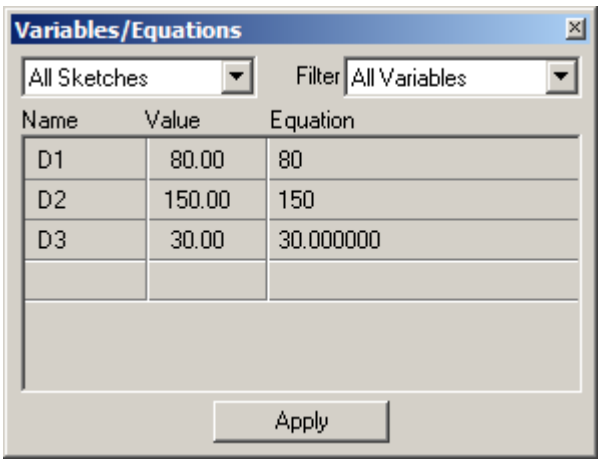
It is not only possible to drive a constrained component via manual modification of dimension values but it is also possible to modify a component using relationships between dimension values.

In this case an equation can be used to add a relationship between the length of the component and the length of the rectangular boss so that the length of the rectangular boss is always 1/3 the length of the long edge of the base extrusion.

1. Select *Equations* from the Window menu.

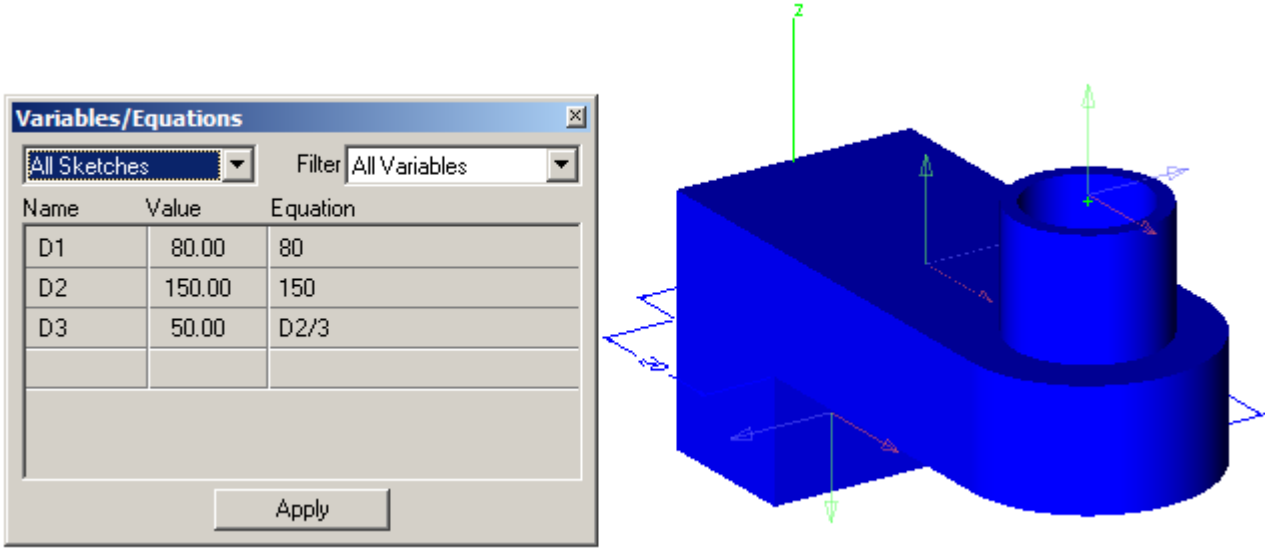


2. Ensure that both the sketches drop down on the left and the variables drop down on the right are set to display all sketches and all variables.



3. Enter D2/3 in the equation field for the variable D3 and then click on the Apply button.

   The value for D3 is resolves to 1/3 D2.



Congratulations. You have just learned how to set up geometric constraints and equations within Cobalt to drive geometry.