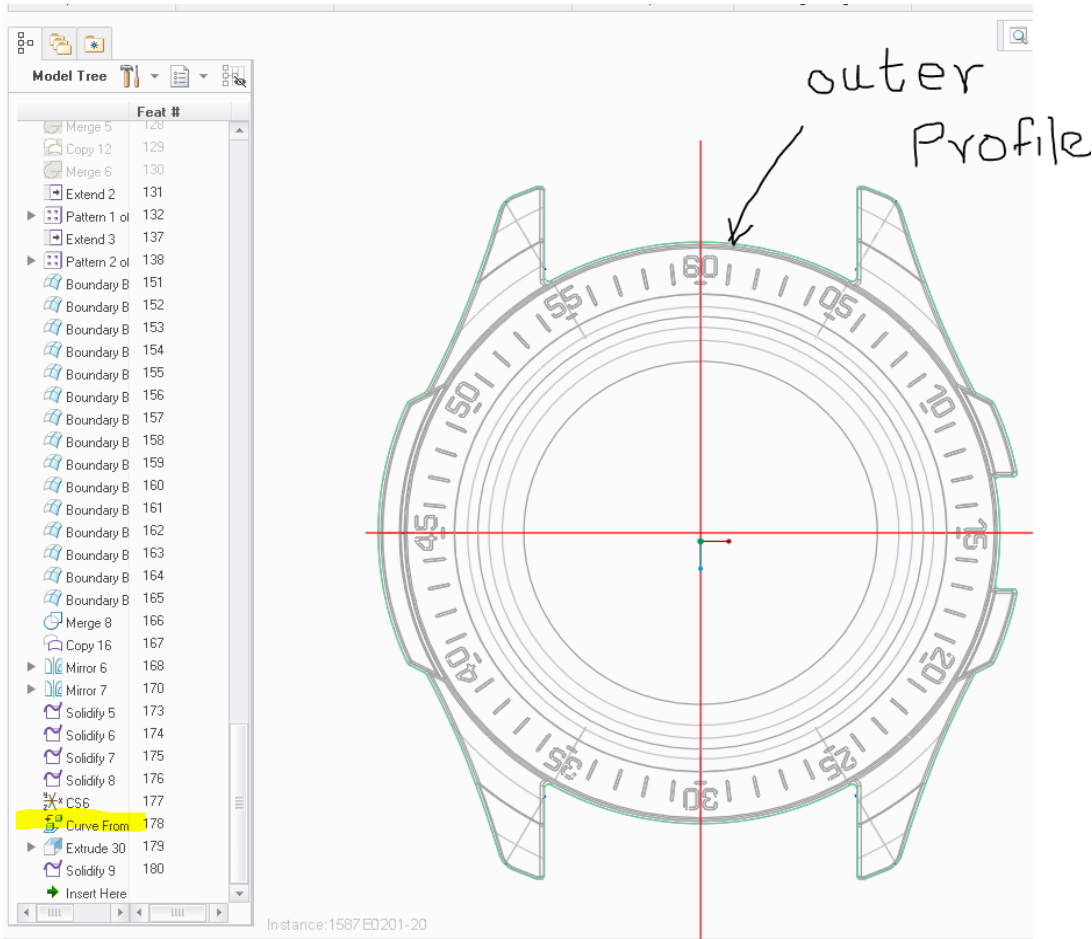
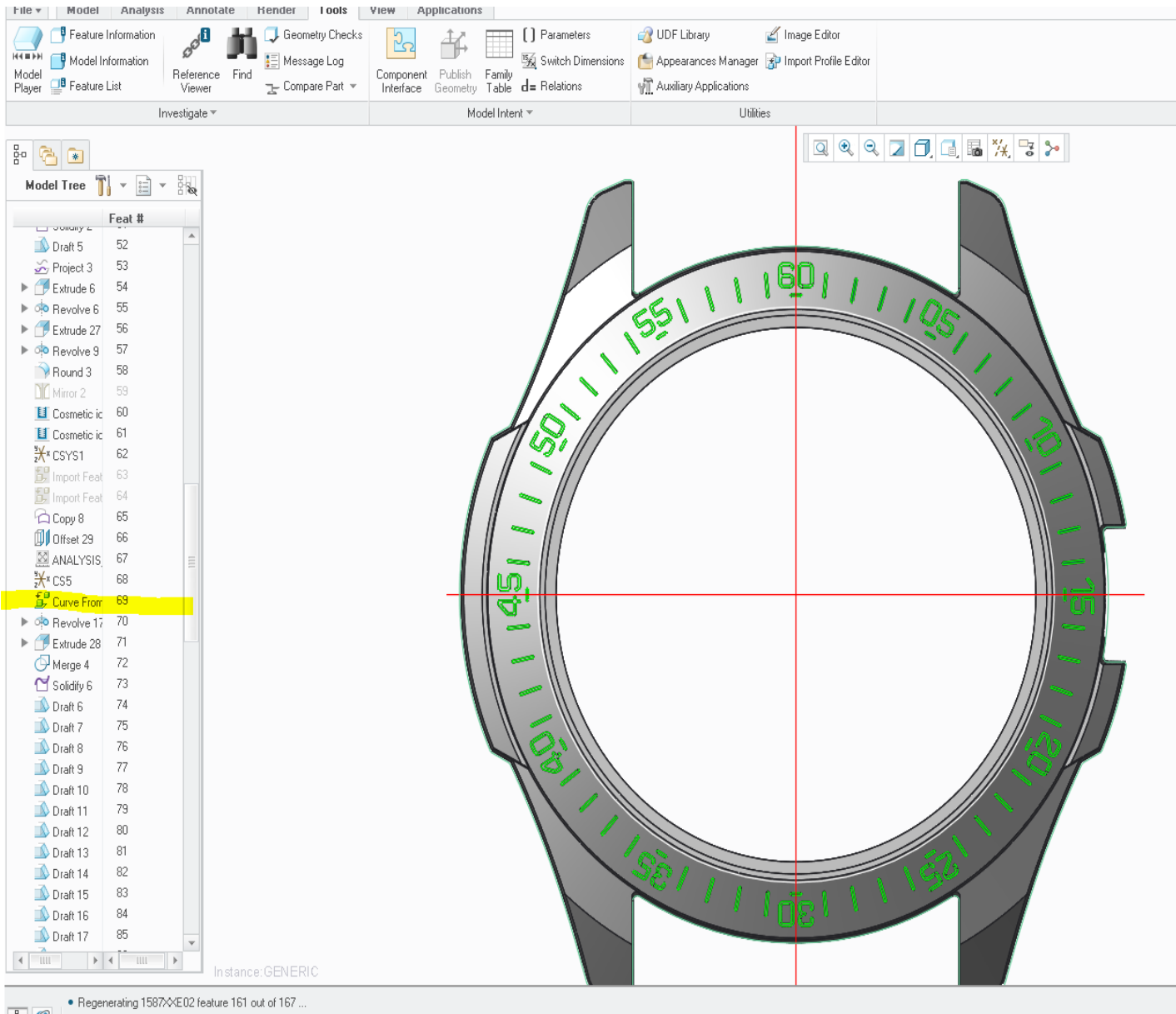


Problem in working with Export file (DXF) in Creo-3

Below Image from Creo 2.0

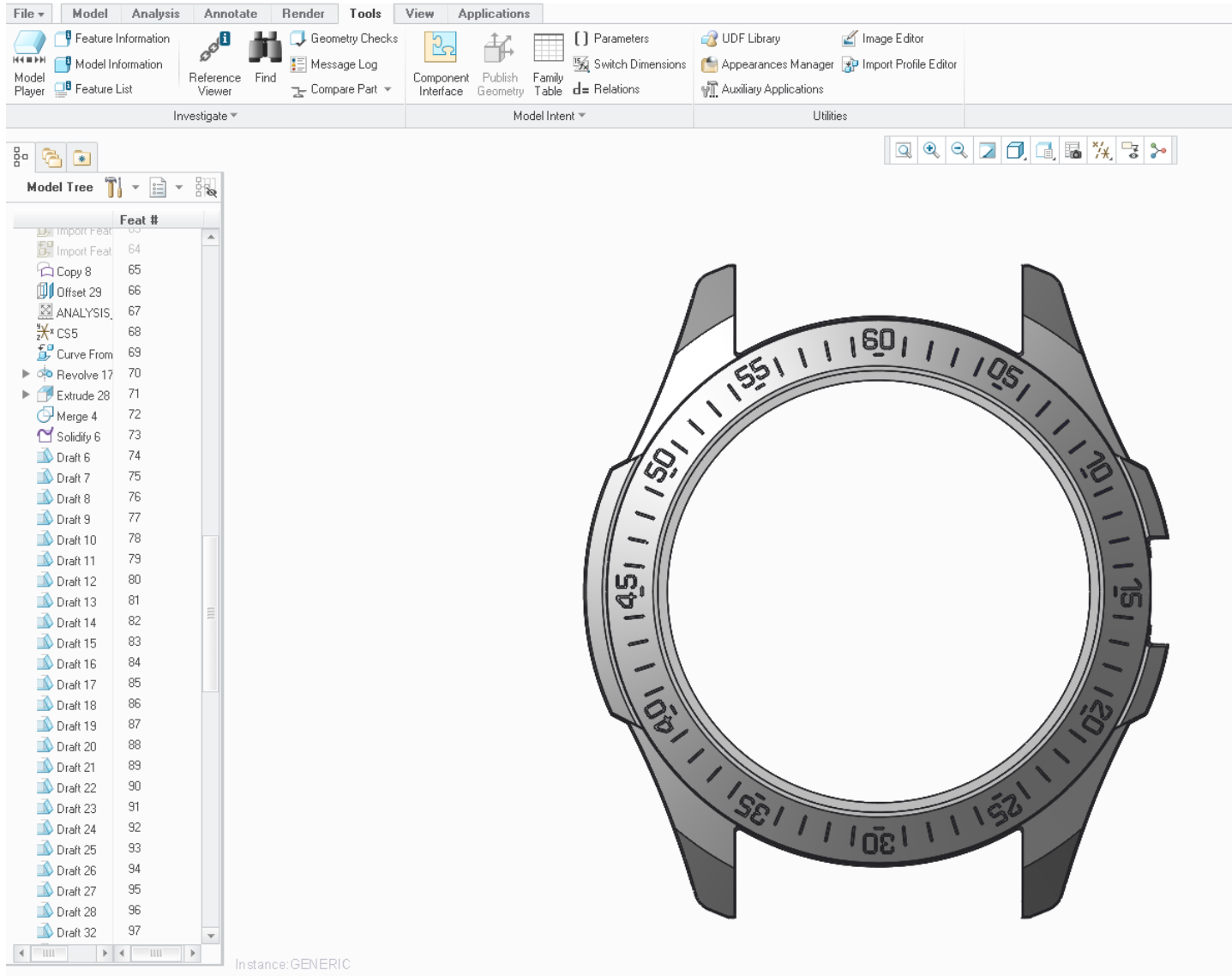


- Refer Below CREO 3.0 ISSUE.

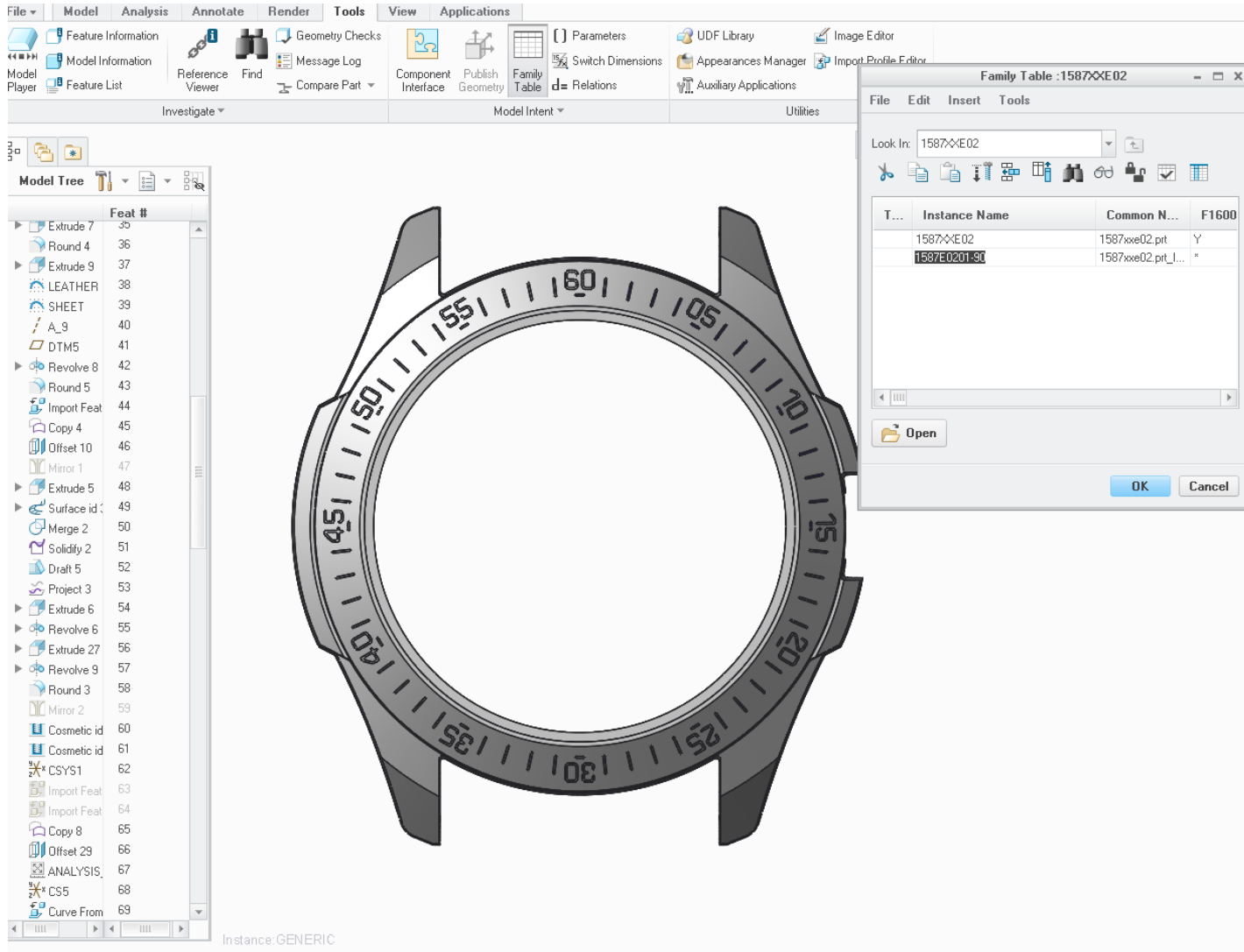


The highlighted Curve (in yellow colour) is the DXF which I have imported

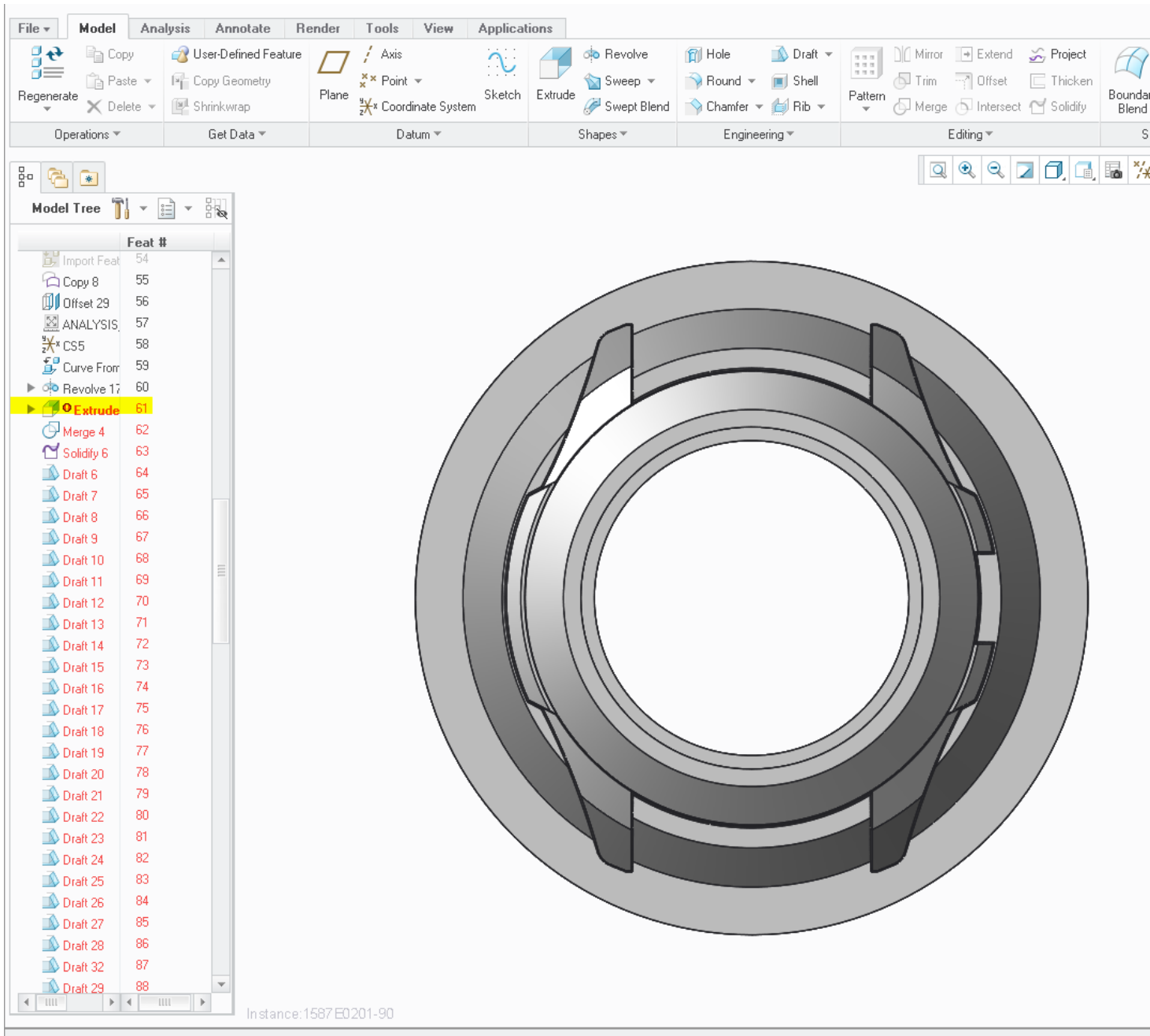
The green colour in the part is the imported file



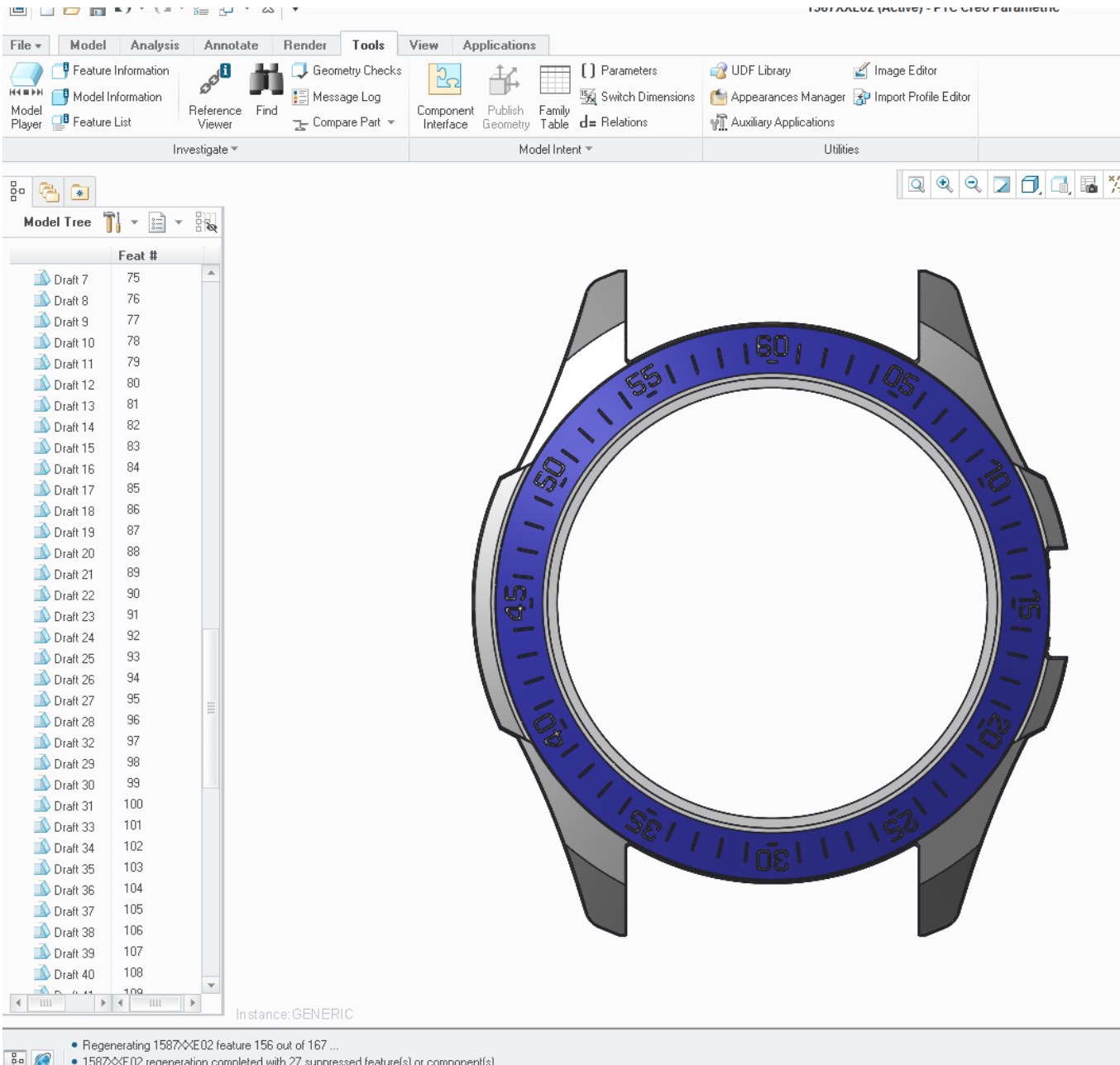
By using the imported feature, I have used “project (use edge)” command in sketcher mode to make the letter font as mentioned in the case.



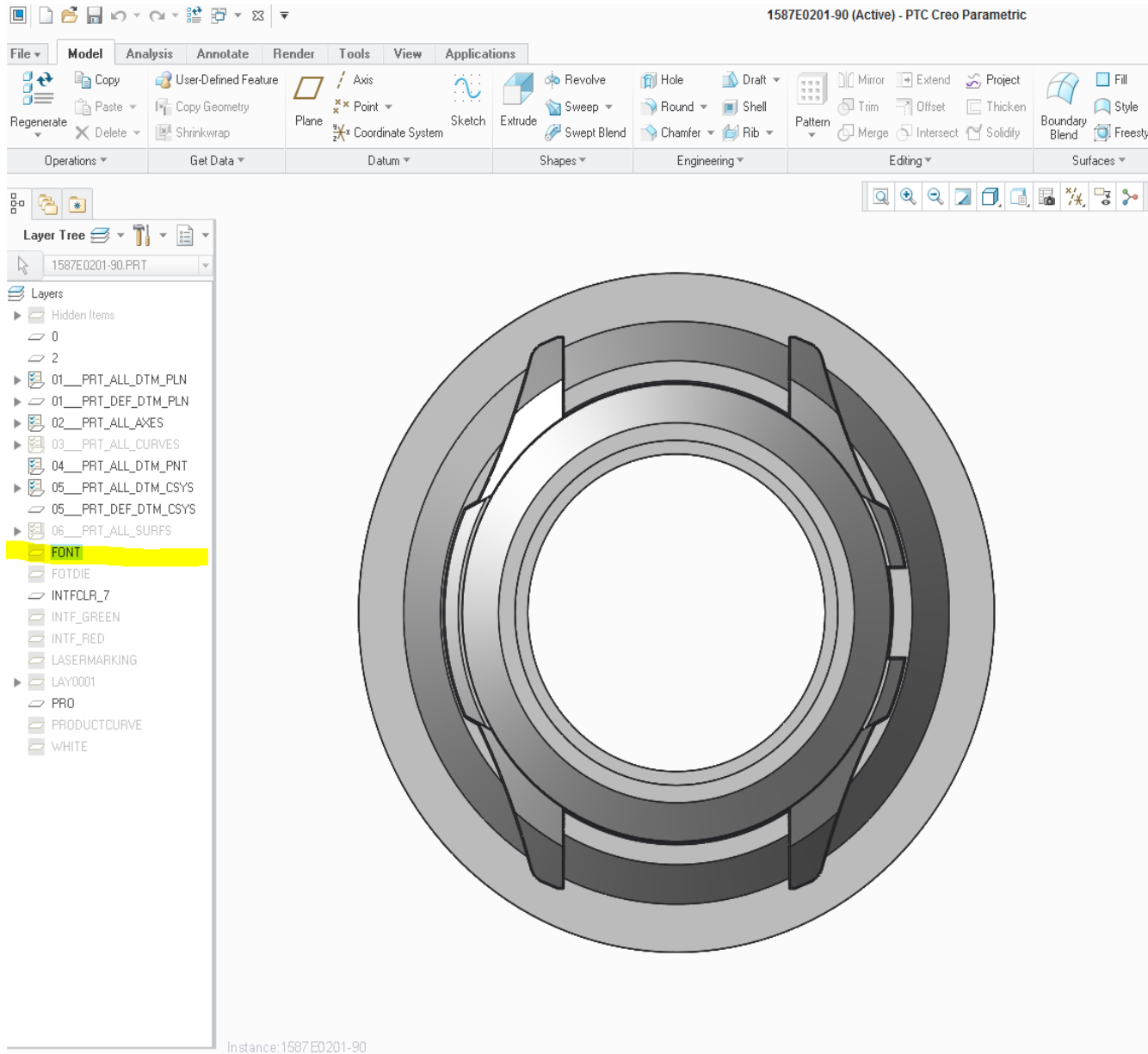
Then I have made a family table and created a child.



The highlighted feature (in yellow colour) is the extrude command, which I used “use Edge (project) “command is getting failed.



Then I tried to open the parent file. It is also getting failed and turned to blue colour as mentioned in the part



The highlighted Layer (in yellow colour) is the layer which I created and for the particular export feature.

After failure, when I tried to open the import feature.

Surprisingly It is not present in *the parent part* and **vanished...** (disappear) during working...which I could not get back

Due to this problem, the feature created after the import feature got fail.

At Titan we do our tools by using family table concept. As the parent part itself is getting failed, all the children in the family table also fails and creates problem up-to manufacturing stage.

Vanishing of imported feature from DXF is a serious problem and it is reflecting in similar models causing a “total stoppage” of our work.

This issue was not there in the earlier versions of Creo and Pro-E. And we have done enormous parts using the software. If it is getting failed in the new version of Creo, it will create a big disturbance in our working structure.

So such issues to be cleared without any delay with a fast response and rectification.

Thank You

And do the needful.....