

Creo Parametric Tutorials

Tutorial 1 : Learning the Creo Interface

Tutorial 2 : Extrusions, Sketching and Cuts

**Tutorial 3 : Revolves, Holes and
Feature Modification**

**Tutorial 4 : Complex parts, Shells, Ribs, Rounds,
Chamfers and Datum Planes**

**Tutorial 5 : Thin solids, Mirror, Pattern,
Datum Axes/Planes and more**

Tutorial 6 : Sweeps, Blends and Helical Sweeps

Tutorial 7 : Assembly

Tutorial 8 : Engineering Drawings

**Part
Modeling**

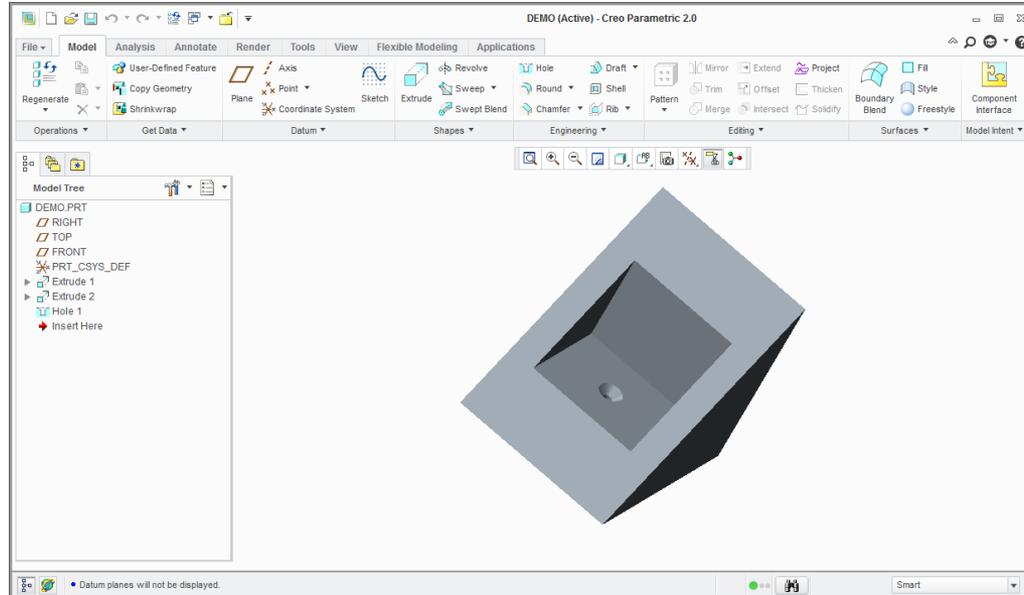


Tutorial 1

Learning the Creo Interface

Outcomes:

- Familiarity with the Creo work environment , most effective keys, shortcuts and options.
- Ability to access part files, manipulate and manage them and save them in different formats.



Homework 1

- Open the file named **Homework1.prt** and change the orientation of the part to look similar to that shown in the figure below
- Add this new orientation to the saved views list and name this view as: **Your_Name_View**
- Save the file and delete its old versions
- Change the name of the file to: **Your_Student_Number.prt**
- Save an image (JPEG file) of the part showing the new view and name it as **Your_Student_Number**

