
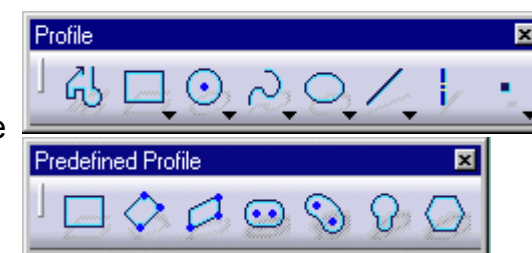


Creating a Cylindrical Elongated Hole

 This task will show you how to create a curved oblong profile as part of the sketch to be created. This curved oblong profile will be created in accordance with previously positioned construction lines. In other words, the curved oblong profile is centered on point B, starts from point C and is assigned a 27 degree angular sector.



 1. Click the Cylindrical Elongated Hole icon  from the Profiles toolbar (Predefined Profile subtoolbar).

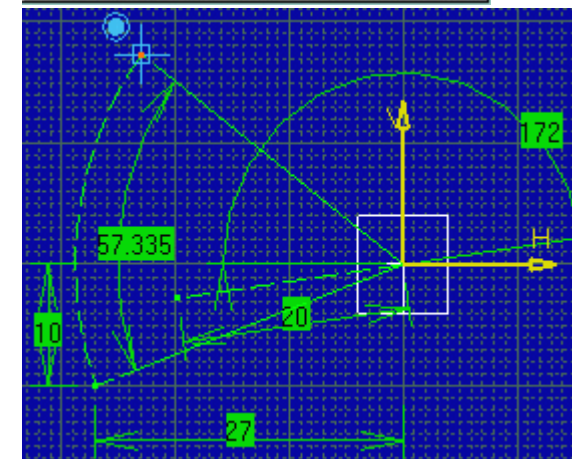




2. De-activate the Construction Element icon  from the `Sketch tools` toolbar.

3. Make sure the Dimensional Constraint icon  is still active.

4. Click point B (arc top extremity point) to define the curved oblong profile center.

 Before you click, a blue symbol  appears to illustrate coincidence between the selected points and the profile angular sector to be created.

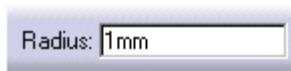


Note that if you position the cursor outside the zone that is allowed for creating a given element, the  symbol appears. In this particular case, trying to create a degenerated arc (0 deg sector) displays the  symbol.

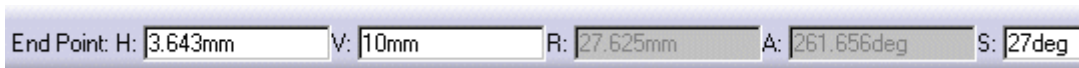
5. Click point C (extremity point of the construction line previously created).

New values and fields appear in the Sketch tools toolbar.

6. Position the cursor in Radius field, key in 1 and press Enter.



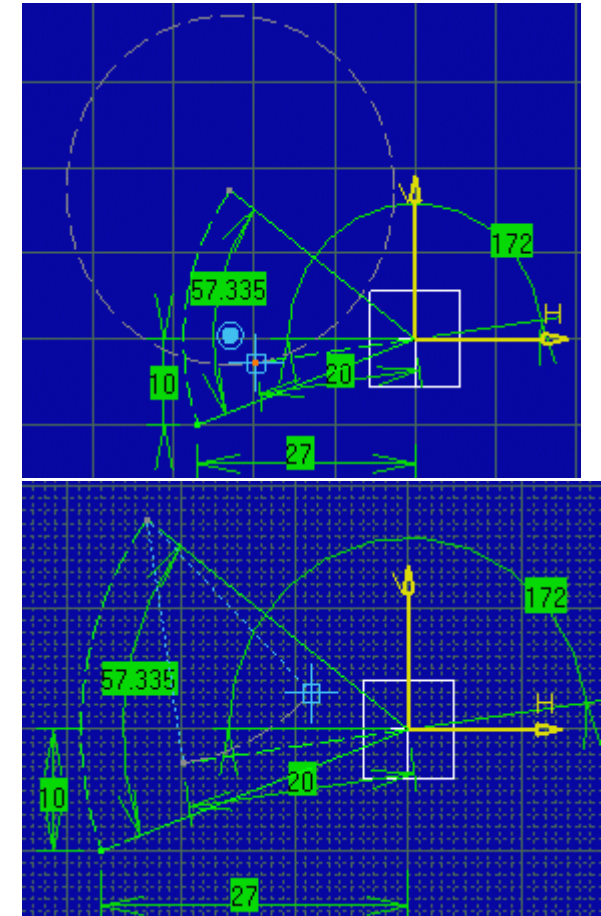
7. Position the cursor in S field (angular sector) and key in 27.




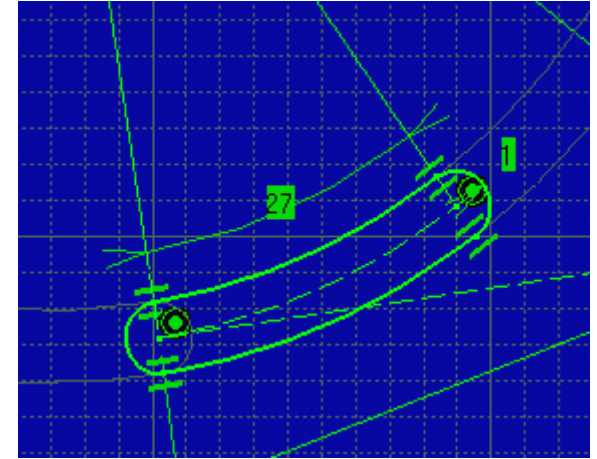
8. Press Enter.


This is the final cylindrical elongated hole:

As you could see above, geometrical elements are assigned given colors. This is to make



 the sketch geometrical configuration more easily understandable. Purple means that geometry is over-constrained. Green means that the geometry is iso-constrained (cannot be moved).



 **At this step, we strongly advise that you save the cylindrical elongated hole.**

