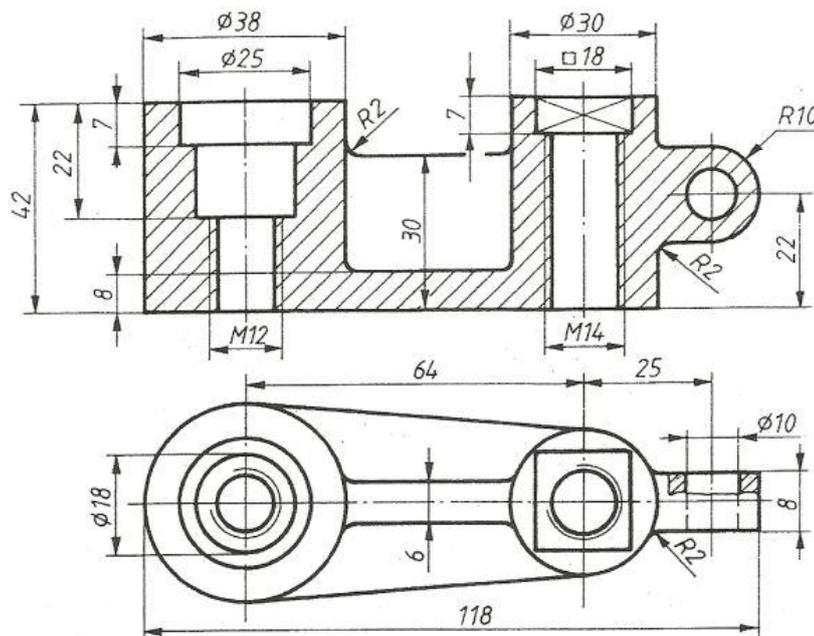


CAD-CAM-CAE Example

example title:	Multi-axis component modelling(CAD)
example number:	ÓE-A10
example level:	basic - medium - advanced
CAX system:	CATIA V5
Related material part with TÁMOP	CAD
Job Description:	Create Cad model of a Multi-axis component in CATIA v5.

1. The task

Create a CAD model in CATIA v5 from the figure!

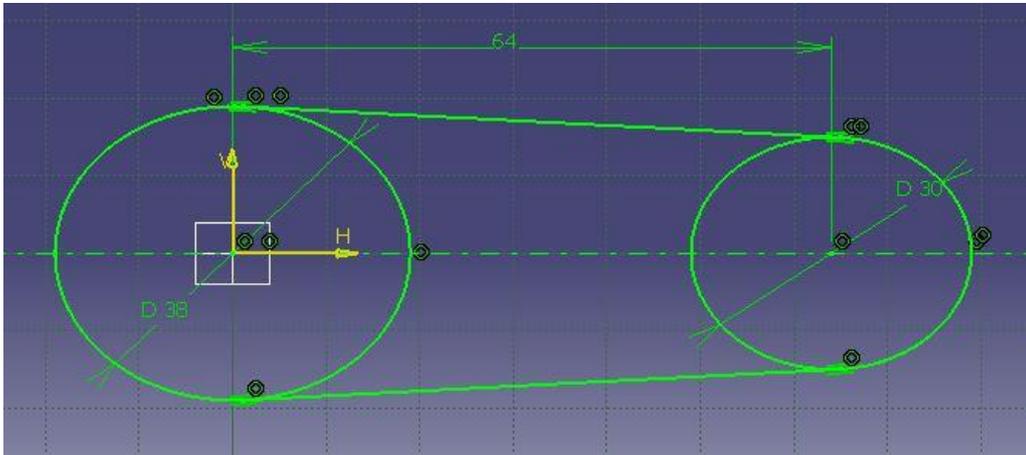


2. The solution steps:

Start the **Part Design** modul in CATIA: **Start**→**Mechanical Design**→**Part Design**
Write a name in new Part window then pres **OK** button.

Select the XY plane so that we click between the X axis and Y axis then click to the **Sketch** icon . Create a line on the horizontal axis with **Line**  command. Draw a circle in the centre of coordinate system. Draw another circle on the horizontal axis. Click to the **Constraint**  icon then click to the circle and anywhere. Double-click on the dimension

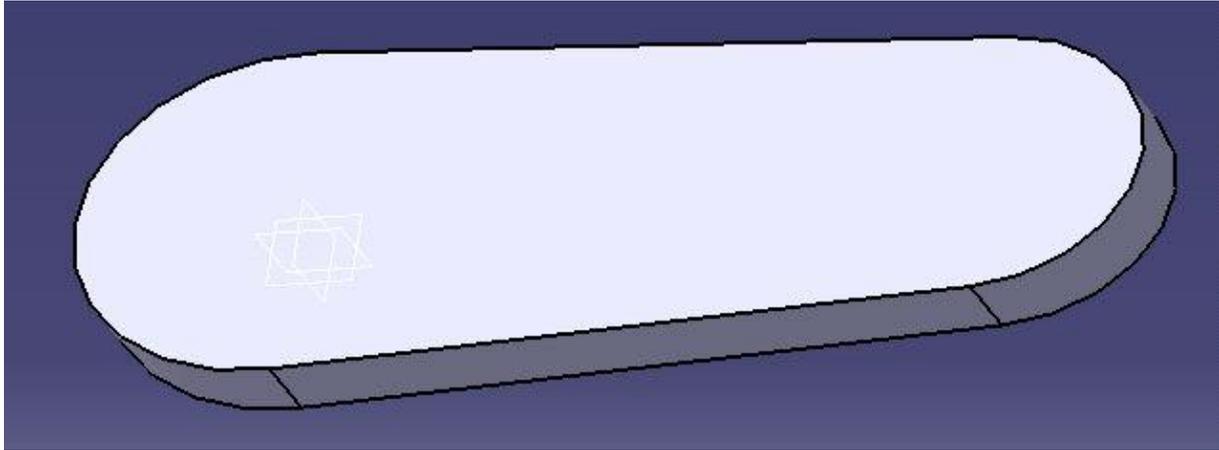
text and write the exact size. Click to the **Line icon** to draw a line. Draw two line between the two circles. See the picture. Click on the **Constraint** icon and click to the line and one of them circle. Then click right mouse button. A new menu is opened. Here select the Tangency.



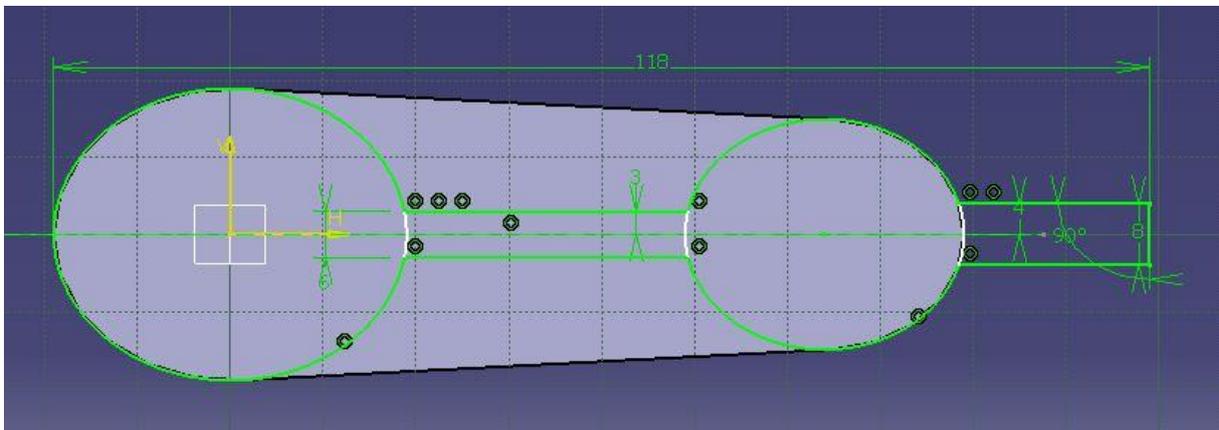
Next step is to click on the Quick Trim Icon . Select the arc you wish to be deleted from Circle. When you are done with the profile then click on the **Exit workbench**  icon to exit.



Select Sketch.1 as the profile to be extruded. Click Pad  icon. The Pad Definition dialog box appears and the application previews the pad to be created. Enter 12 in the Length field to increase the length value.



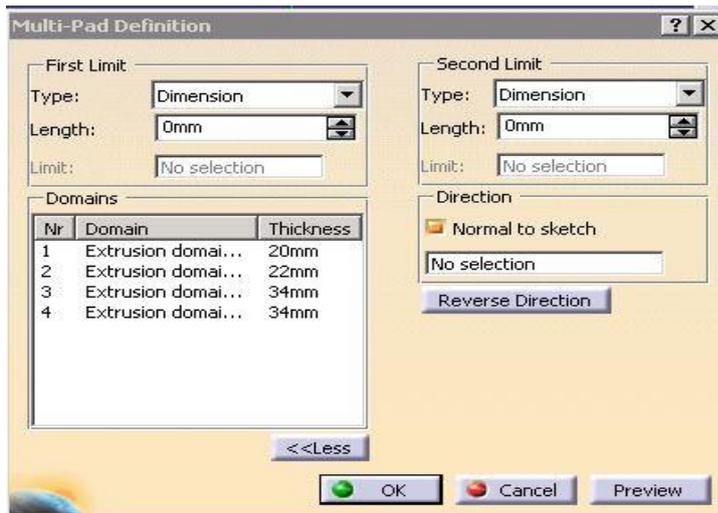
The next step is to use the following sketch (top view is essentially a profile of the holes not included):



Similarly to the previous Sketch of the two circles fit the rounded profile in the previous Sketch in form, we have this command **Coincidence** profile and clicking on the circle can be done. Then connect the two circles in the middle of the horizontal lines that are parallel and ending points of the match circles, their distance from the central axis of 3-3 mm. This can be done with two lines, they are properly aligned and parallel and the intersection in order to decide that a rectangle to fit us in circles, which likewise intersects the profiles according to given dimensions, then the circles obstruction lines on the **Quick Trim**  command to cut it. As fit the current profile protruding elements. Click Exit workbench  to exit the Sketcher.

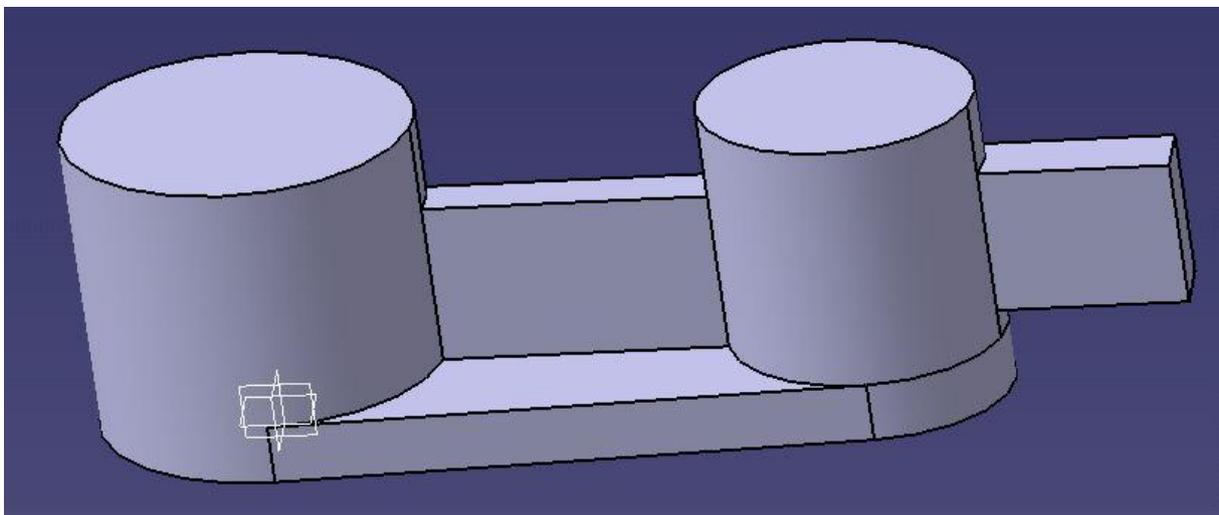
Click **Multi-Pad**  icon. Select Sketch.2 that contains the profiles to be extruded. Note that all profiles must be closed and must not intersect. In case a profile would be open, the application would not take it into account.

The Multi-Pad Definition dialog box appears and the profiles are highlighted in green. For each of them, you can drag associated manipulators to define the extrusion value.



The red arrow normal to the sketch indicates the proposed extrusion direction. To reverse it, you just need to click it.

The Multi- Pad Definition dialog box displays the number of domains to be extruded. In our example, the application has detected seven extrusions to perform, as indicated in the Domains section. Select **Extrusion domain.1** from the dialog box. Extrusion domain.1 now appears in blue in the geometry area. Specify the length by entering a value. Enter 20mm. You need to repeat the operation for each extrusion domain by entering the value of your choice. Select **Extrusion domain.2** and **Extrusion domain.3** and **Extrusion domain.4** enter 22mm and 34mm and 34 mm respectively.



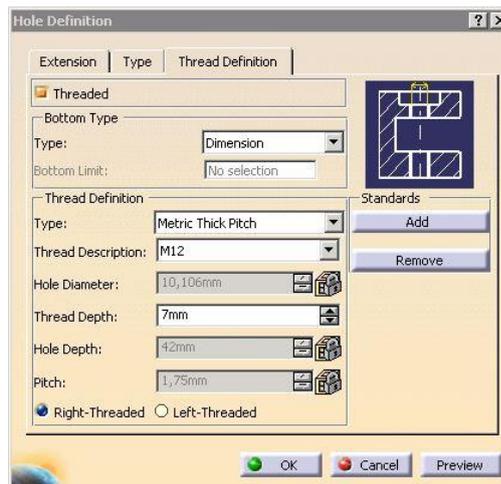
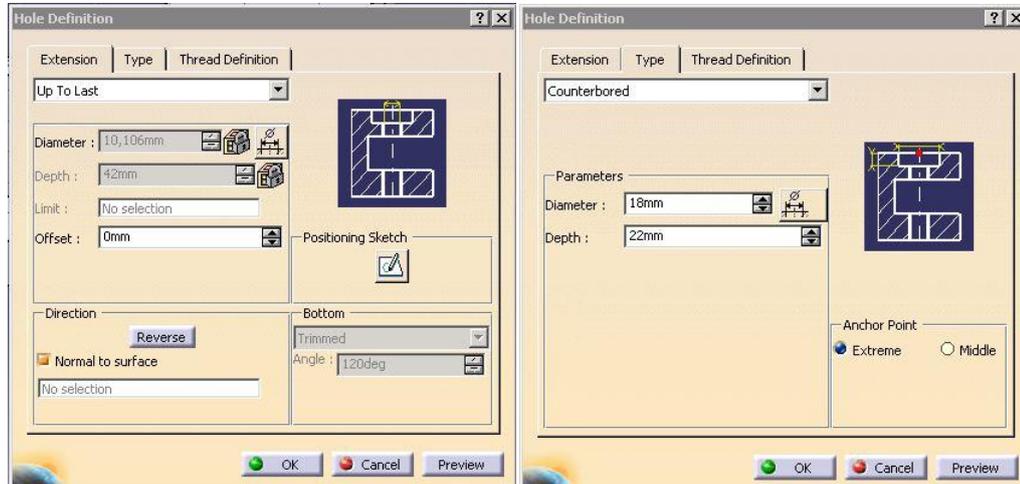
Preparation of holes

Preparation of the left three levels holes.

Click on the top of the larger cylinder body then click the Hole  command.

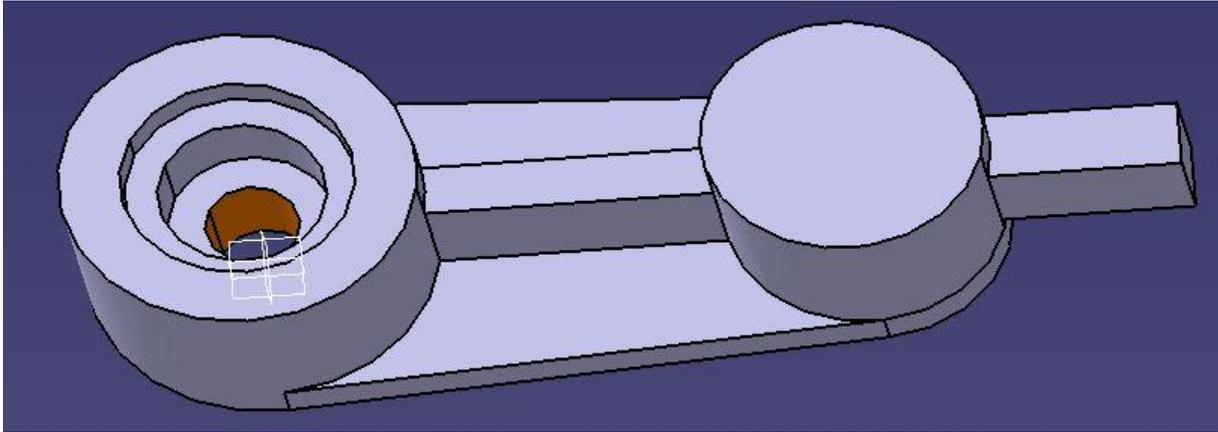
We exact location of the hole with Positioning Sketch; here it is enough if the point of the circle with Concentricity constrain, so we get centre bore.

Set the **Up to last** parameters in the extension tab. Set the **Counterbored** parameters in the Type tab. Enter 18 mm in the Diameter field. Enter 22 mm in the Depth field.



You can also define a threaded hole by clicking the Thread Definition tab and selecting the Threaded button to access the parameters you need to define. Select the **Dimension** parameter in the Bottom Type field. Select the **Metric thick pitch** parameter in Thread Definition Type field. Select the **M12** parameter in the Thread Description field. Enter 13mm value in the Thread Depth field. Check the Right-threaded settings. Click OK. The threaded counterbored hole is created.

Click on the top of the larger cylinder body then click the Hole  command. Create a new hole with 25 mm diameter and 7mm depth blind type parameters.

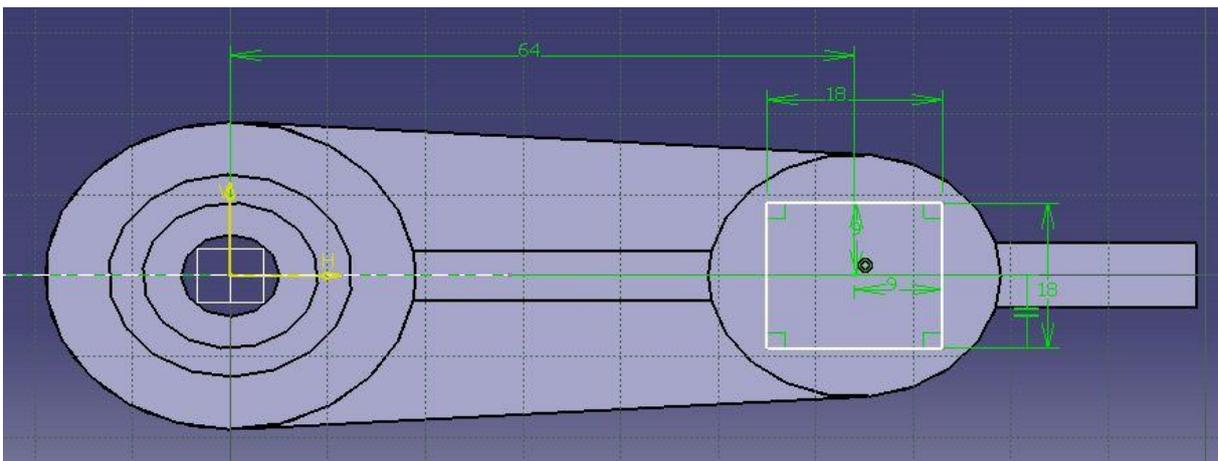


Preparation of the right hole.

Click on the top of the smaller cylinder body then click the Hole  command. We exact location of the hole with Positioning Sketch; here it is enough if the point of the circle with Concentricity constrain, so we get centre bore. Set the **Up to last** parameters in the extension tab. You can also define a threaded hole by clicking the Thread Definition tab and selecting the Threaded button to access the parameters you need to define. Select the **Dimension** parameter in the Bottom Type field. Select the **Metric thick pitch** parameter in Thread Definition Type field. Select the **M14** parameter in the Thread Description field. Set the **Support Depth** parameter in the Bottom Type field. Check the Right-threaded settings. Click OK. The threaded hole is created.

Create the last hole in the side of the work piece with Hole command.

Click on the top of the smaller cylinder body then click the Sketch icon. Click on the Centered Rectangle . Click a point in the geometry area or select an existing one. Drag the cursor to create the centered rectangle, then click when you get the size you want.

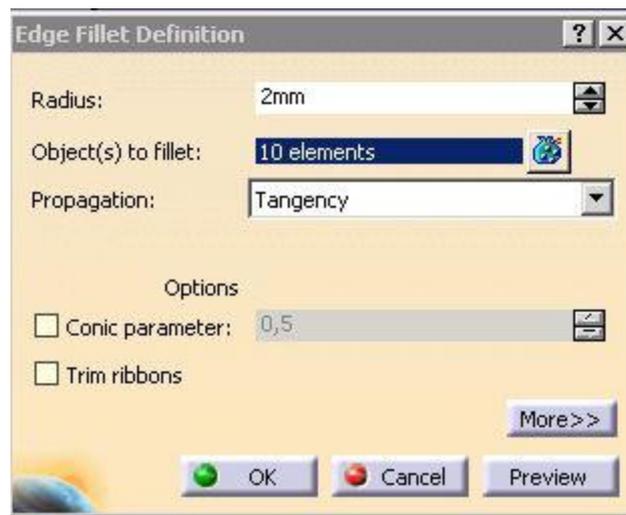


Give the full dimension and constrain to the profile. When finish then Click Exit workbench to exit the Sketcher.

Select the profile and click the **Pocket**  icon. Set 7 mm value in the Depth field. Click OK to confirm

Preparation of fillet

Click Edge Fillet  in the Dress-Up Features toolbar (Fillets sub-toolbar). The Edge Fillet Definition dialog box appears. Select ten edges. The edge selected then appears in the Object(s) to fillet field. Enter 2 mm in the Radius field to increase the radius value. The application displays the radius value. Clicking Preview previews the fillet to be created. Click OK. The edges are filleted.



More than one fillet can also specify if they are the same size, simply select the row of edges and adjust the value.

The model is completed.

