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CAD MODELING ESSENTIALS IN 3DEXPERIENCE® 2016x

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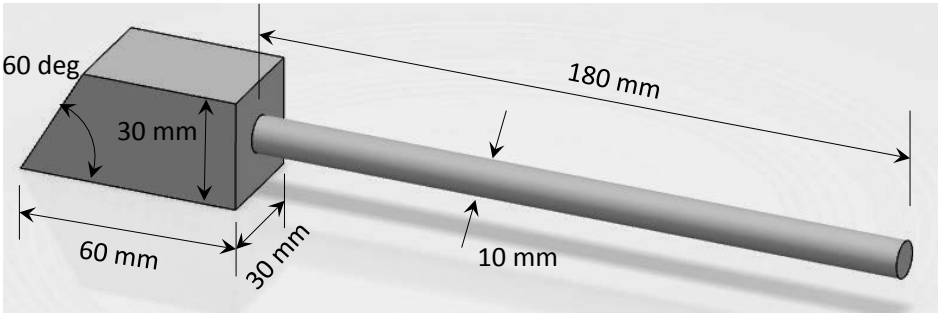
Chapter
02

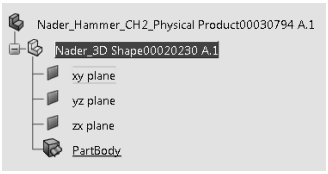
Making a Hammer

Objective:

In this chapter, you will explore the process of making a simple hammer as shown in the figure below. These are nominal dimensions in (mm) and only certain basic icons will be used in this problem. After making the hammer, some other features of the Part Design App will be explored which are not reflected in the drawing. In order to create the model, including accessing the App, setting up your units, and naming conventions, you need to follow the instructions in chapter 1.

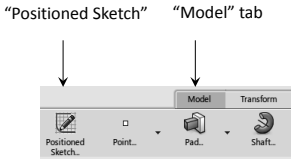


Enter the Part Design App and create a part with the name of your choice. Here, the name below was used.



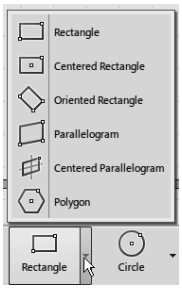
Nader_Hammer_CH2_Physical Product00030794

1- Select the yz plane from the tree or the screen and choose the "Position Sketched" icon from the bottom row menu (i.e., from the action bar). Note that the "Positioned Sketch" is in the "Model" tab. This will land you in the Sketcher App.

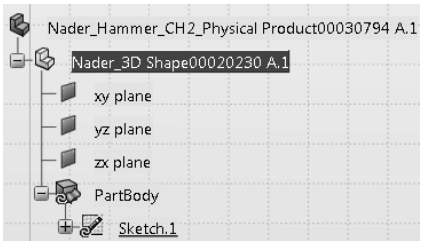


2- Select the "Rectangle" icon from the bottom row (i.e., the action bar) and draw a rectangle in an arbitrary location. Note that the "Rectangle" icon belongs to a submenu with many other choices shown on the right.

Obviously there are other ways of making a rectangle.

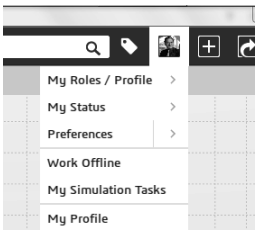


The tree shows you that rectangle has been created in "Sketch.1." Also, note that there is a background grid with default spacing.

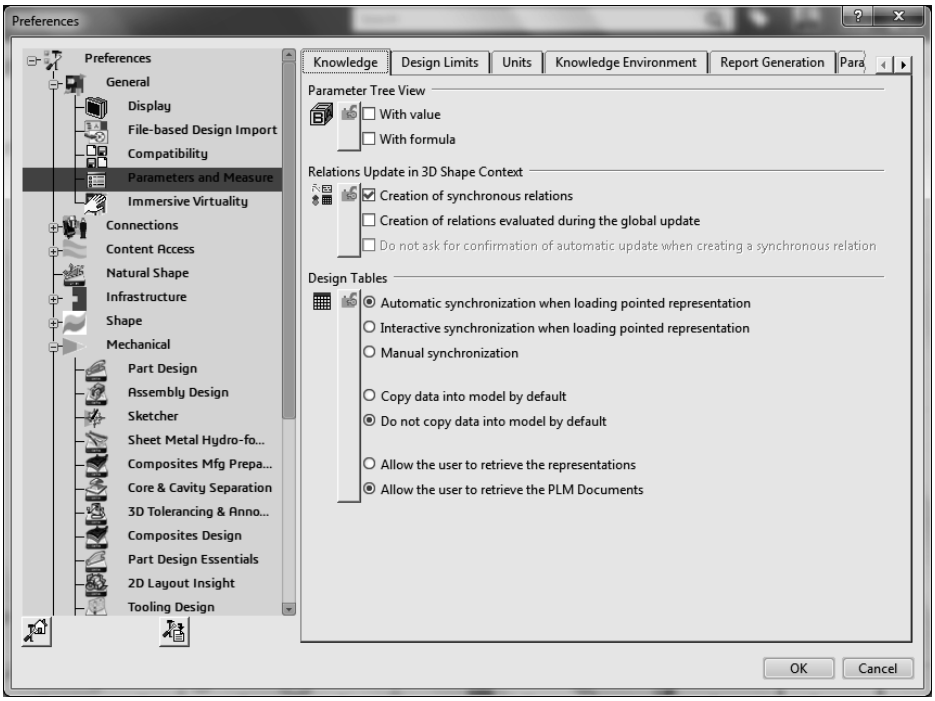


Before setting the dimensions of the hammer in the sketcher, a few words about the background grid is in order.

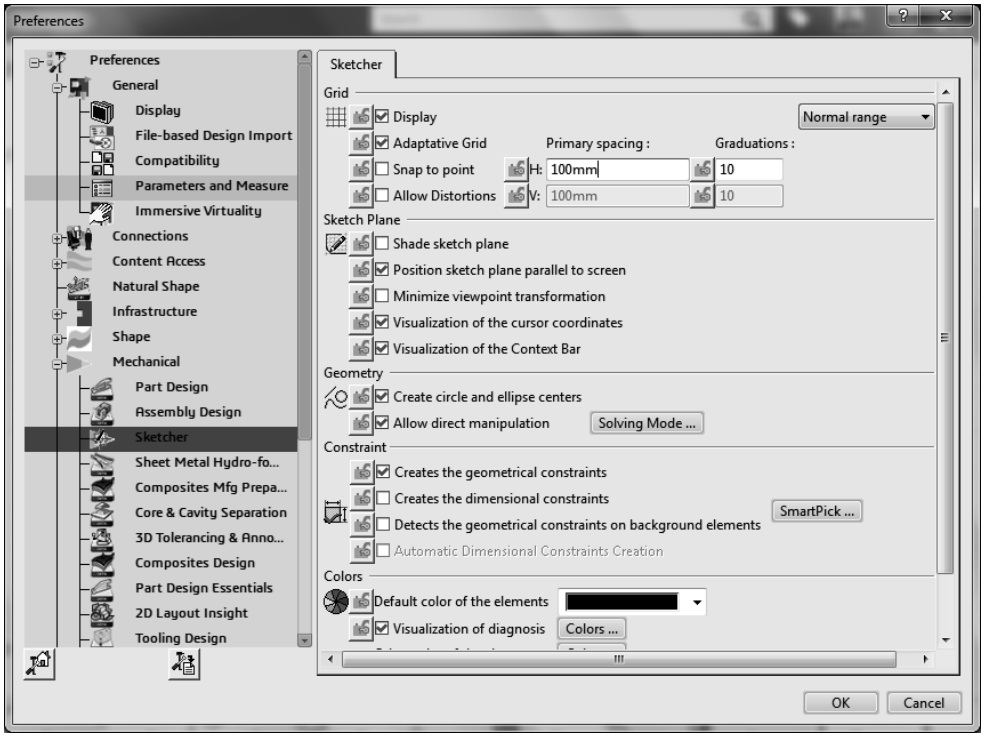
The grid spacing and the "graduation" of the grid can be changed by following the steps given next. Select the "Me" icon from the right top corner of the screen and click on "Preferences."



This leads to the following window.

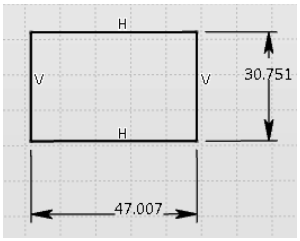


Choose the "Mechanical" branch followed by "Sketcher." This is where these parameters are set. The current values are 100mm and a graduation of 10 (see the top of the next page). The changing of the units was already discussed in chapter 1 (page 9).

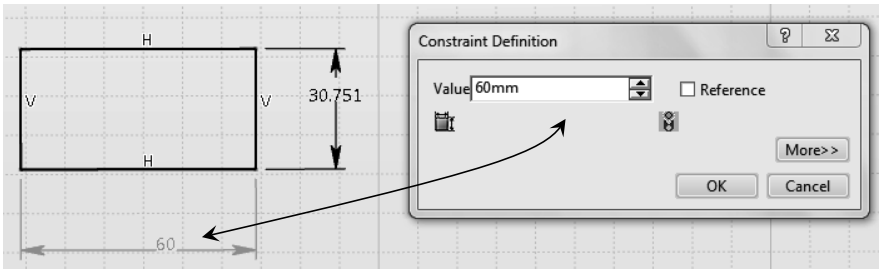


3- Select the "Constraint" icon

from the bottom menu and create dimensions for the sides as shown. Obviously these are not the correct dimensions. To do so, double click on the dimension (one at a time) and in the resulting dialogue box, change the number to the correct dimensions.

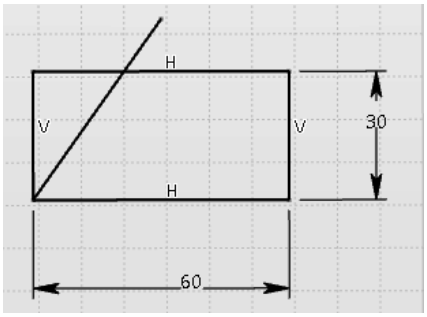


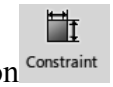
The resulting rectangle must be 60mm x 30mm.



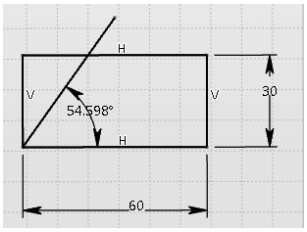
4- Next, using the "Line" icon

draw a slanted line from the bottom left corner in an arbitrary orientation.

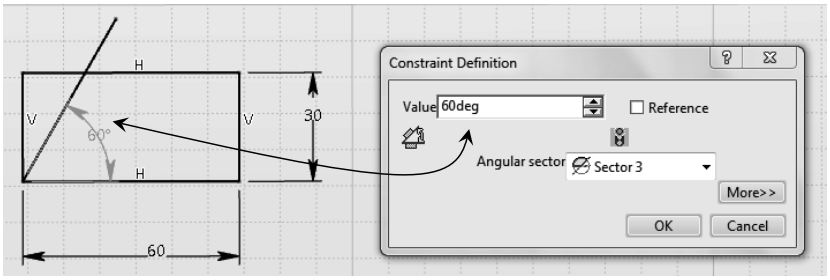


Select the "Constraint" icon 

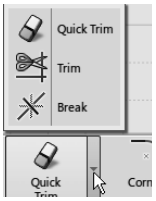
from the bottom menu and create angular dimensions between the slanted line and the horizontal base of the rectangle. To create an angle dimension, after selecting the icon use the cursor to select the two lines involved. Do not panic if the color of the dimensions begin to change after selecting the first line.



To change the dimension to 60 degrees, double click on the dimension and change it in the resulting dialogue box.

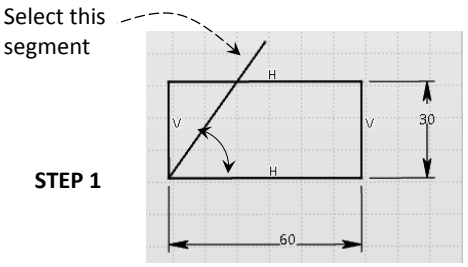


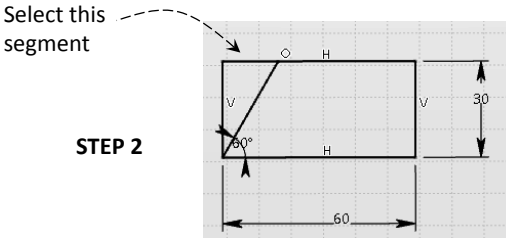
5- The next step is to trim the rectangle with the line. There are different ways of doing this but it is recommended to use the "Quick Trim" .

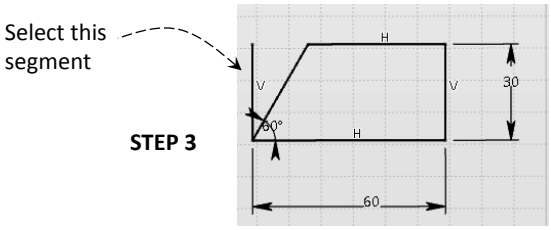


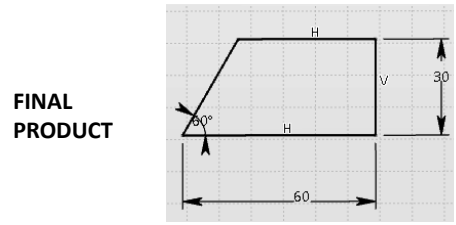
Note: Since you have to use the "Quick Trim" icon repeatedly (three times), it is better if you double clicked on it, which keeps it active until you select a different icon.

With the "Quick Trim" icon selected, follow the three steps shown below.

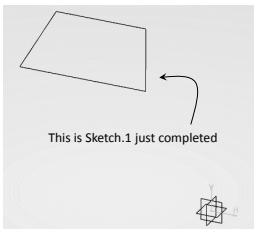


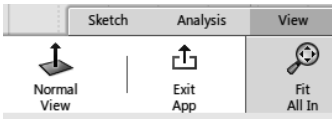




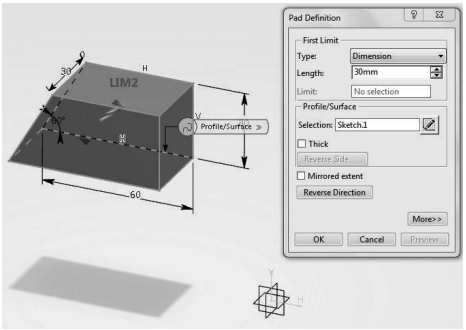


6- Use the "Exit App" icon to exit the sketcher and land in the 3D space. This App is one of the icons in the bottom row of icons (i.e., in the action bar).





7- Select the "Pad" icon from the action bar to open the "Pad" dialogue box. For the "Selection" pick "Sketch.1" from the screen or from the tree. For "Length," input 30 mm.

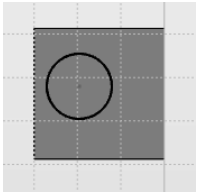


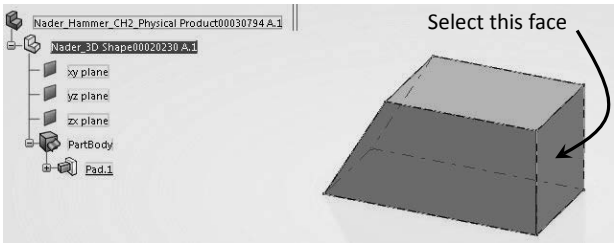
8- Once the pad is completed, select the face of the hammer as shown and pick the "Positioned Sketch" icon

. You will be drawing a circle of diameter 10 mm on this face.

Using the "Circle" icon

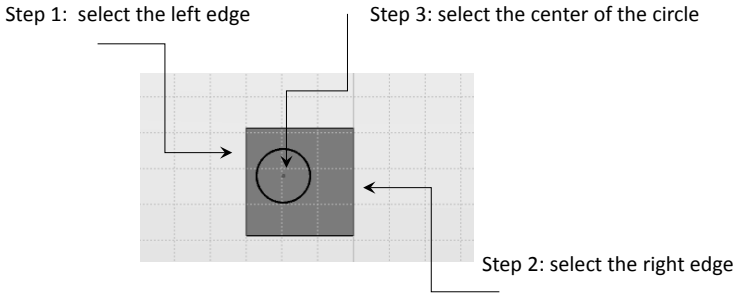
draw a circle.



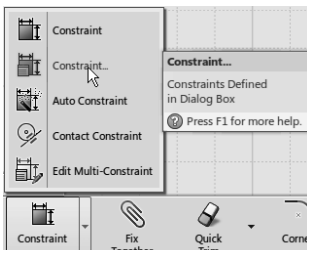


Notice that the center of the circle is not symmetrically positioned within the square face selected. Of course, this can easily be corrected by imposing dimensional constraints between the center and the edges. However, you are shown an alternative way to achieve this.

Using the Ctrl key (**held down for multiple selection**), select three entities in the following order:
Step 1: select left edge of the square.
Step 2: select the right edge of the square.
Step 3: select the center of the circle.

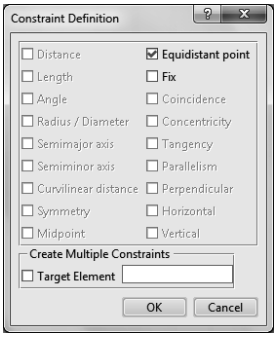


Click on the "Constraint…" icon from the available choices shown on the right.



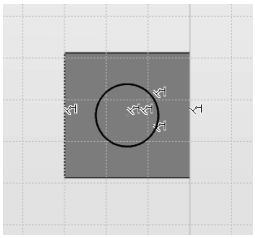
Upon completing the three steps and the selection of the above icon, a dialogue box appears.

Within this window, check the box labeled "Equidistant point." This will center the circle horizontally. Repeat the same process for the top edge, the bottom edge, and the center of the circle.



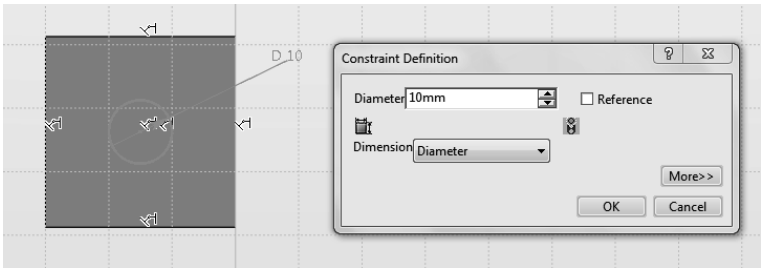
Using the Ctrl key (**held down for multiple selection**), select three entities in the following order:
Step 1: select top edge of the square.
Step 2: select the bottom edge of the square.
Step 3: select the center of the circle.

Once again, select the "Constraint…" icon and check the "Equidistant point" option. The circle is now completely centered with the side face of the hammer regardless of the dimensions of the hammer. The design intent was to have a centered circle and this accomplished the task.



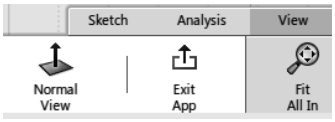
You now need to specify the diameter of the circle. Select the circle followed by the "Constraint" icon

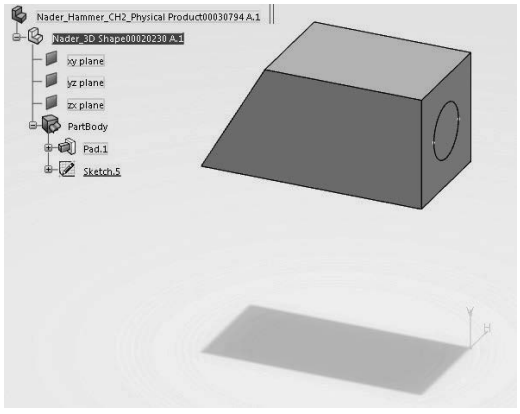
and in the resulting dialogue box enter 10 mm.



Use the "Exit App" icon

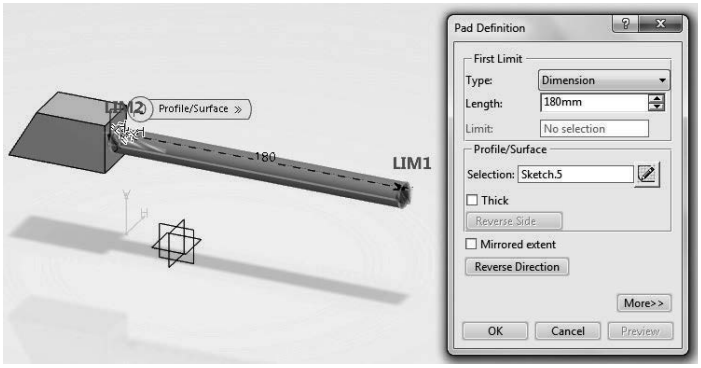
to exit the sketcher and land in the 3D space. This App is one of the icons in the action bar.



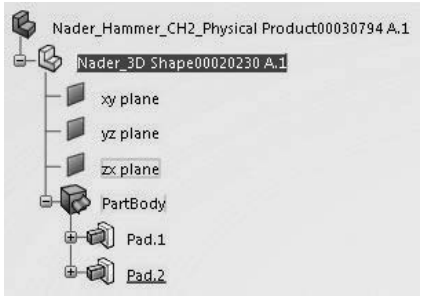


9- Select the circle just created in the previous step followed by the "Pad" icon

. In the resulting dialogue box input 180 mm and close the window.



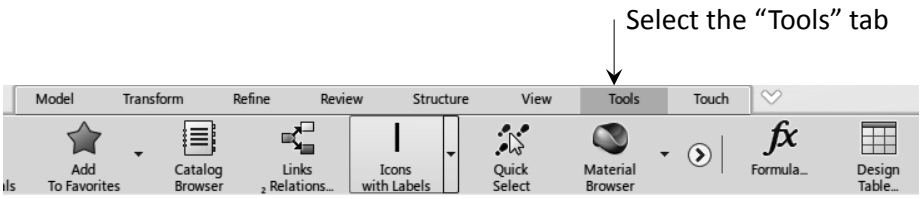
Upon closing the "Pad Definition" window and checking the tree, you will notice two "Pad" features, Pad.1 which was the hammer head, and Pad.2 which is the hammer handle.



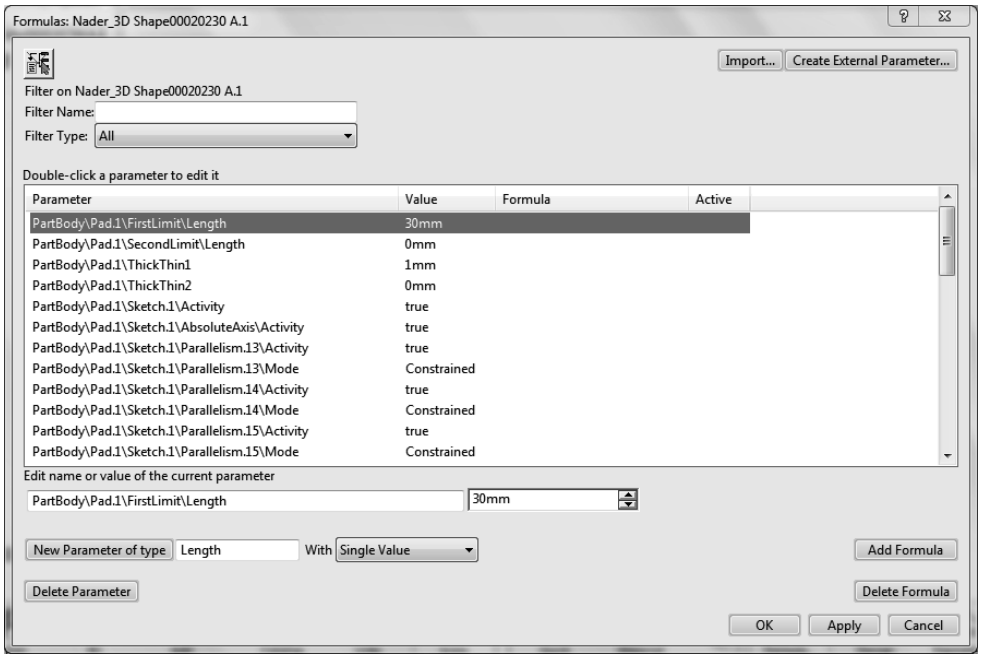
This completes the steps needed to create the hammer. The point discussed below is a very useful tool to change the dimensions of the geometry. The most straightforward approach to change a dimension (in the sketch) is to locate the appropriate sketch, and double click on the branch of the tree where the sketch resides. This will land you in the sketch where the appropriate changes can be made. An alternative approach is presented below.

Select the "Formula" icon

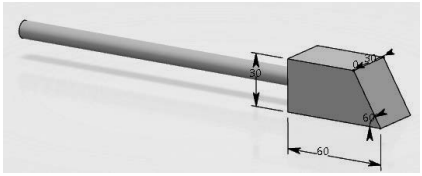
from the "Tools" tab shown below.



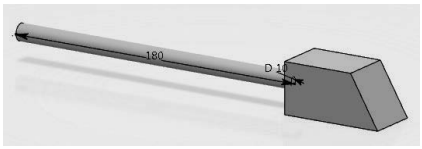
This leads to the following dialogue box.



Selecting the Pad.1 from the tree (or with the cursor pointing to the hammer head) shows you the dimensions that were used to create the part as shown on the right.



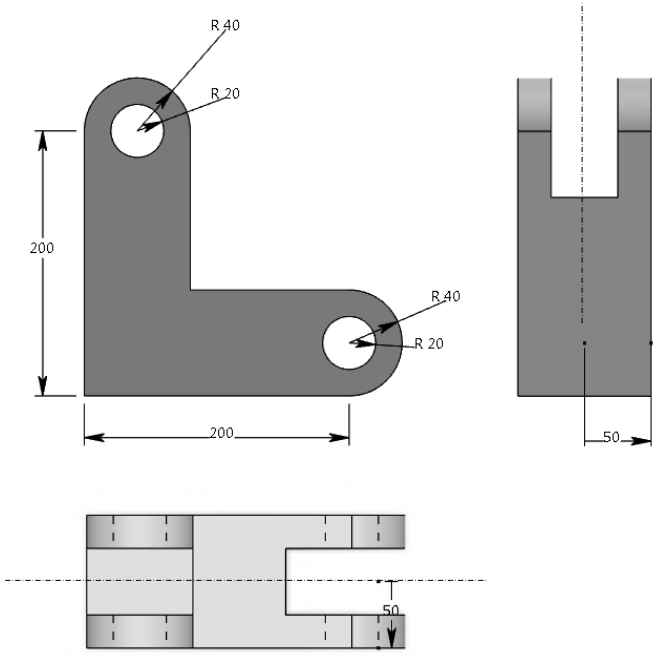
Selecting the Pad.2 from the tree displays the dimensions of the hammer handle.

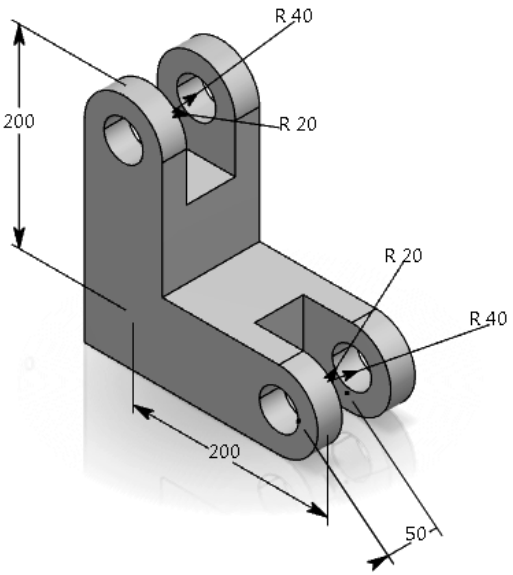


Double clicking any of the dimension on the screen opens a dialogue box which enables you to change the values without going into the sketcher environment.

Exercise 1:

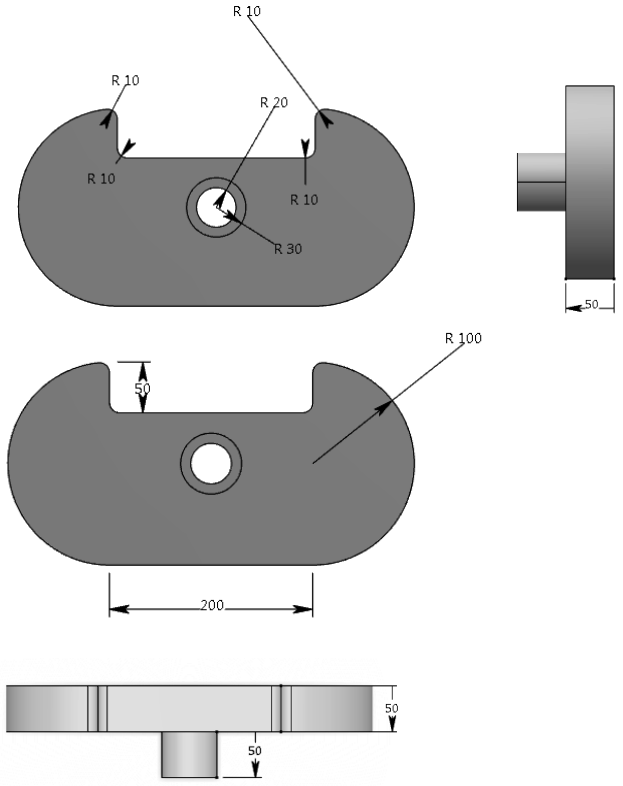
Create the following solid object assuming that the supplied dimensions are in mm. In the event that some minor dimensions are missing estimate them based on the overall scale.

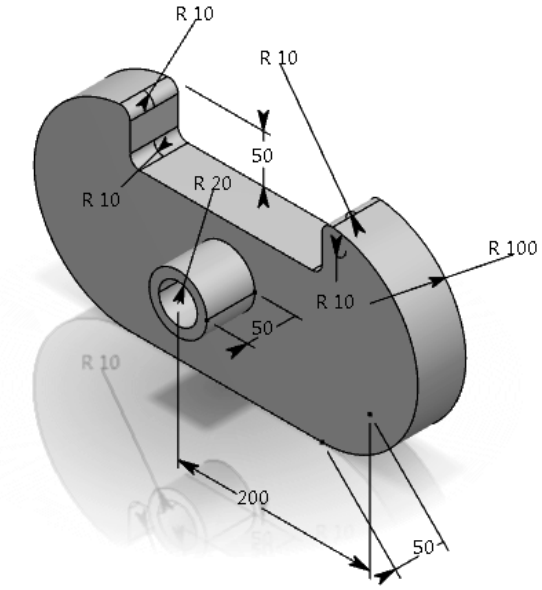




Exercise 2:

Create the following solid object assuming that the supplied dimensions are in mm. In the event that some minor dimensions are missing estimate them based on the overall scale.




R 10
R 10
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R 20
R 10
R 100
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