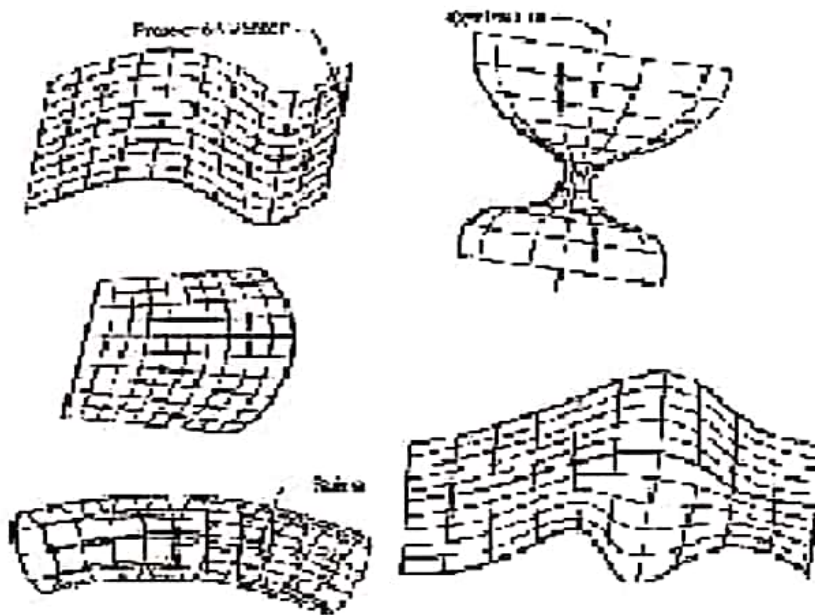


### Surface modeling:

A surface model of an object is more complete and less ambiguous representation than its wire frame model. It is also richer in associated geometric contents, which make it more suitable for engineering and design applications. Surface model takes one step beyond wire frame models by providing information on surfaces connecting the object edges. Creating a surface has some quantitative data such as point & tangents & some qualitative data like desired shape & smoothness. Choice of surface form depends on type of application.

## 3-D Surface Models



### Solid modeling:

A solid model of an object is more complete representation than its surface model. It is unique from the surface model in topological information it stores which potentially permits functional automation and integration. Defining an object with the

solid model is the easiest of the available three modeling techniques. Solid model can

be quickly created without having to define individual locations as with wire frames.

The completeness and unambiguity of solid models are attributed to the information

that is related database of these models stores ( **Topology** **It determine the**

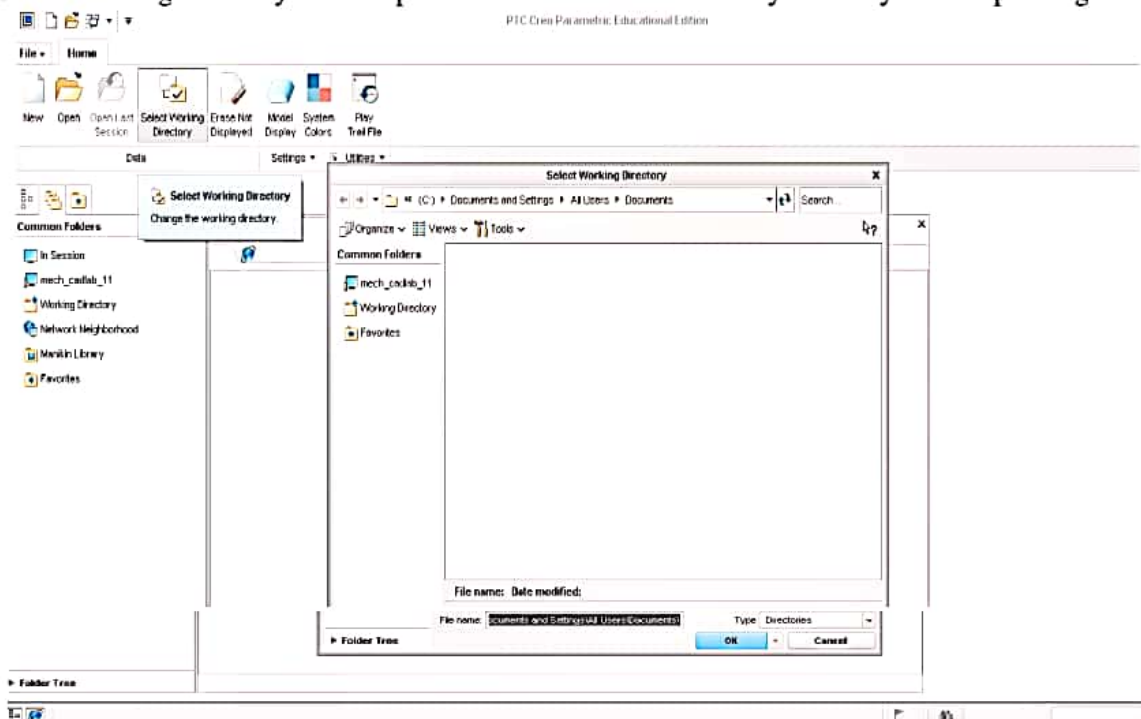
**relational information between objects.**) To model an object completely we need both geometry & topological information. Geometry is visible, whereas topological information are stored in solid model database are not visible to user. Two or more primitives can be combined to form the desire solid. Primitives are combined by Boolean Operations



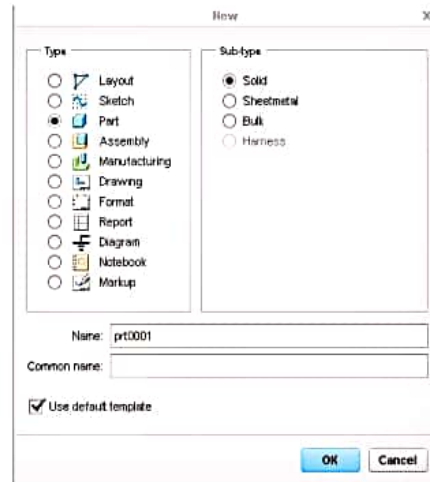
### 1. Open Creo Parametric 3.0



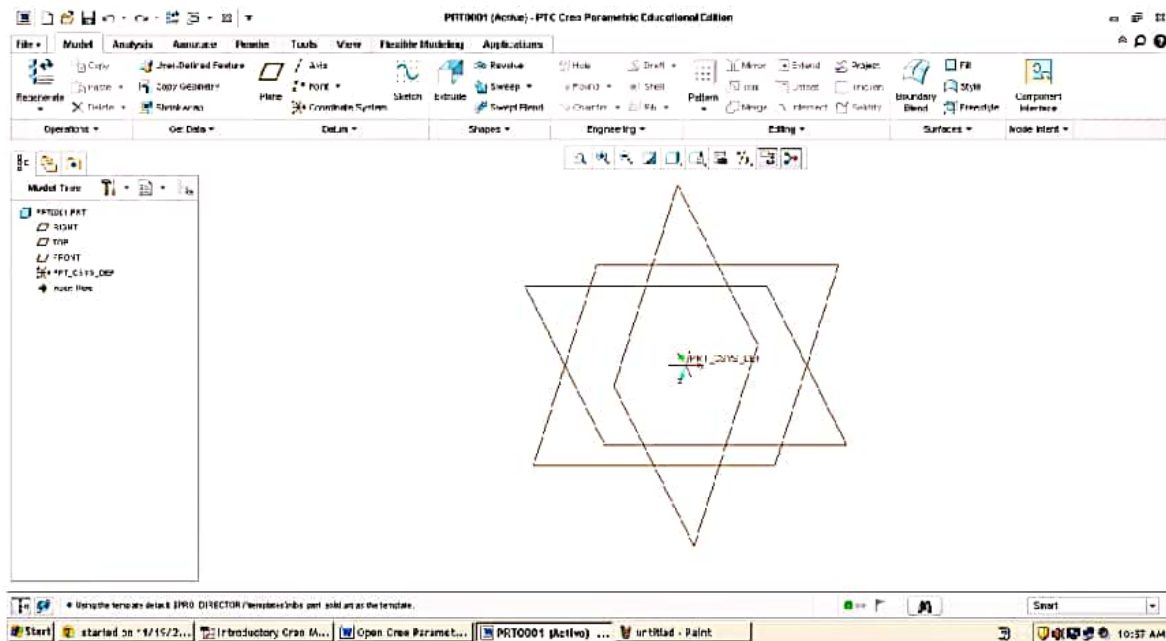
### 2. Hit Select Working Directory on the top bar and select whatever folder you want your new part to go into.



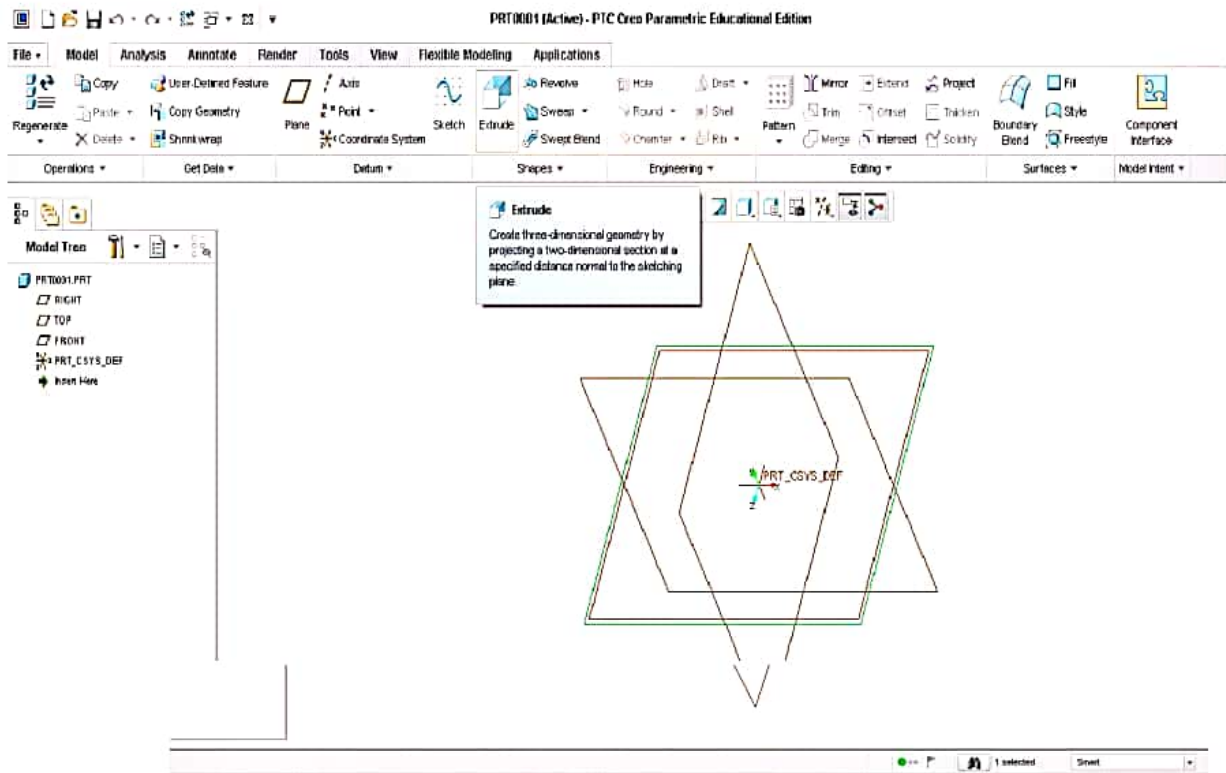
3. Next hit the New Button, make sure the type is set to part. Change the name to whatever you want to name your part and hit OK



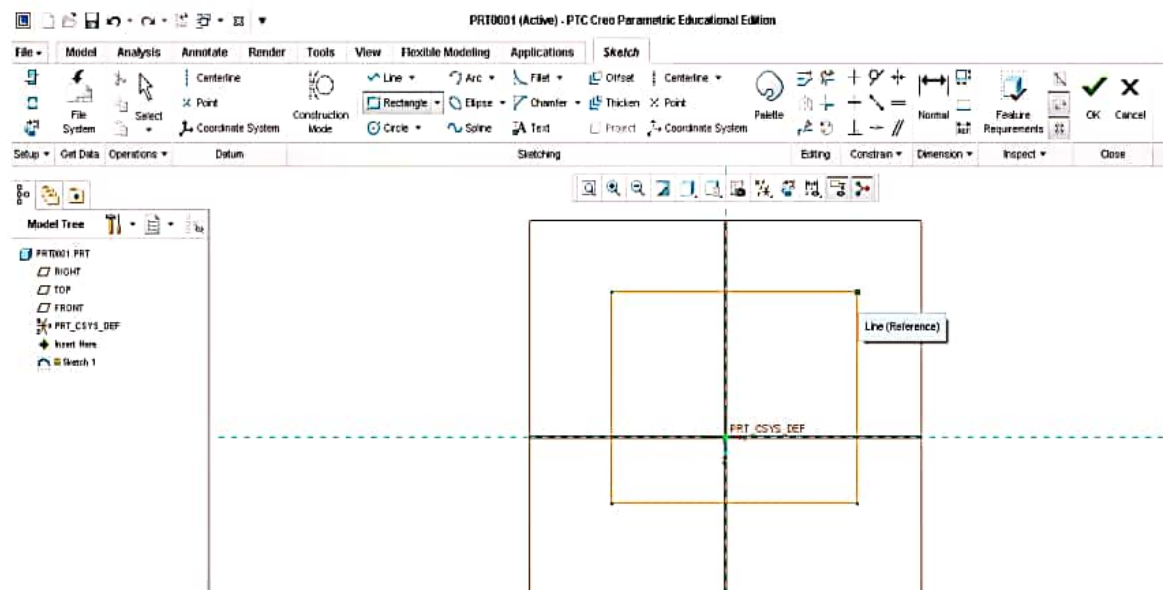
Your screen should now look like this. Hit the View tab along the top ribbon and hit the plane tag display button so you can see the names of the planes.



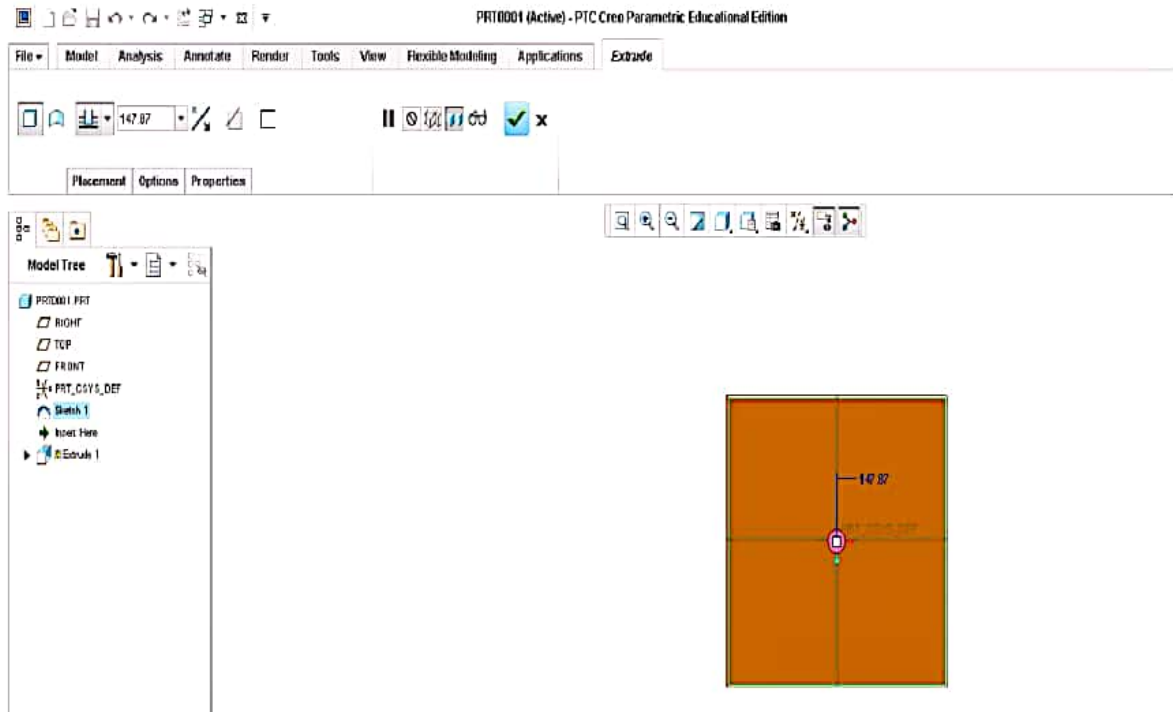
4. Next, select the Top plane, return to the model tab, and hit the extrude button



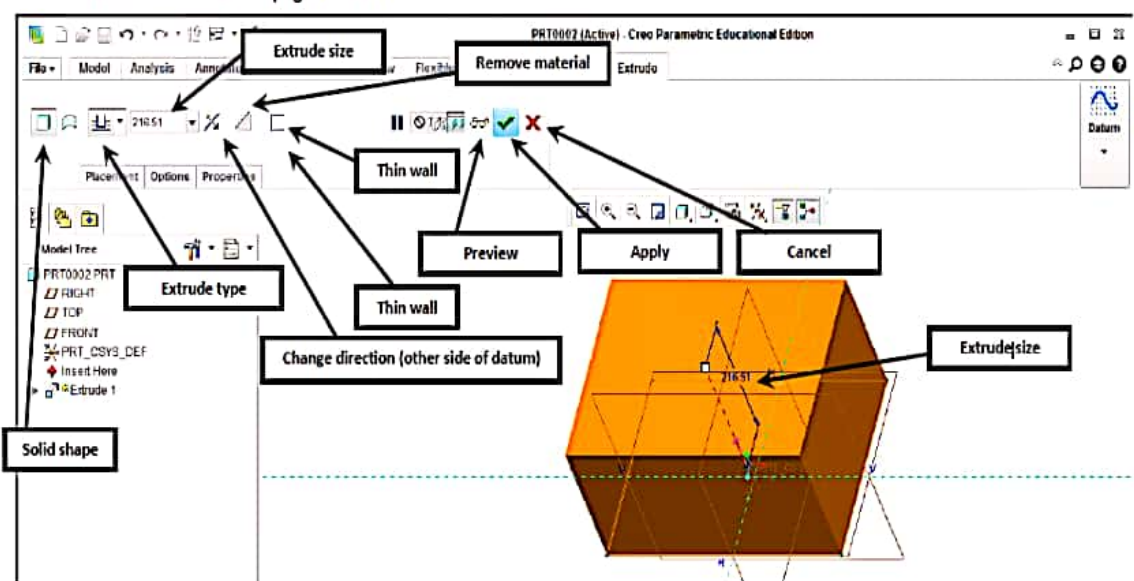
5. Then you should be in sketch mode. Hit the sketch view button on the small ribbon in the work area. So that you are viewing the top plane in 2D. Then select Rectangle in the sketch ribbon and click and drag a rectangle in your workspace .




6. Once you click again and then click the select button you will see that inside the rectangle turns orange, and dimensions appear. Change the dimensions so that it has a length and width of 6.00 and the dimensions to the centerlines are 3.00. This is where you start to have freedom. If you want to make something smaller you can, but this is the maximum size you can have for your part. Once you are satisfied hit the check mark to finish the sketch.



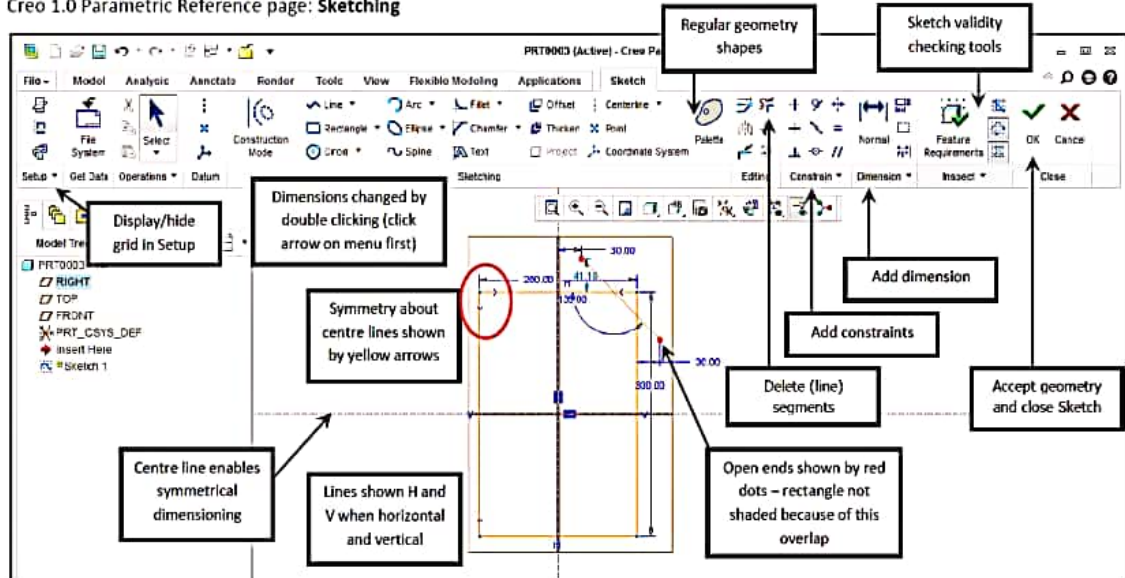
Creo 1.0 Parametric Reference page: Extrusion





1. Select Extrude from top menu (Shapes)
2. Click on required datum plane (e.g. TOP) or a flat surface on an existing model
3. Draw required shape (e.g. rectangle) – centre lines can be used for symmetrical shapes
4. Click green tick  to exit from sketcher
5. Select extrusion type (e.g. Blind or Symmetric) – extrusion can be removed from existing solids
6. Drag extrusion size (or type value in Dashboard) and click green tick to apply
7. Save the part

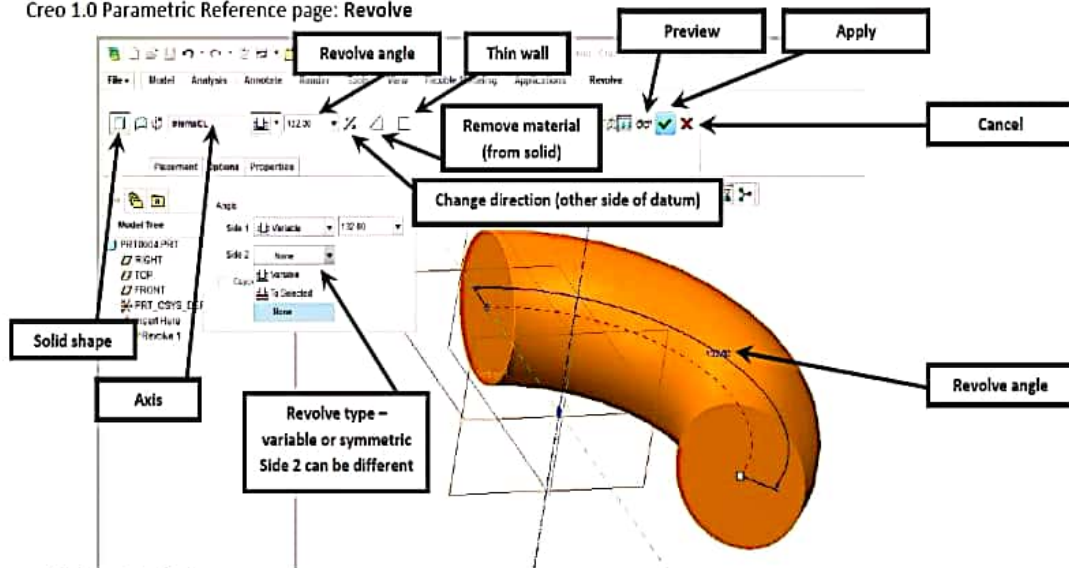
Creo 1.0 Parametric Reference page: Sketching



Delete line segments tool can be used in two ways:

- (1) Click on line or part of line, (2) Drag the red line over all lines to delete
2. Inspect Sketch Tools show whether the sketch will work with 3D feature or not. Shape is color filled when OK.
3. Constraints (equal length etc.) can be added (or removed with the delete key)
4. For construction lines click Construction Mode before drawing
5. All the above applies to internal and external sketches.

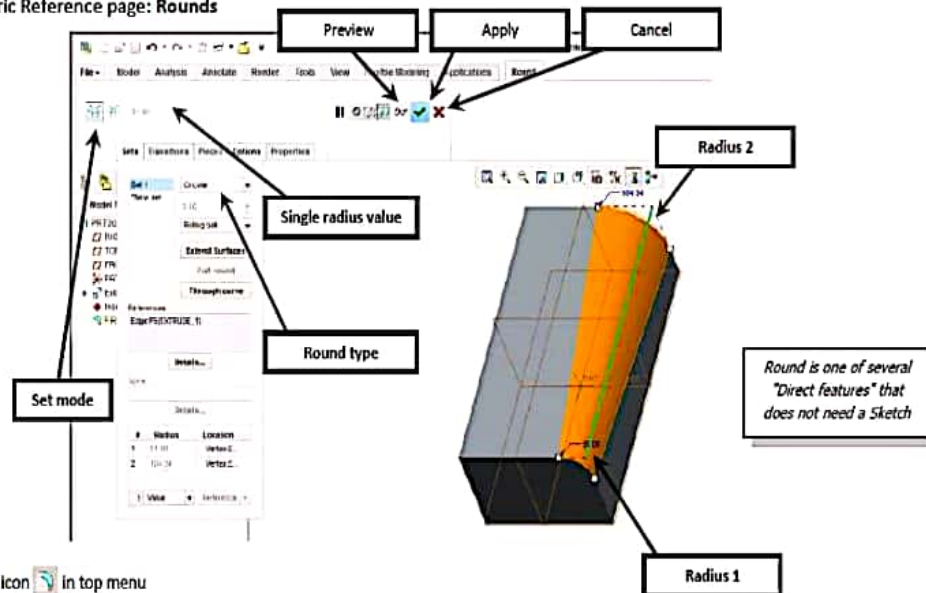
Creo 1.0 Parametric Reference page: Revolve





Select Revolve from top menu

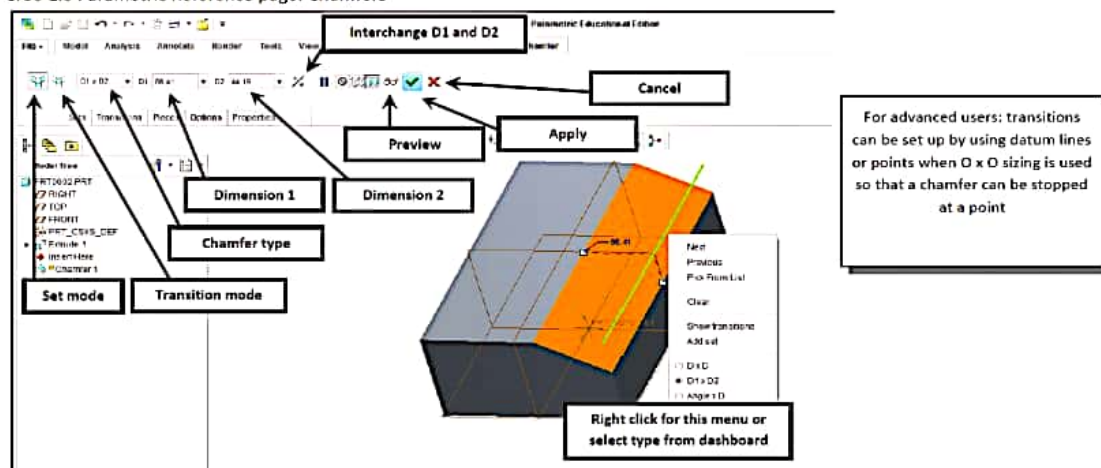
1. Click on required datum plane (e.g. RIGHT) or a plane surface of an existing solid
2. Draw required shape (e.g. circle) which must be closed - use Sketch Validity Tools to check
3. Add a Geometry centerline (or a datum axis must have been created first) - the existing edge of a solid can also be used as axis
4. Click green tick
5. Change the revolve angle (default is 360°) by entering value or drag (white) handle and click green tick to apply
  - a. Save the part .

Creo 1.0 Parametric Reference page: Rounds



1. Select Round icon  in top menu
2. Select an edge on your part - hold down CTRL to add further edges to the set
3. Drag white marker to radius required or enter value
4. To add second (or third) radius, right-mouse-click on the white drag handle and Add Radius – drag size of second radius
5. The position of third radius can be dragged along the edge to position required
6. To return to plain round, right-click on green line and Make Constant or open Sets tab to delete radius
7. Additional round sets (with single or multiple radii) can be added to other edges - click on New set, enter radius required
8. Click green tick to apply 

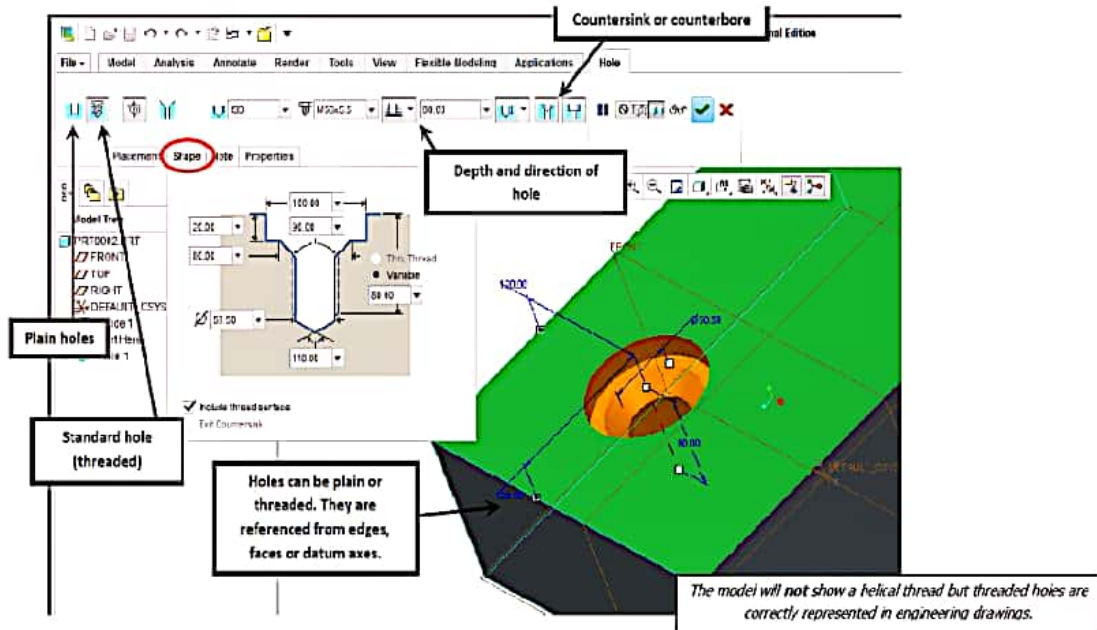
Creo 1.0 Parametric Reference page: Chamfers



1. Select the chamfer icon from the top menu
2. Select an edge on the solid part - hold CTRL to add further edges to the set

3. D x D is default for equal sizes of chamfer.
4. Use white drag handles to get the sizes D1 and D2 (or enter in the boxes above after selecting type)
5. Save the part

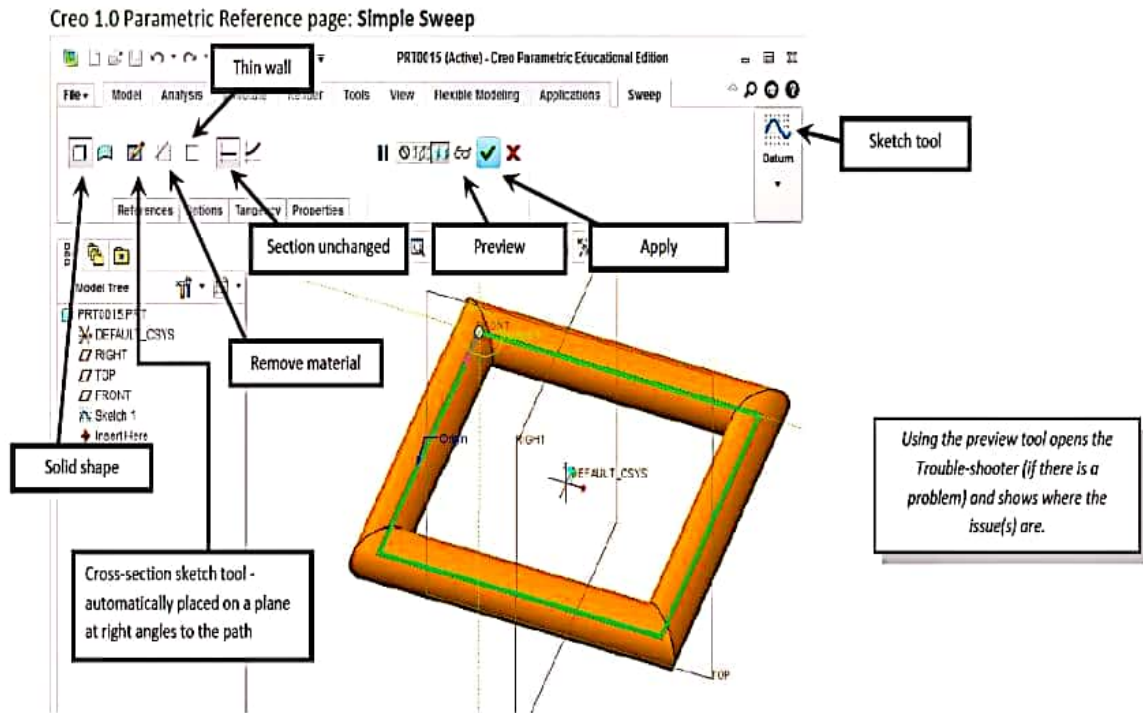
### HOLE



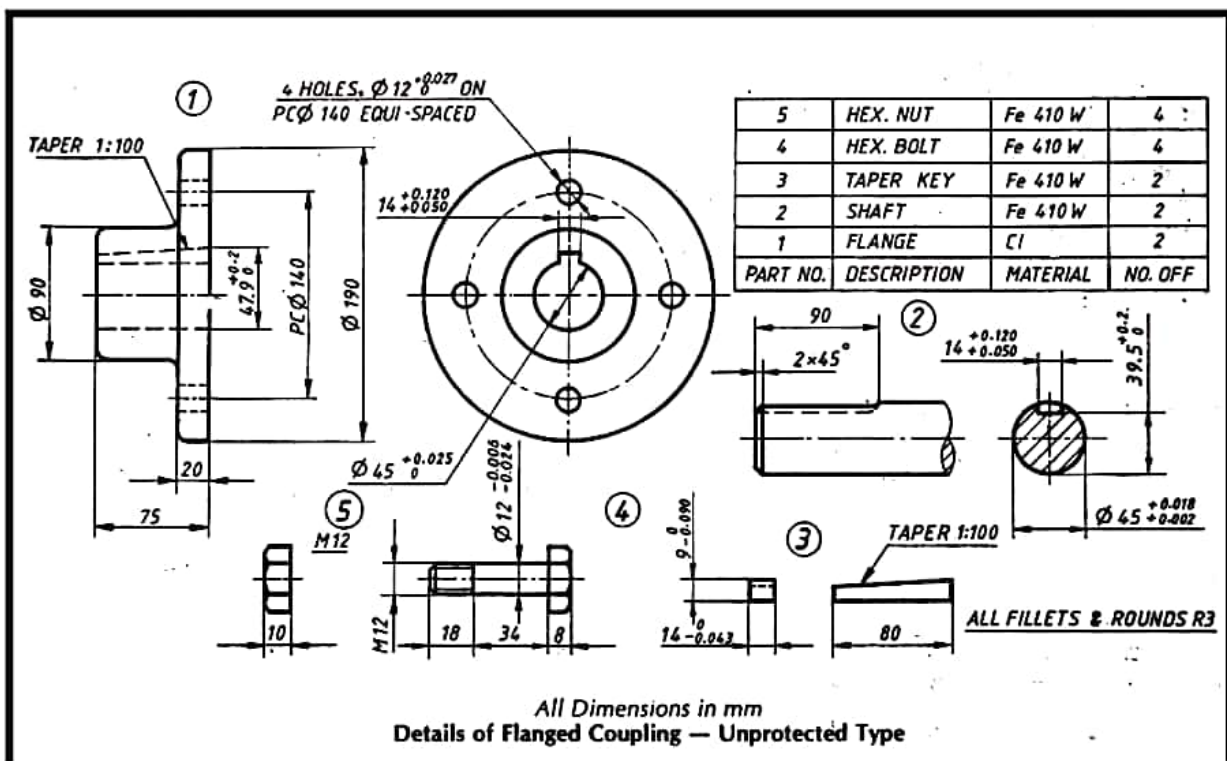
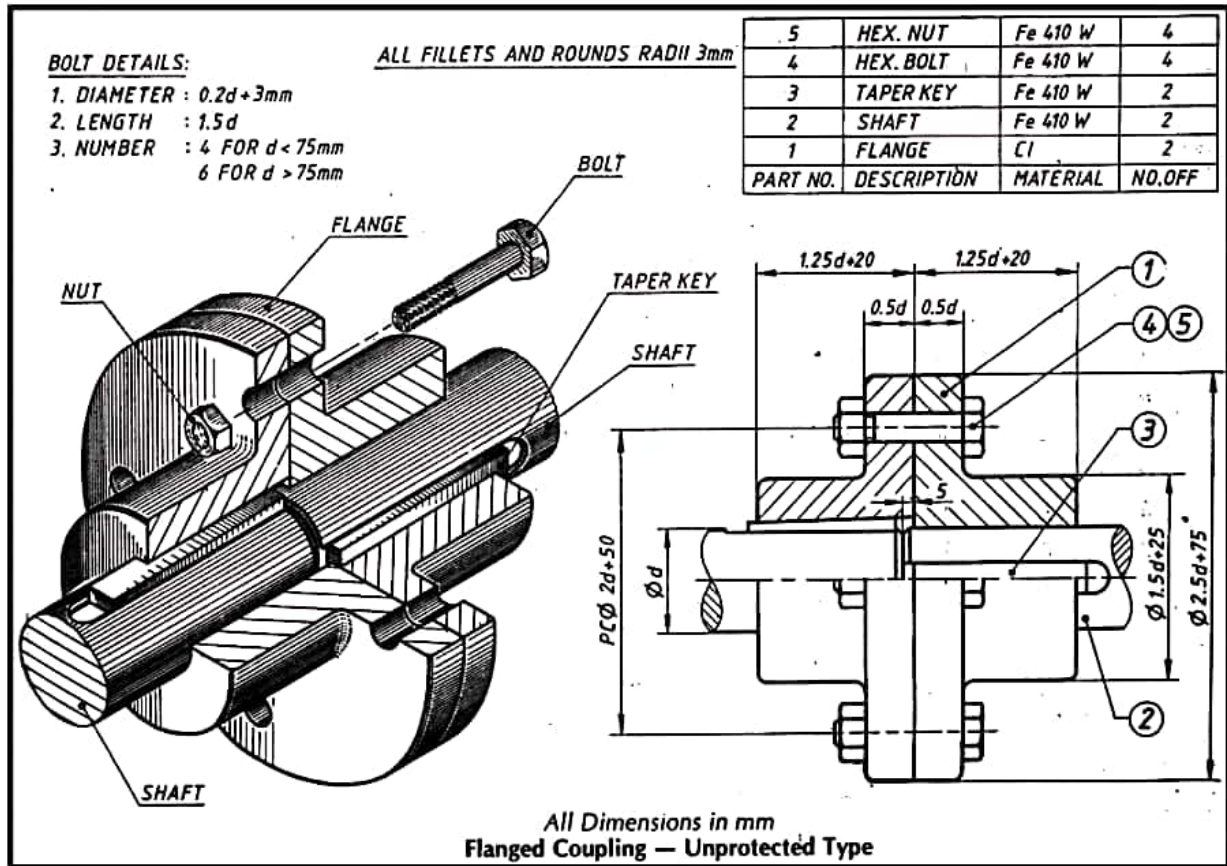
Plain holes can also be made using the Extrude feature. Holes can be made in any solid part.

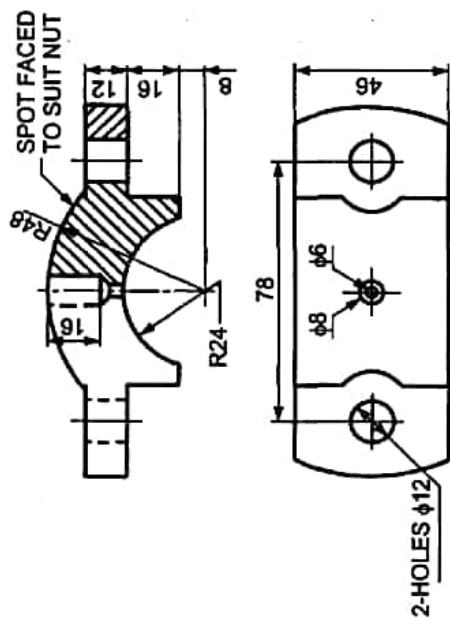
1. Select the Hole icon on the right hand menu
2. Select a face of your solid
3. Drag the green markers to faces (provides dimensional references), drag the white marker to place the hole centre
4. Drag or enter the hole diameter and hole depth
5. For threaded holes click the icon and select the required thread size and countersink, counterbore etc. – add measurements using the Shape tab.



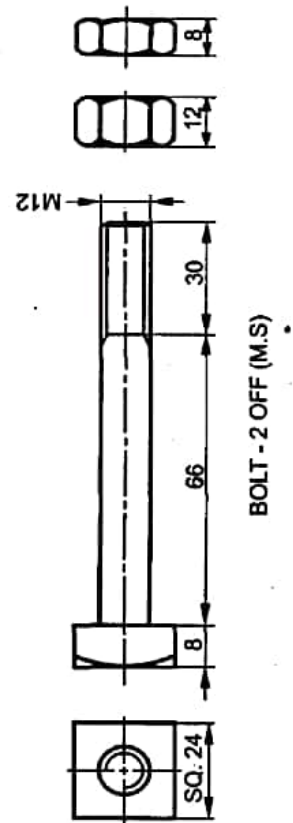
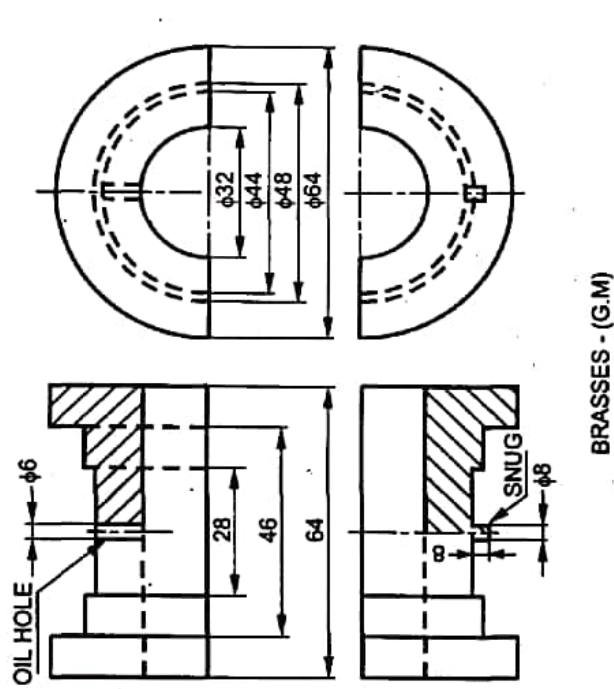
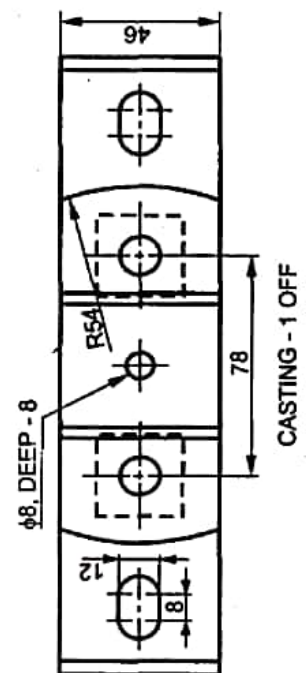
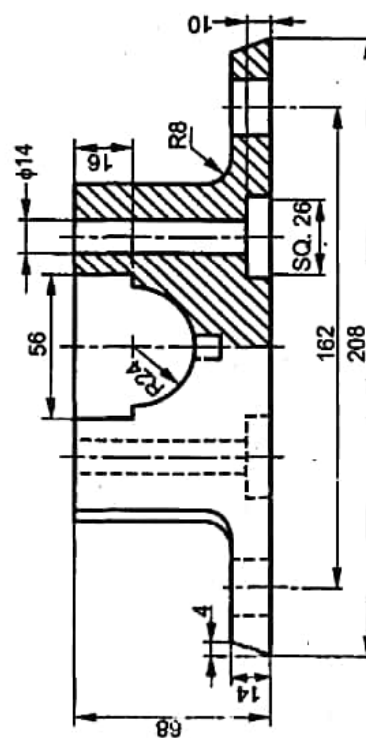
**SWEEP**

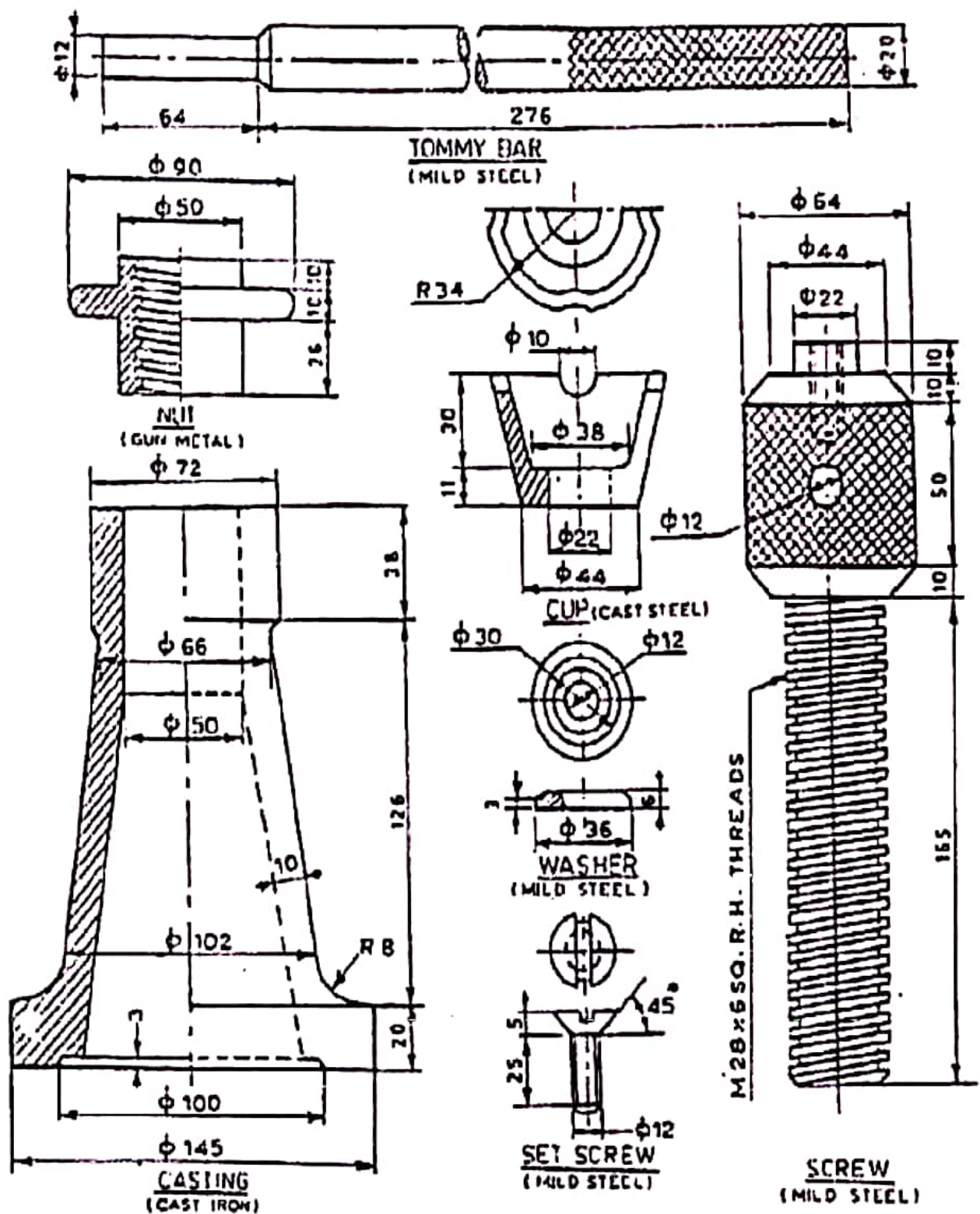
1. First draw the path as an **external sketch** on the required datum plane (e.g. TOP) or on a surface of an existing part - click green tick - leave the line highlighted (don't click on the window)
2. Select the Sweep feature in the top menu (Shapes)
3. For a simple sweep, keep Solid and Section remains unchanged (defaults)
4. Click on the **Create or edit sweep section tool** and draw the cross-section (e.g. a circle) at or near the end of the sketch line
5. Click green tick to accept the sketch and click green tick to apply
6. Save the part



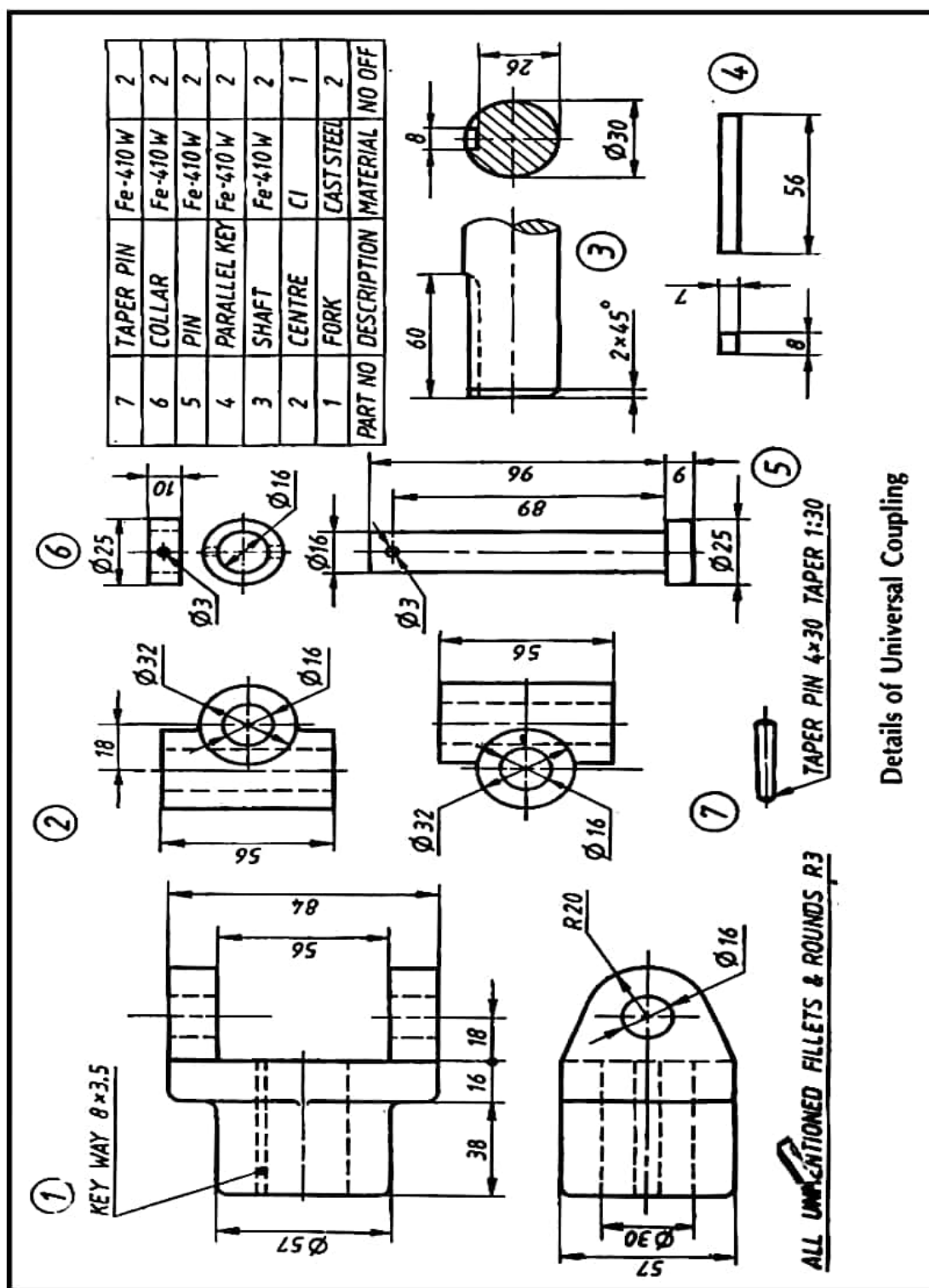


CAP - 1 OFF







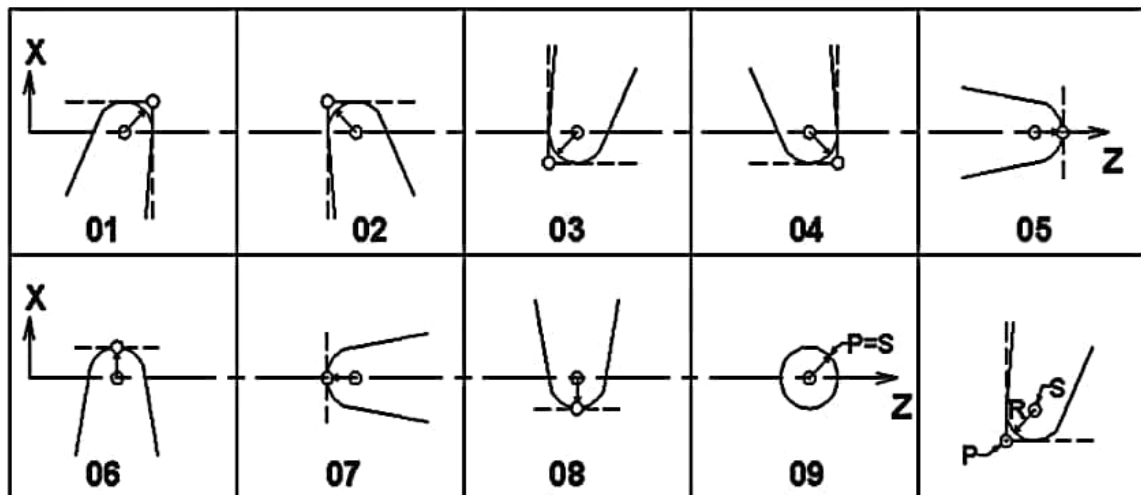


VARIOUS TOOL NOSE RADIUS

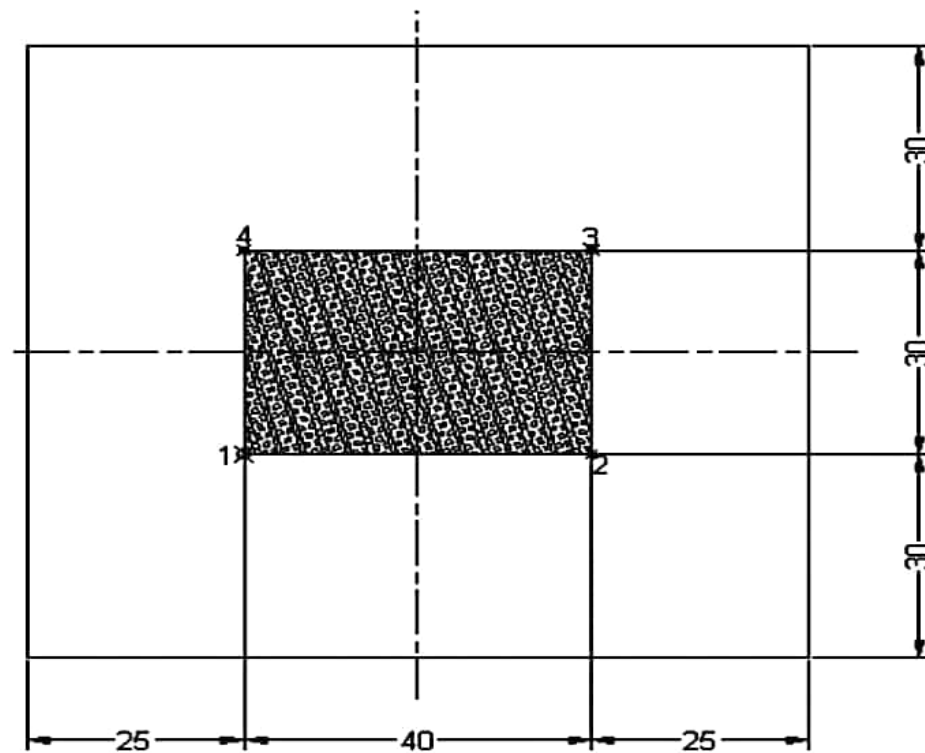
R - Tool Nose Radius ( Tool Radius )

S - Position of Cutting Edge Center Point

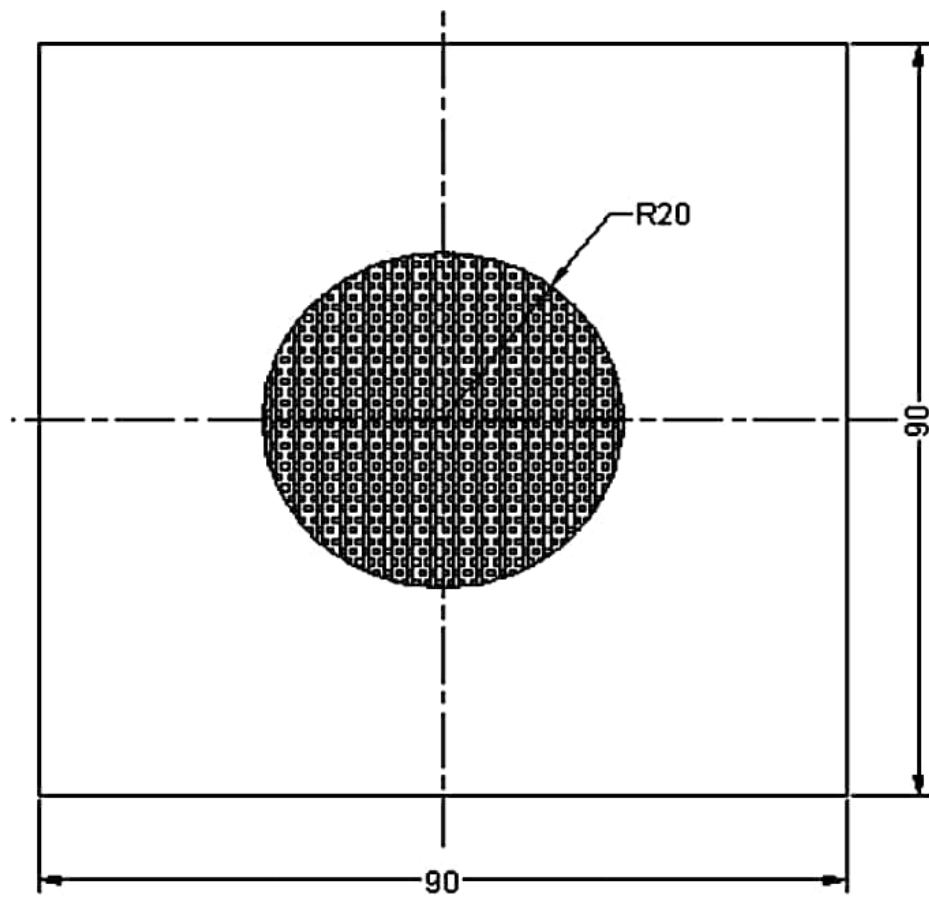
P - Tool Tip ( Cutting Edge )

**RESULT:**

Thus the G- Codes and M- Codes were studied.

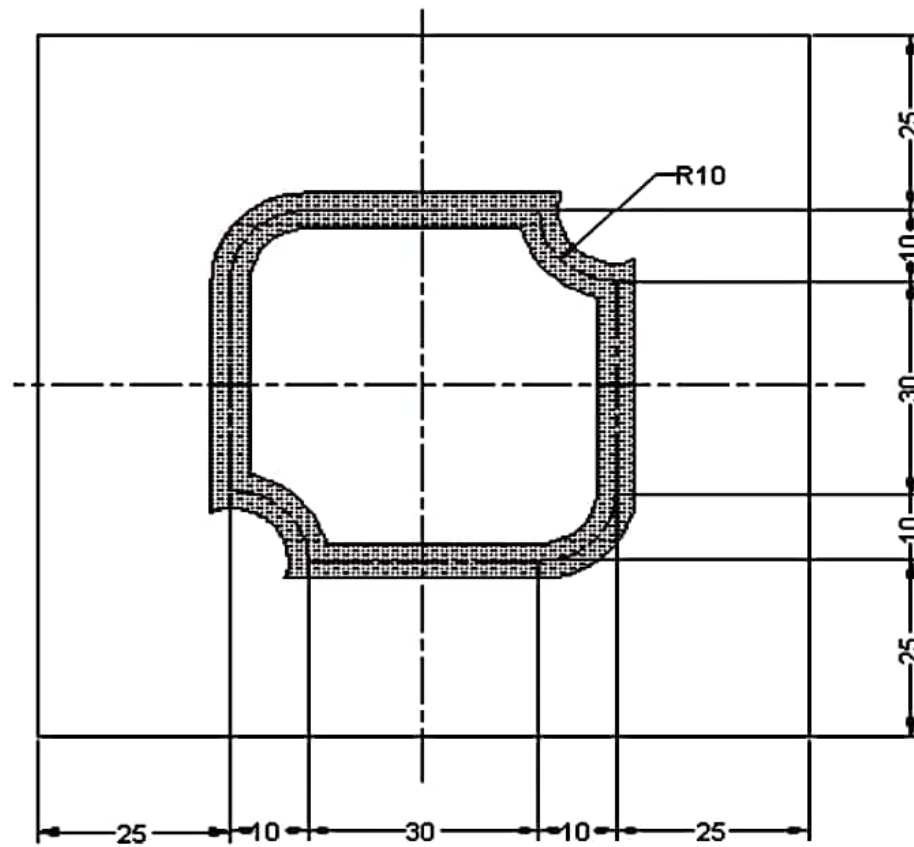


**Rectangular Pocketing :** Billet size: - 90 x 90 x 10 mm; Depth of cut – 3 mm

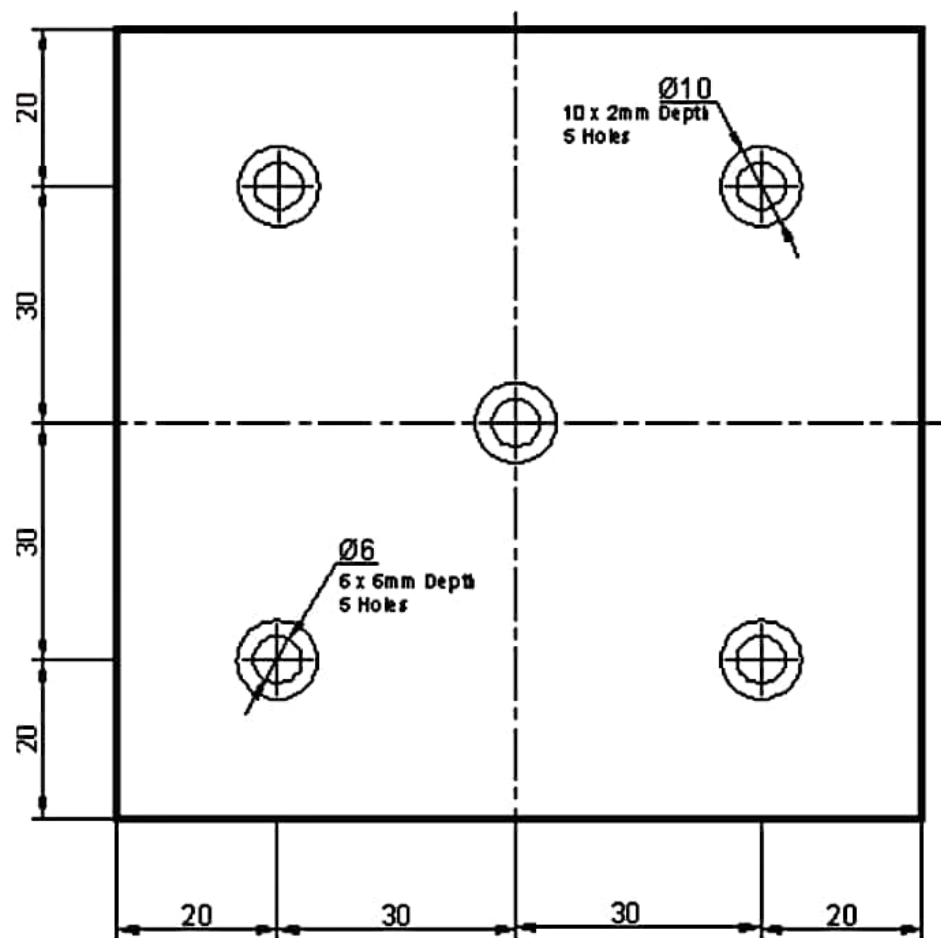


**Circular Pocketing :** Billet size: - 90 x 90 x 10 mm; Depth of cut – 3 mm

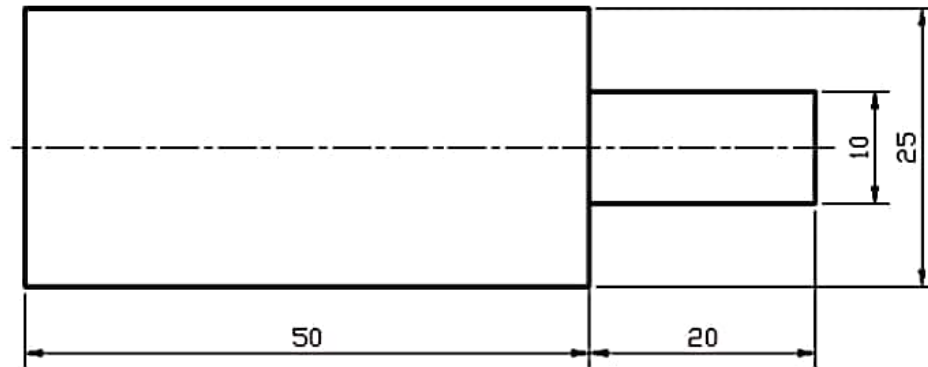




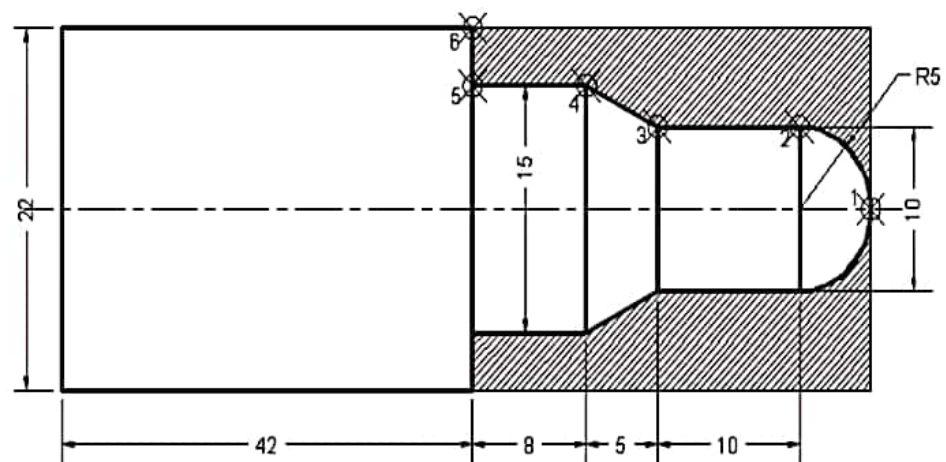
**Simple Contour Milling:** Billet size: - 100 x 100 x 10 mm; Depth of cut – 1 mm



**Canned Cycle :** Billet size: 100 x 100 x 10 mm; Drilling Tool :  $\varnothing 6$ ; Boring Tool :  $\varnothing 10$

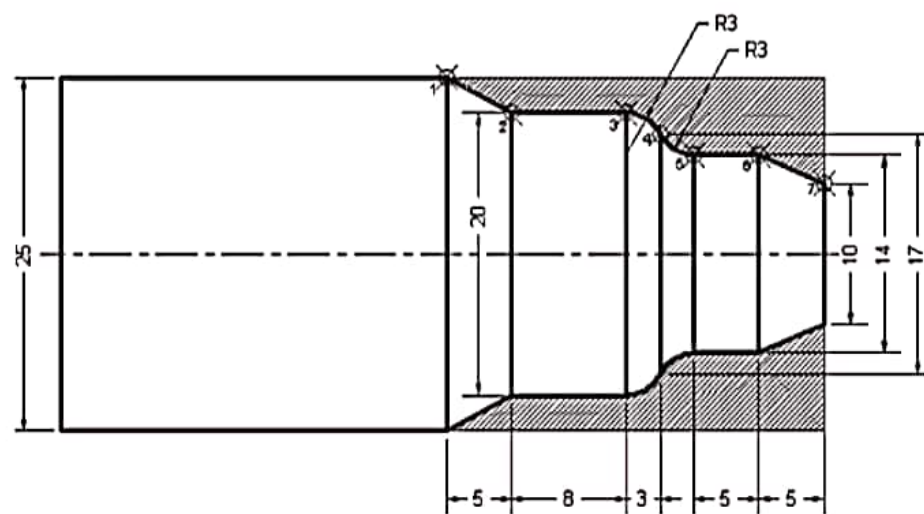


**Turning Cycle :** Billet size:  $\varnothing 25 \times 70$  mm

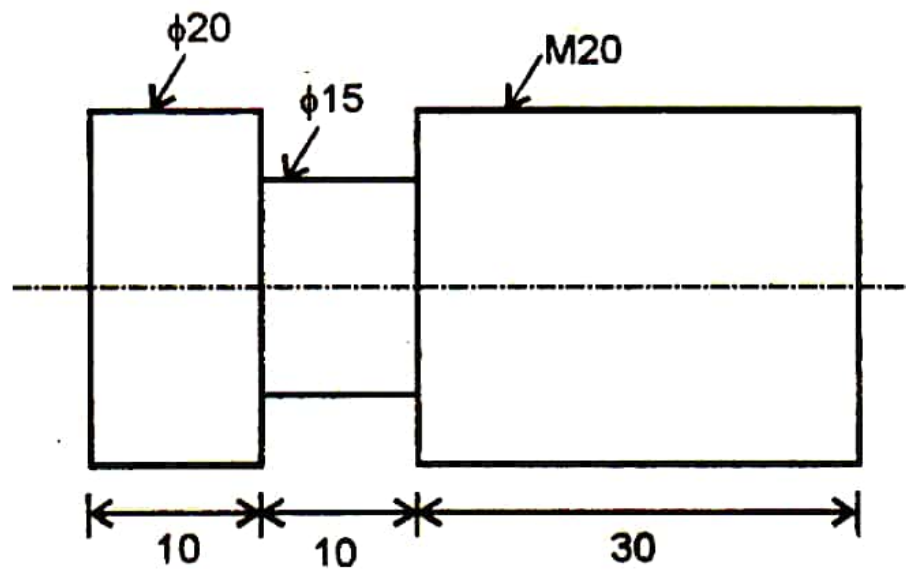


**Multiple Turning Cycle :** Billet size:  $\varnothing 22 \times 65$  mm

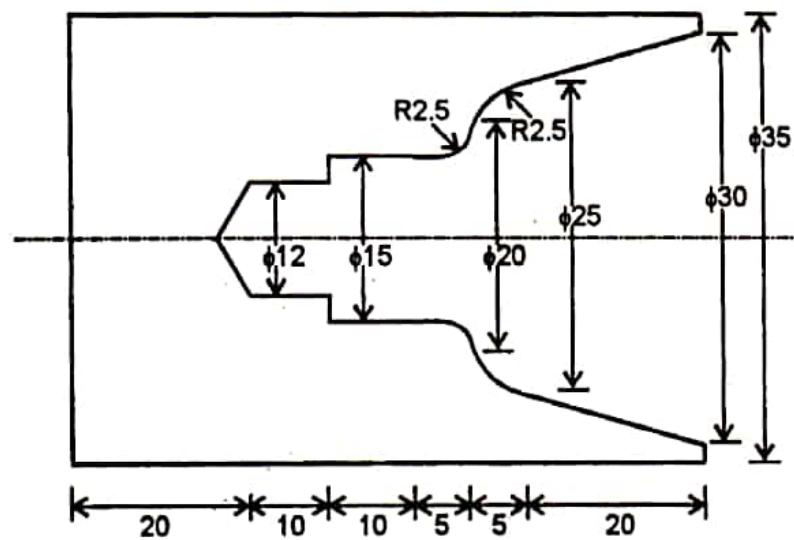




**Multiple Facing Cycle :** Billet size: Ø 25 x 70 mm



**Grooving and Thread Cutting Cycle :** Billet size:  $\phi 20 \times 50$  mm



**End Face Peck Drilling and Boring Cycle :** Billet size:  $\phi 35 \times 70$  mm