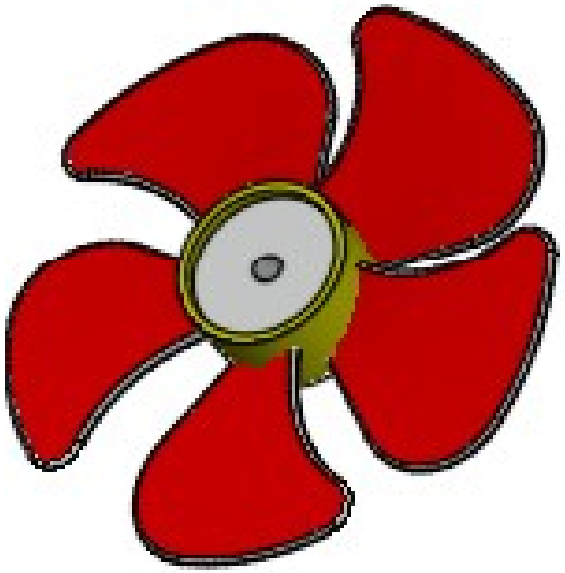
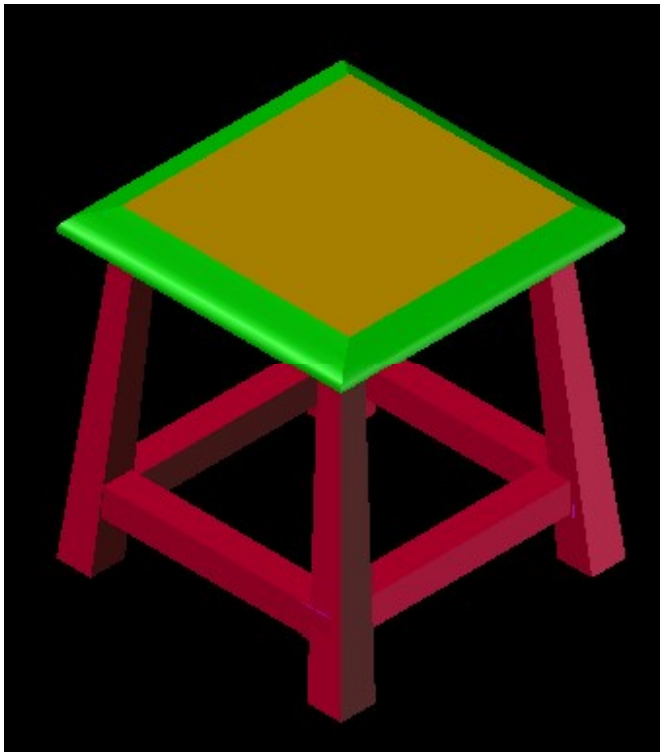
21MES102L
Engineering Graphics and Design

E12 Solid Modeling Using Features





Significance in Solid Modeling

Manufacturing of a product is the main activity in engineering profession. The design of a product may start with n number of trial designs. A knowledge in Solid modeling helps the designer to

- Realize his ideas.
- It eliminates the need for building expensive prototypes and makes the product development cycle shorter
- Presenting the results of the design in a useful form for decision making and possible improvement.
- Presenting the improved model for evaluation.

Primitives based design are

Shapes based designs

Model is developed using

- **Box**
- **Wedge**
- **Cylinder**
- **Cone**
- **Sphere**
- **Torus**

Feature based design are

2D Profile based designs

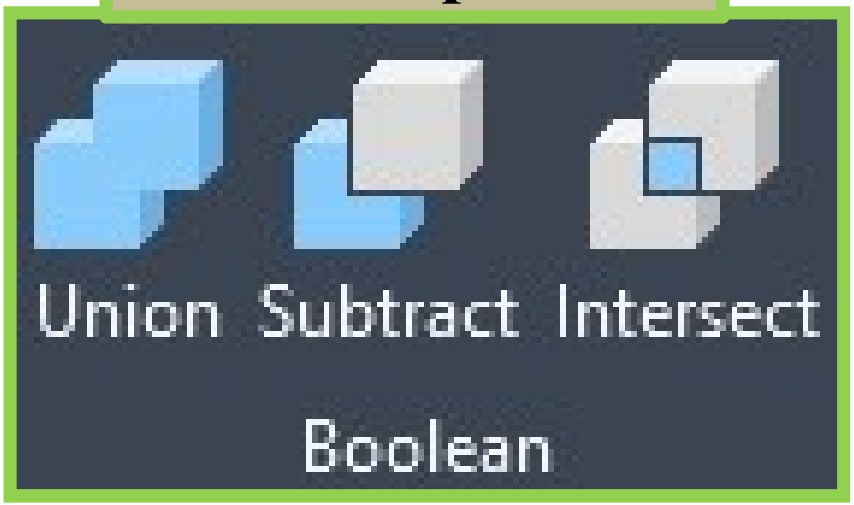
Model is developed using

- **Extrude**
- **Loft**
- **Revolve**
- **Sweep**
- **Shell**

Feature Based Tool bar

Solid Editing Tool bar

Boolean Operators



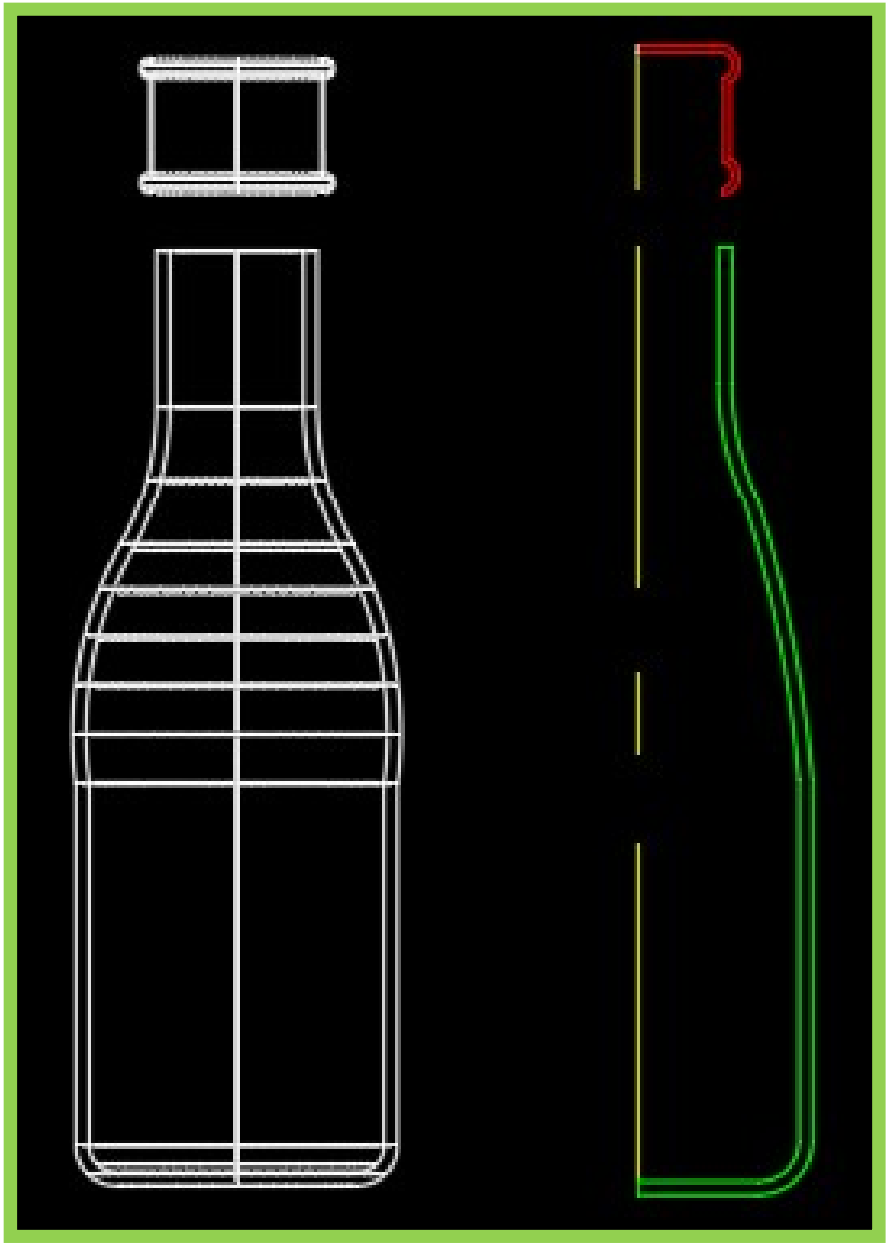
Features Based Design

- Helps the user for understanding about the creation of the solid.
- It increases the accuracy and productivity of designer.
- It allows design alterations to be made easily.
- It offers better drawing visualization through colors.
- It improves the quality of drawings produced.

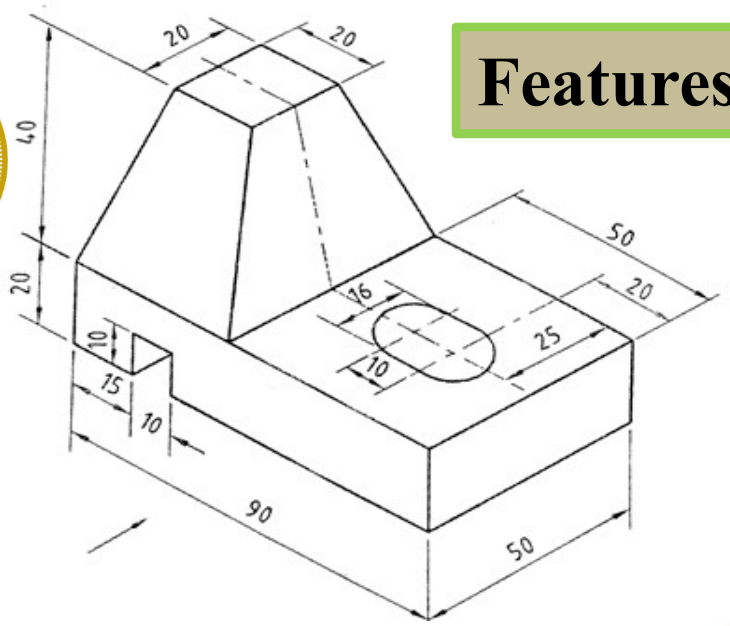
Profile based solids: The following steps are performed

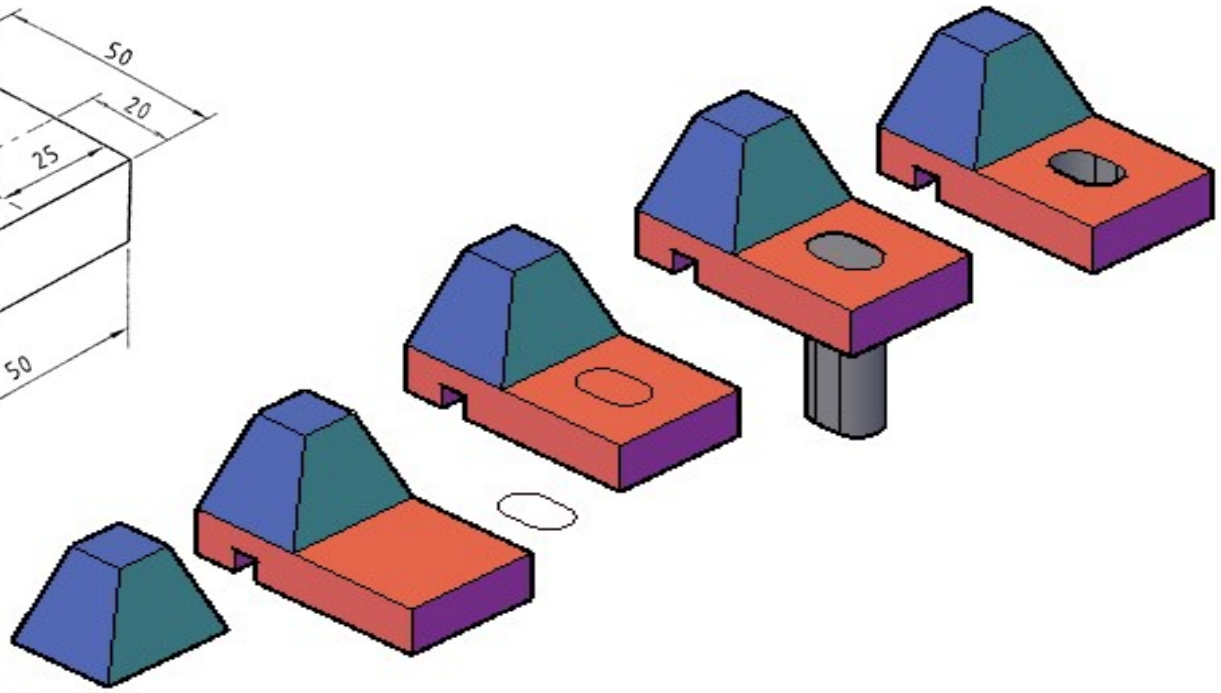
- Create **2D** closed profile in **XY** plane using **2D** commands.
- Generate surface on this profile by using **REGION** command.
- Use **EXTRUDE** command to provide height to this profile in **Z** direction OR
- Use **REVOLVE** command to revolve this profile about **X** or **Y** axis of **UCS** and thus the solid will be created.
- Finally use **SHADEMODE** command to get real look of solid.

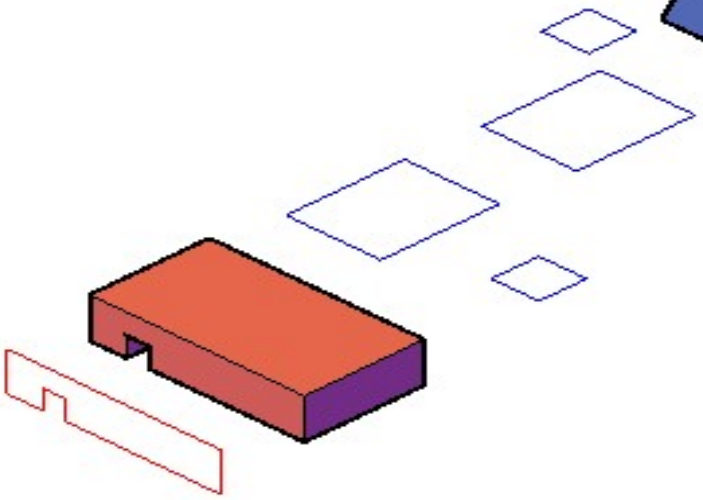
Features Based Design using Revolve after creating a Profile as shown in the figure



Features Based Design using **Extrude** & **Loft**

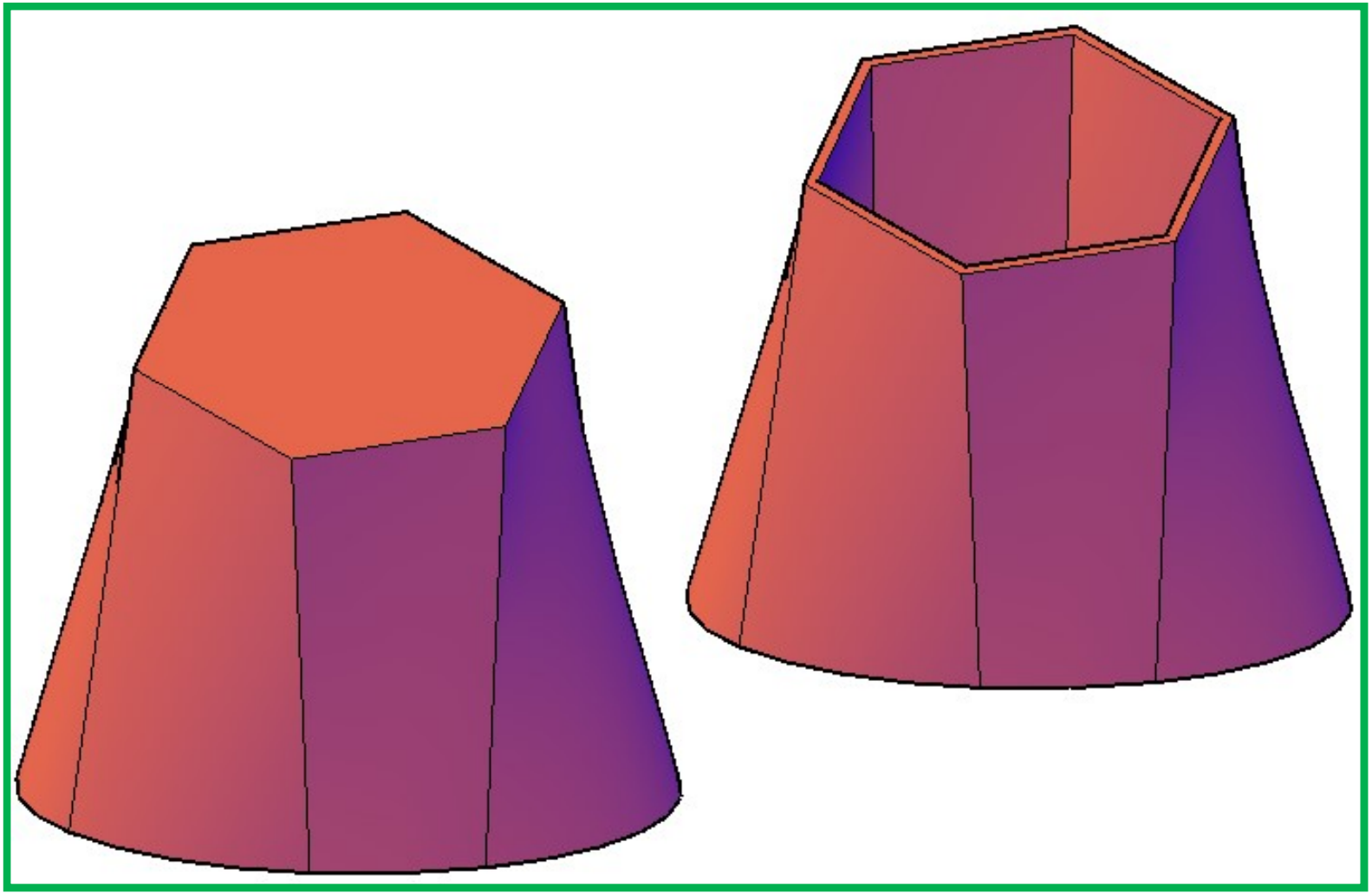


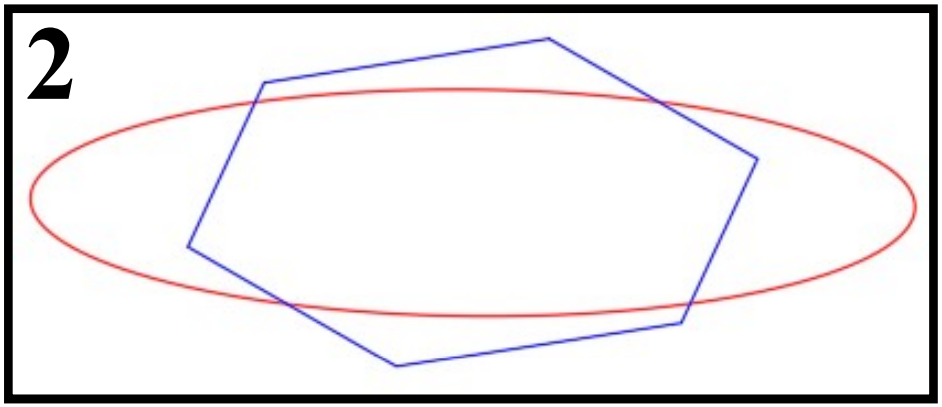


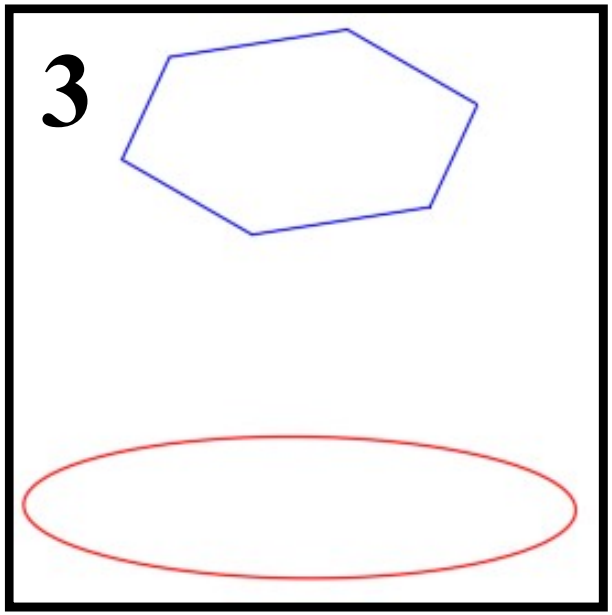


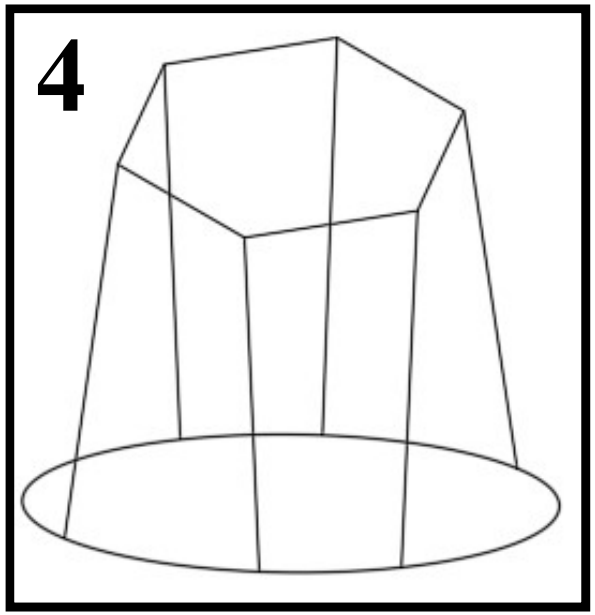
- Generate the **2D Profile** as per dimensions
- **Extrude** & **Loft** are the **Feature** based commands used with Boolean Operations to complete the model

Features Based Design using LOFT & SHELL
after creating a **2D** Profile

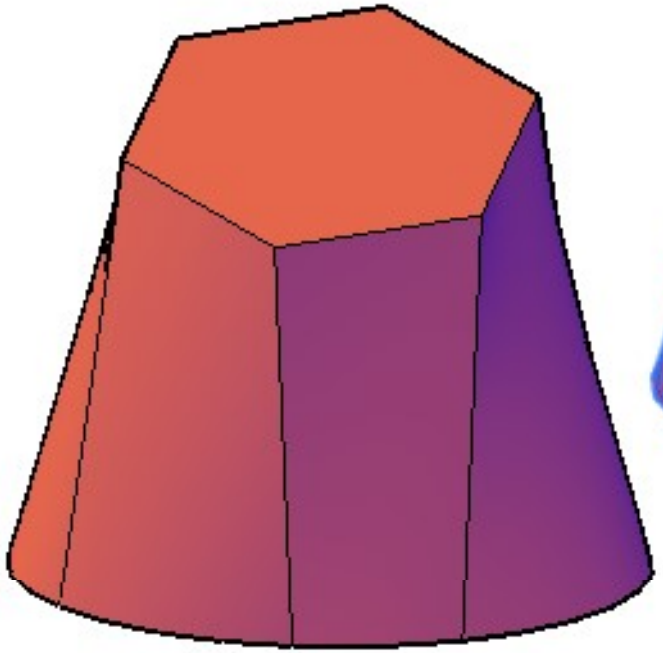


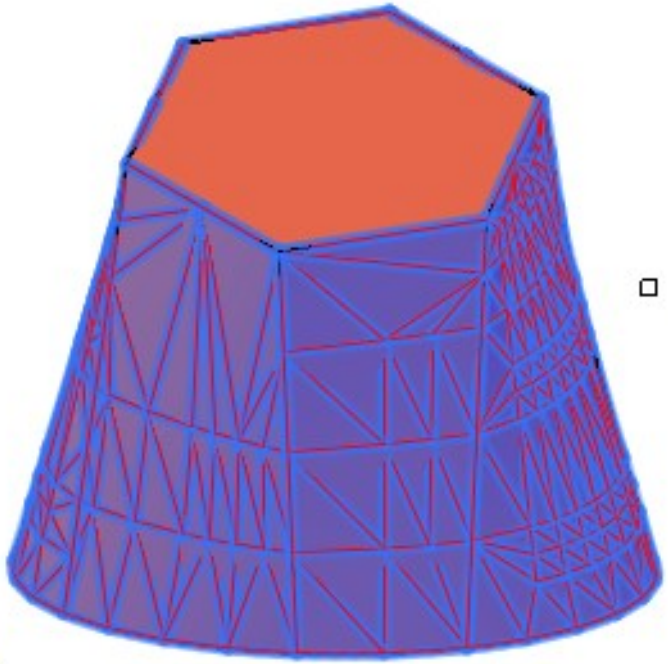


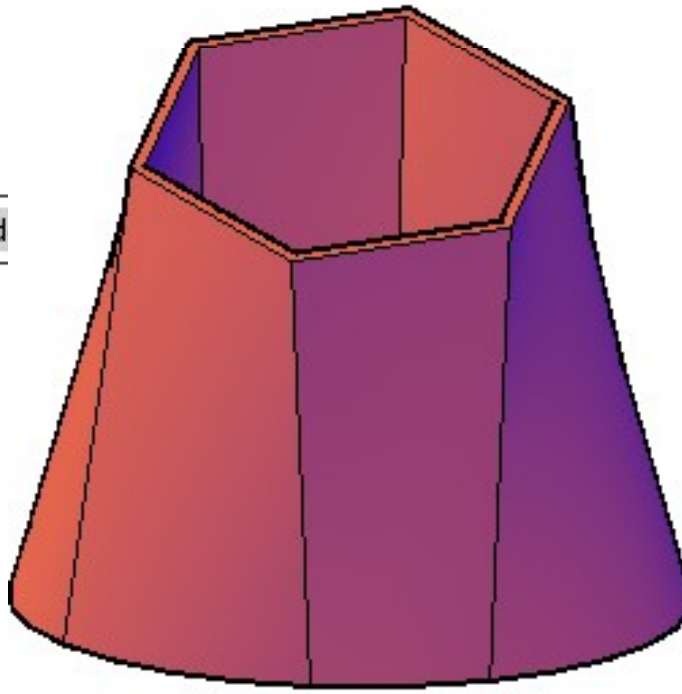




- Draw any **2D** profiles
- Move & Merge with Geometric Centers
- Move any one profile in vertical up or down direction
- Select **Loft** command & select the profiles in any order



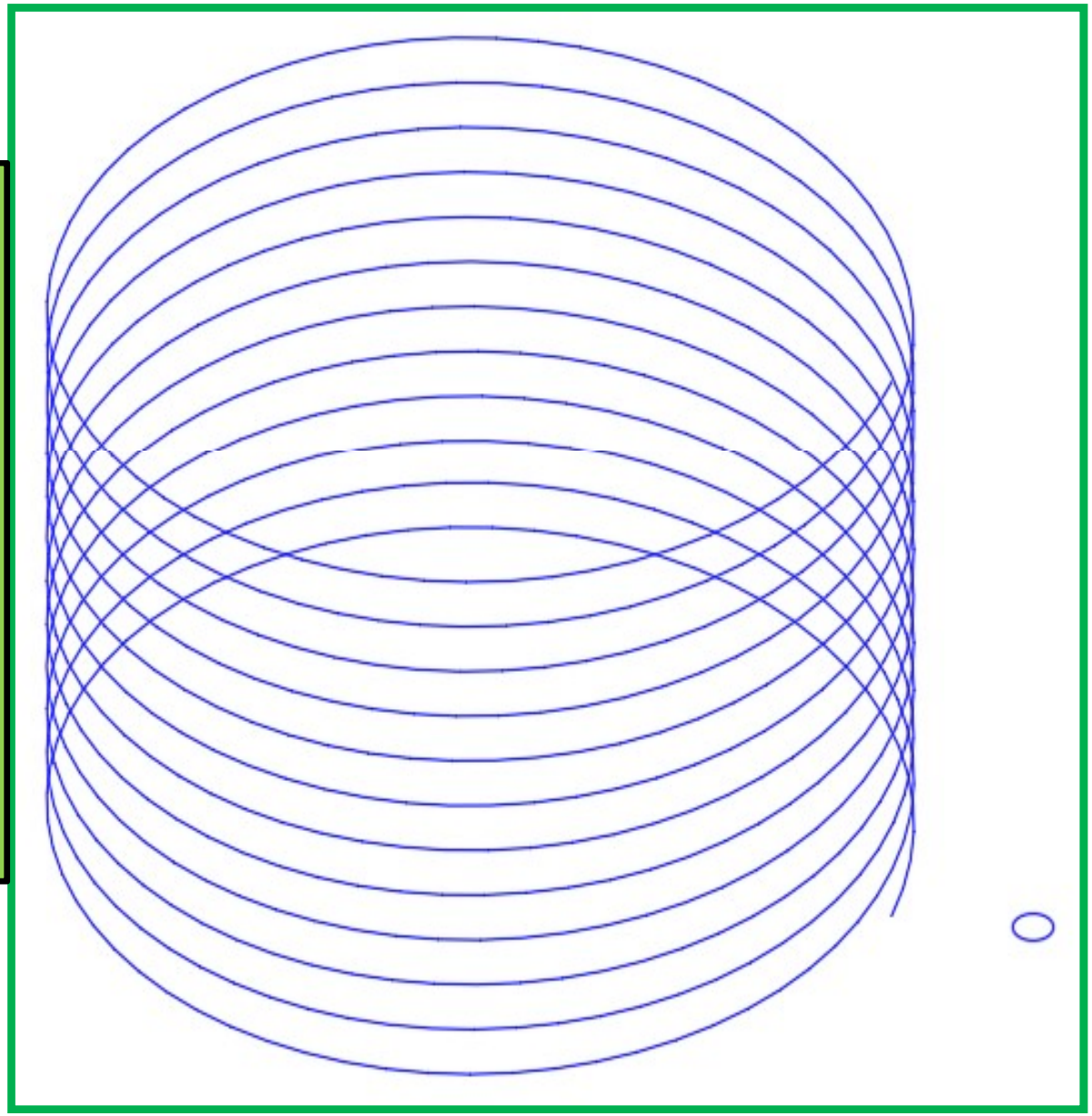


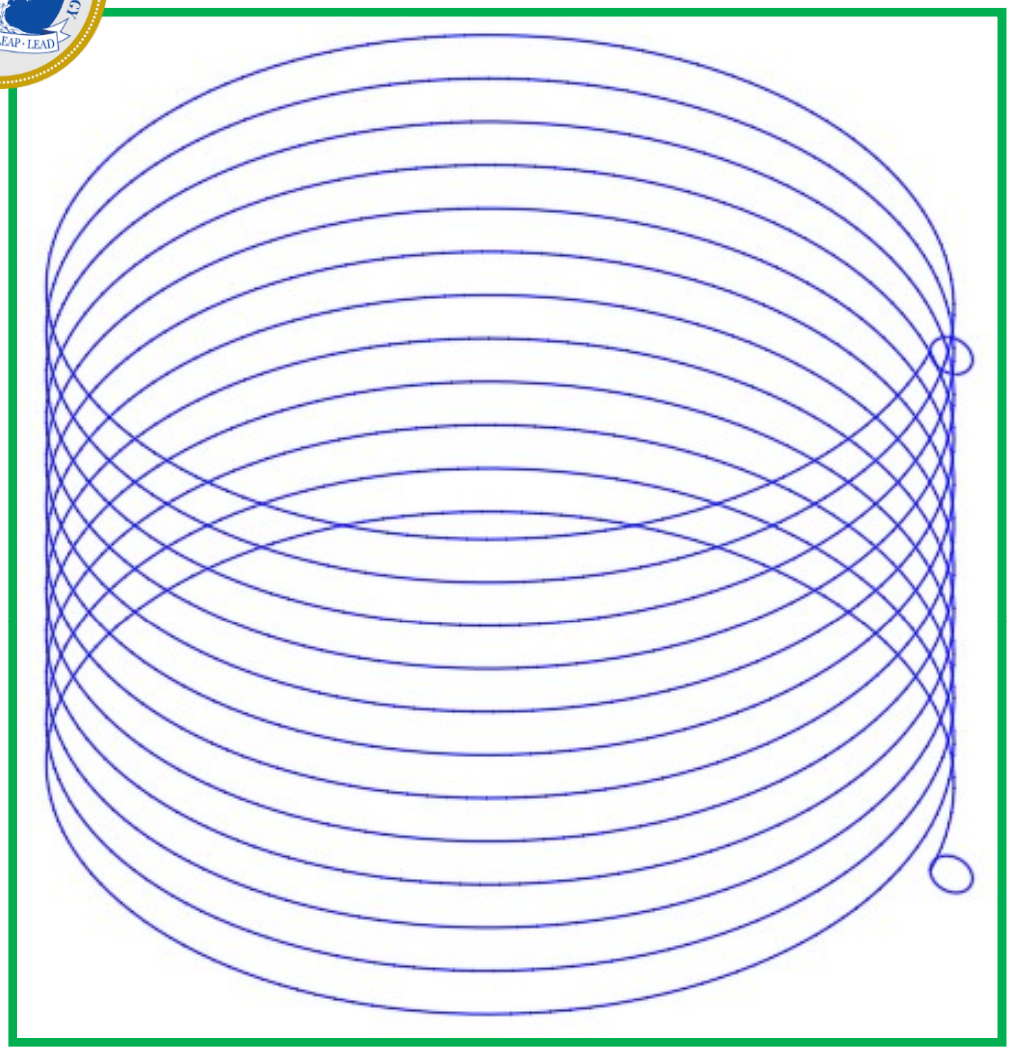


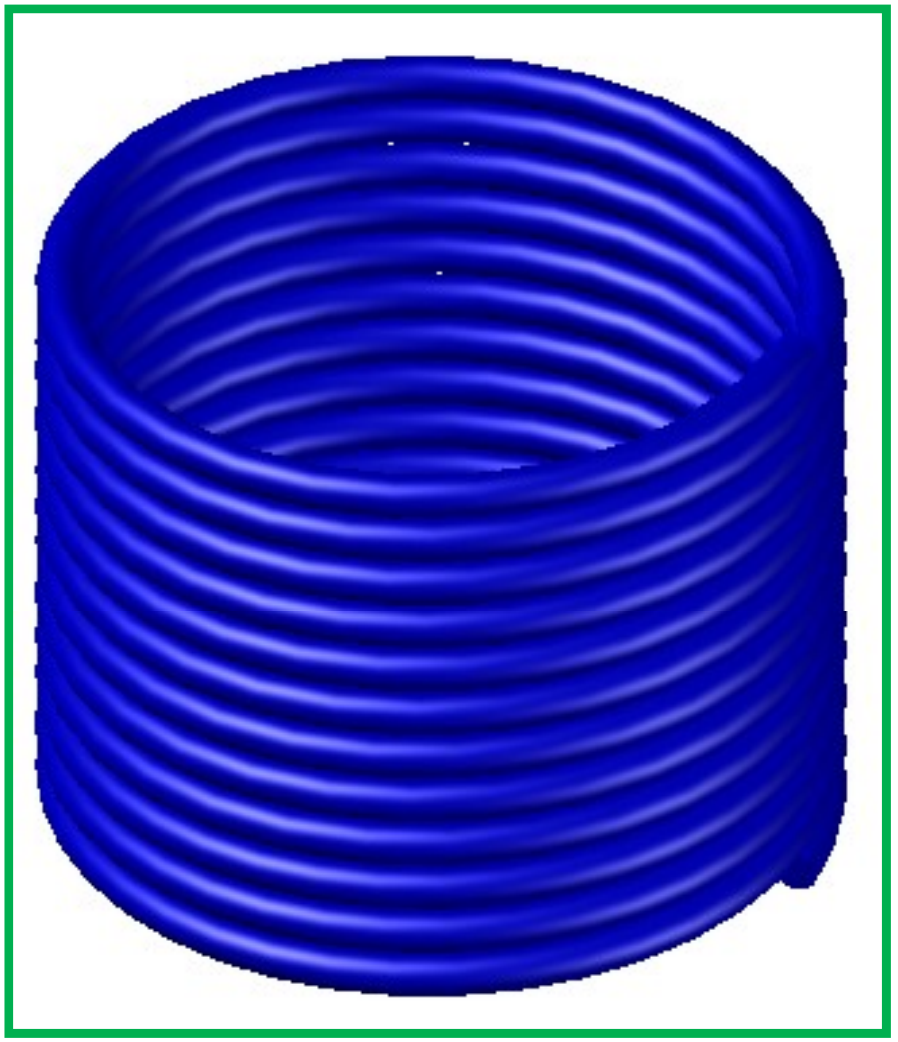
- Select **SHELL** command
- Select the **Solid** & the respective **Face** of the solid
- Enter the Thickness value Say **4** or **5**
- Press enter to see the Solid with Shallow

Features Based Design using **Sweep**

- To Create a Helical Profile
- Select Helix Command
- Specify **Base Radius 120**
- Specify **Top Radius 120**
- Specify Height 150
- Specify No of Turns 12







- Select **SWEEP** command
- Select the **Circle** to Sweep Along the Path **Helical coil**

REFERENCE BOOKS

- JEYAPOOVAN T, “ENGINEERING GRAPHICS AND DESIGN”, 2023, Vikas Publishing House Pvt Ltd,
- K.V.NATARAJAN, “Engineering Graphics”, 2015, Dhanalakshmi Publishers.