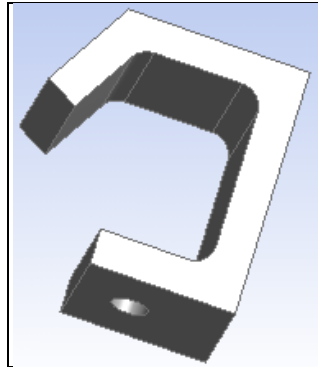


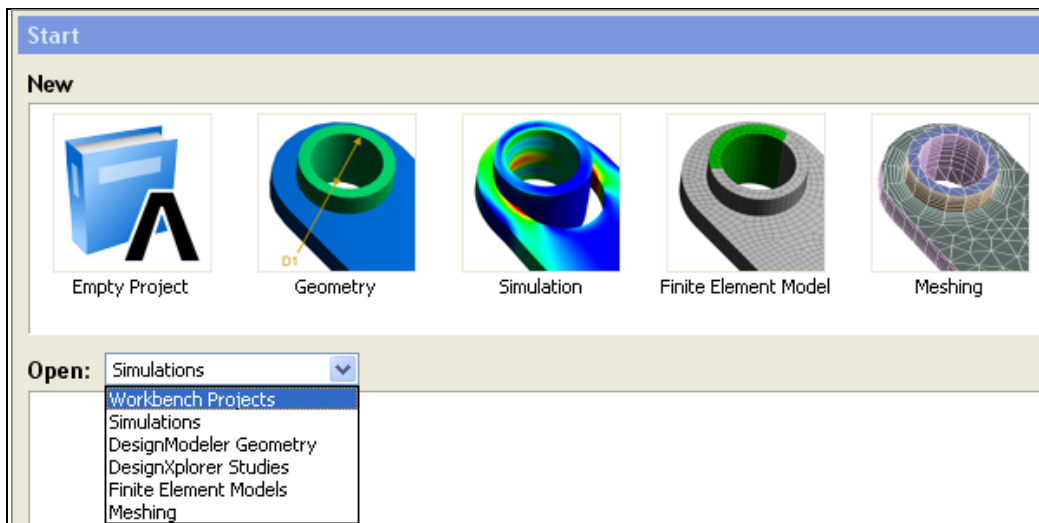
Introduction

This tutorial is intended to introduce Ansys Workbench 11 by constructing a C-bracket shown below.

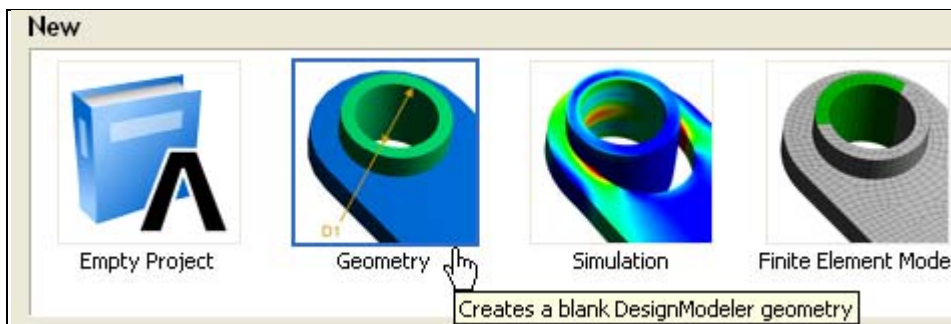


You can open Ansys from within SolidWorks (via **Tools**→**Add Ins**→**Ansys 11**) or directly:

1. **Start**→**Programs**→**Ansys**→**Workbench** (which may take about 10 minutes to open) lets you access the Workbench **Start** window



2. Clicking on **Geometry** will let you begin sketching and modifying the part:

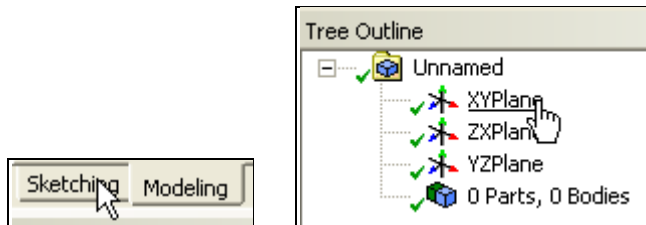


3. The **Design Modeler** page opens (top region shown) and the user is required to select the desired units.

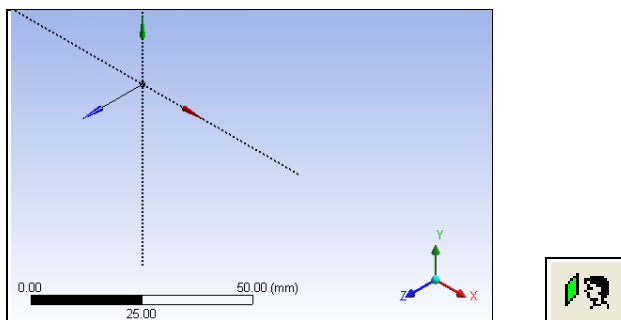


Sketching

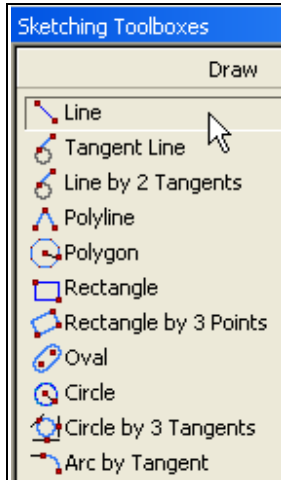
- On the left of the page select **Sketching** to open the construction **Tree Outline**



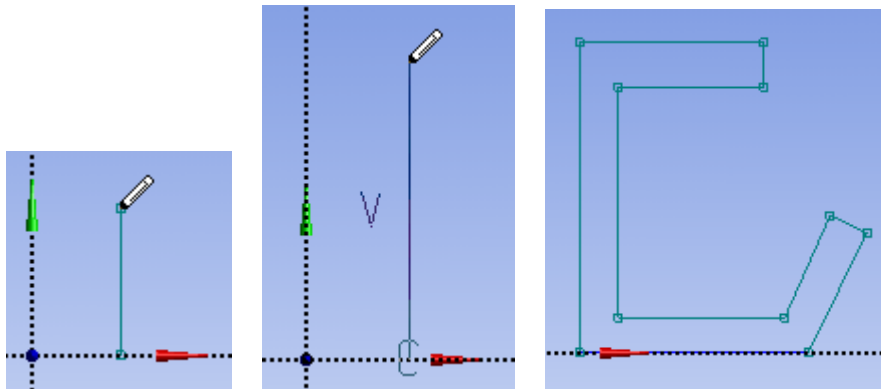
- In the **Tree Outline** select the **XYPlane** for the initial sketch. It opens as an isometric view. Select the **Look At** (or **Normal To**) icon to get the true shape view



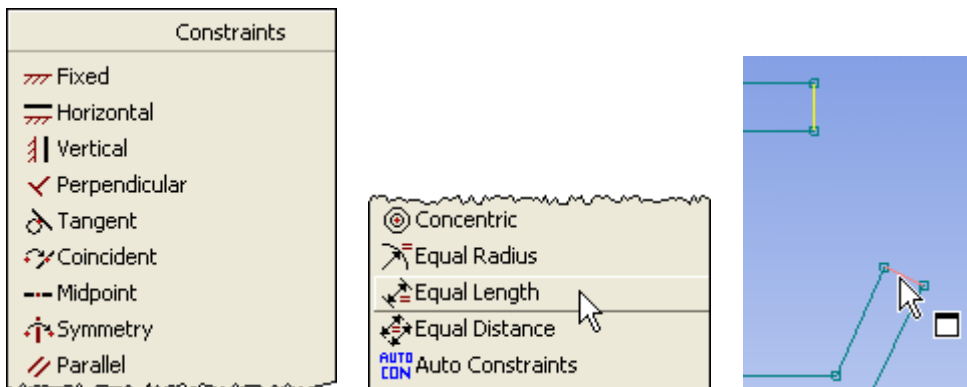
- On the left of the page the **Sketching Toolboxes** panel appears with the list of line options



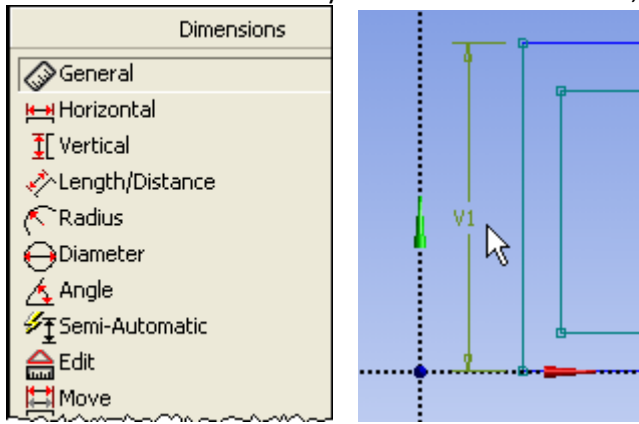
- From the list pick **Line** to begin the input of straight lines outlining the bracket shape. Symbols such as **V**, and **H** appear to denote when the line is vertical or horizontal, etc.



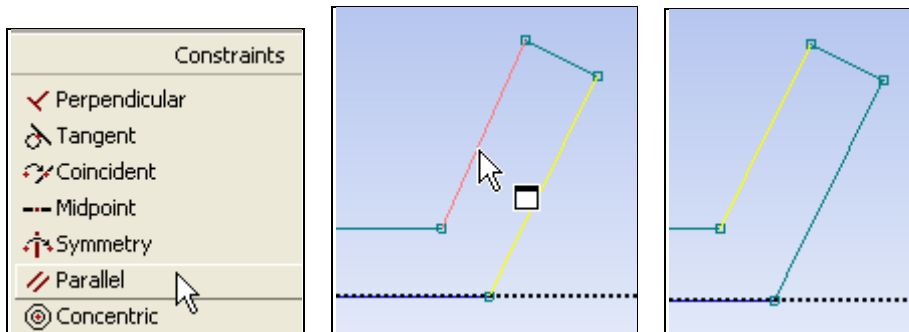
- To require the two end segments to have the same length select **Constraints**→**Equal Length** and click on the two end segments.



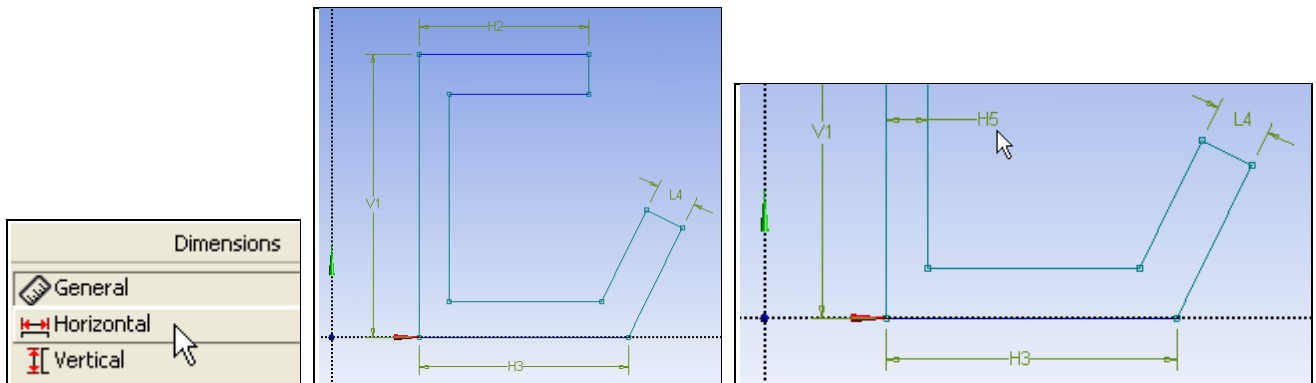
- Begin describing the dimensions to be available as parametric design features with **Dimensions**→**General** and select the leftmost vertical edge. The parameter **V1** appears as a dimension. The letter means that it is a vertical line. The value of the actual dimension will be assigned shortly. Note that there are several choices for dimensions in the **Dimensions** panel.



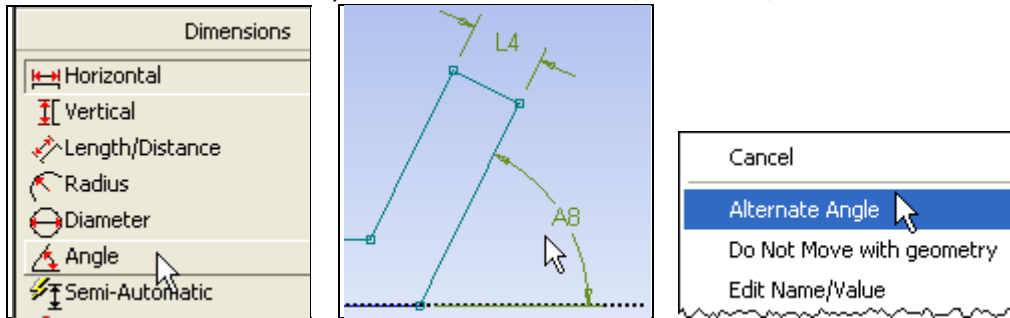
10. The inclined segment of the bracket does not appear to have parallel sides as desired. To assure that it does select (on the left of the page) **Constraints**→**Parallel** and pick the two lines. Note that the **Constraints** panel has several common types available for the user.



11. Continue with the remaining length dimensions via **Dimensions**→**Horizontal** and select the three locations.



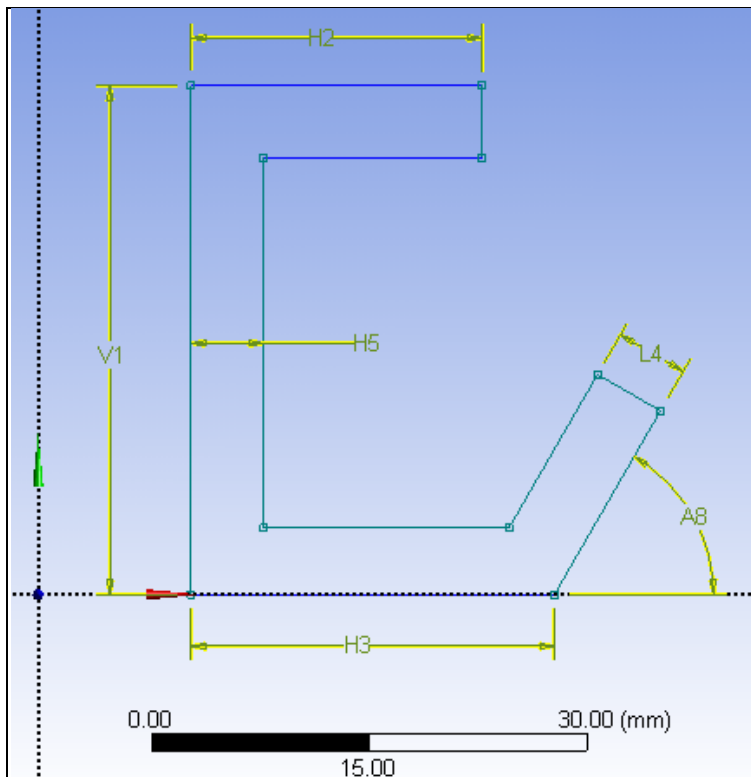
12. For the final parameter specify the angle of the inclined leg: **Dimensions**→**Angle** and pick the inclined line and the bottom line. Sometimes you will want to **right click** to specify an **Alternate Angle** choice.



13. At this point the actual parametric dimensions will be input to replace those in the initial sketch. On the left of the page the **Details View panel** contains the **Dimensions** sub-panel with the default dimension names (V1, L4, etc) and their as sketched initial values. Click on each value to be changed and type in the desired values. The sketch changes shape to reflect the new dimension values.

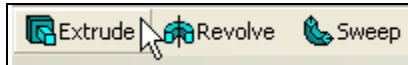
| Details View | |
|----------------------------------------|-------------|
| [-] Details of Sketch1 | |
| Sketch | Sketch1 |
| Sketch Visibility | Show Sketch |
| Show Constraints? | No |
| [-] Dimensions: 6 | |
| <input checked="" type="checkbox"/> A8 | 63.8 ° |
| <input type="checkbox"/> H2 | 20.213 mm |
| <input type="checkbox"/> H3 | 24.976 mm |
| <input type="checkbox"/> H5 | 3.5837 mm |
| <input type="checkbox"/> L4 | 4.7732 mm |
| <input type="checkbox"/> V1 | 33.848 mm |

| Details View | |
|-------------------------------|-------------|
| [-] Details of Sketch1 | |
| Sketch | Sketch1 |
| Sketch Visibility | Show Sketch |
| Show Constraints? | No |
| [-] Dimensions: 6 | |
| <input type="checkbox"/> A8 | 60 ° |
| <input type="checkbox"/> H2 | 20 mm |
| <input type="checkbox"/> H3 | 25 mm |
| <input type="checkbox"/> H5 | 5 mm |
| <input type="checkbox"/> L4 | 5 mm |
| <input type="checkbox"/> V1 | 35 mm |

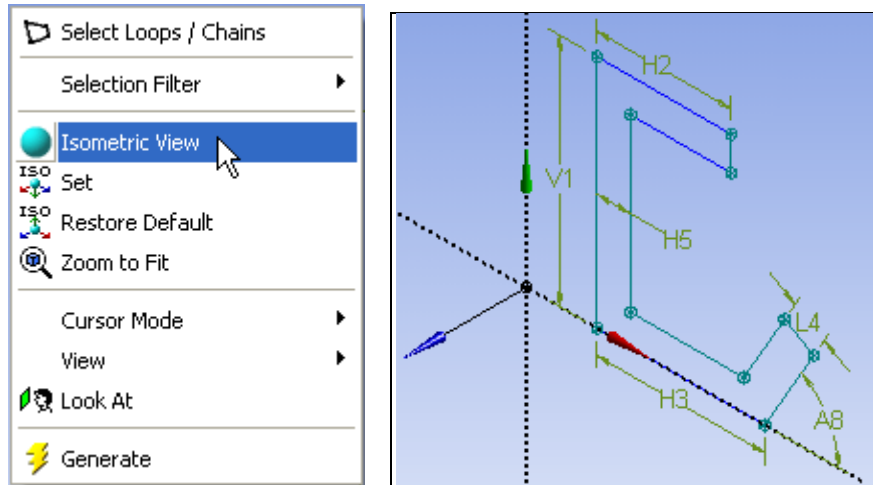


Extrusion

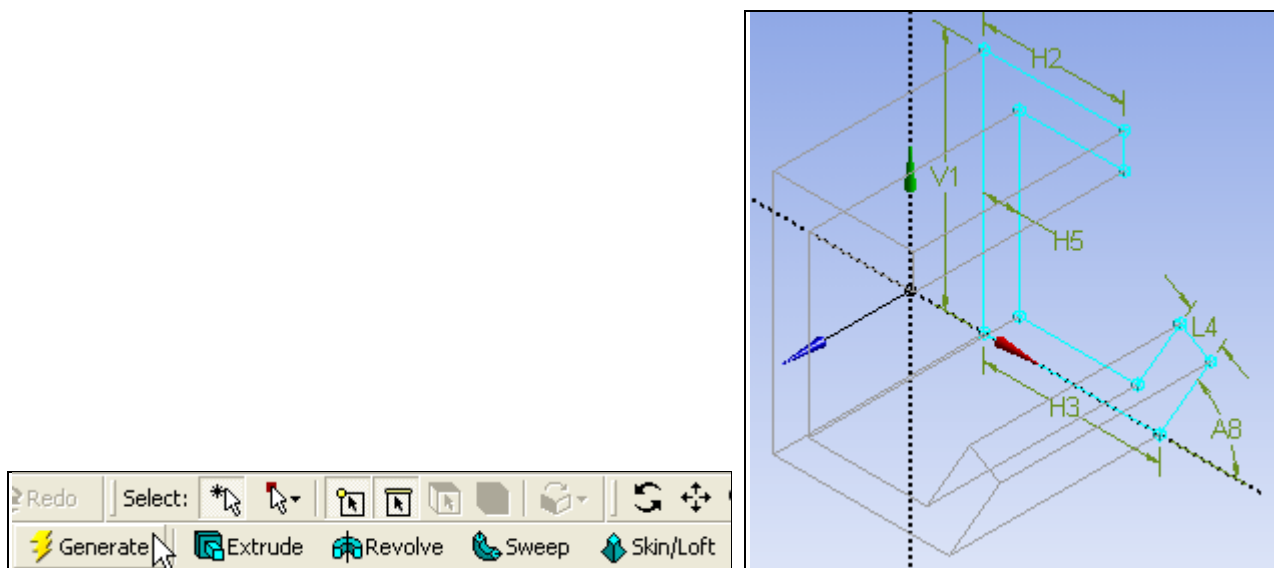
14. Having completed the cross-sectional sketch you are ready to form a solid with an **Extrude** operation. Note that icon at the top of the page, along with other construction features like Revolve, Sweep, Loft, etc.



15. Change to an **Isometric View** (by any of various methods) to be able to see the extrusion normal to the sketch.

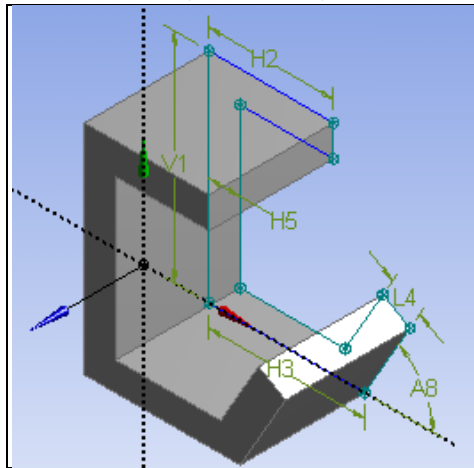


16. Pick the **Generate** icon at the top (this is a frequently used icon) to actually perform the extrusion. A default extrusion length appears in the isometric view.



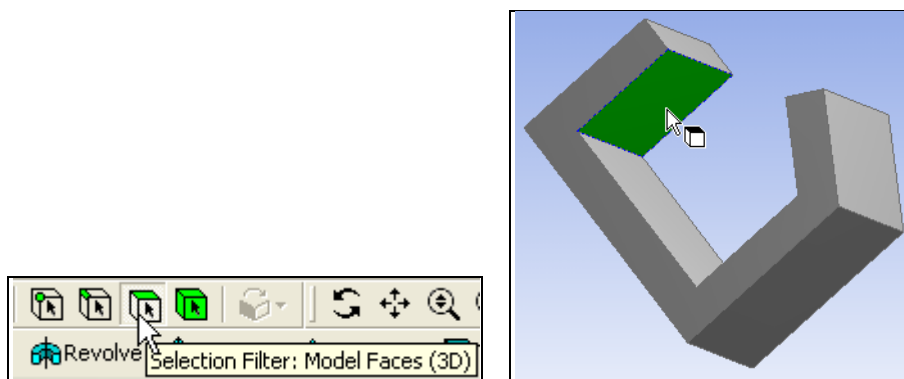
17. To specify a desired extrusion length use **Details View** → **Details of Extrude 1** and click on **Depth** and type in the actual length (here 20 mm).

| Details View | |
|----------------------------|---------------|
| Details of Extrude1 | |
| Extrude | Extrude1 |
| Base Object | Sketch1 |
| Operation | Add Material |
| Direction Vector | None (Normal) |
| Direction | Normal |
| Extent Type | Fixed |
| FD1, Depth | 20 mm |
| As Thin/Surface? | No |
| Merge Topology? | Yes |

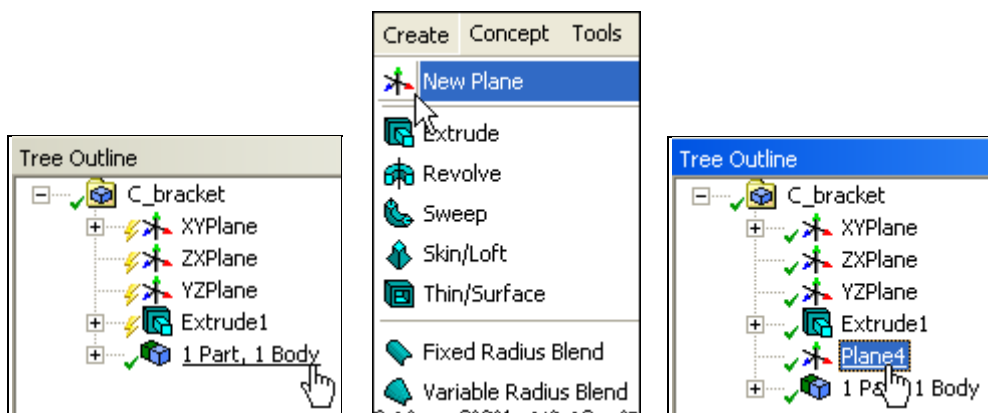


Features

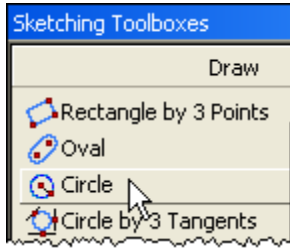
18. The part is almost complete except for interior fillets and the support bolt hole. To create the bolt hole it is necessary to select one of the two top planes, insert a circle sketch, and cut out the hole. At the page top pick the **Select Face** icon, and click on the lower surface (the top would have actually been easier).



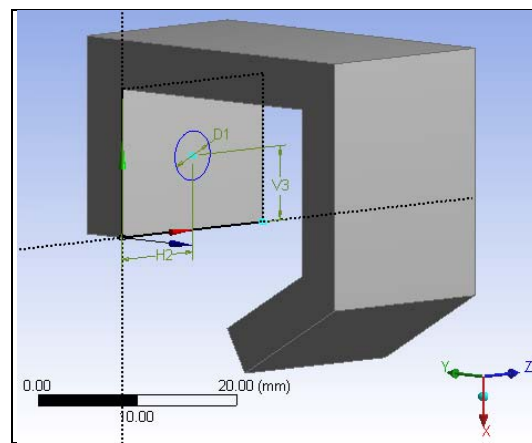
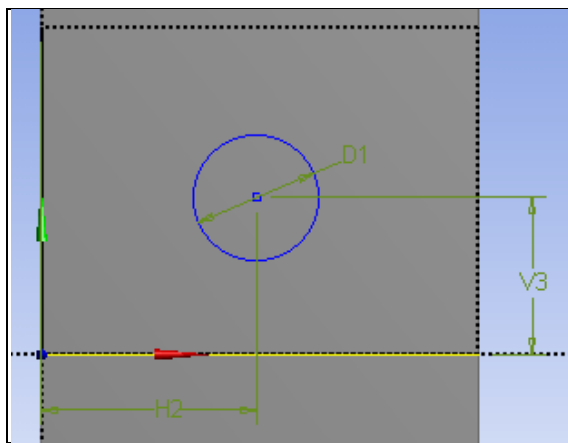
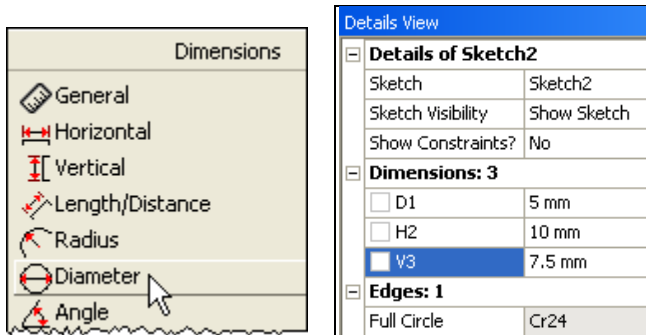
19. In the **Tree Outline** right click on **Body**→**Create**→**New Plane**. A default new plane name will appear in the tree.



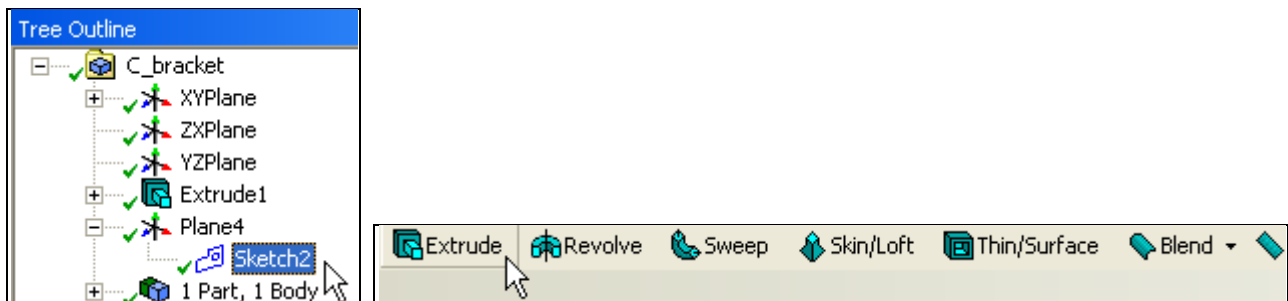
20. Use **Sketching**→**Sketching Toolboxes**→**Circle** and place the circle near the center of the surface.



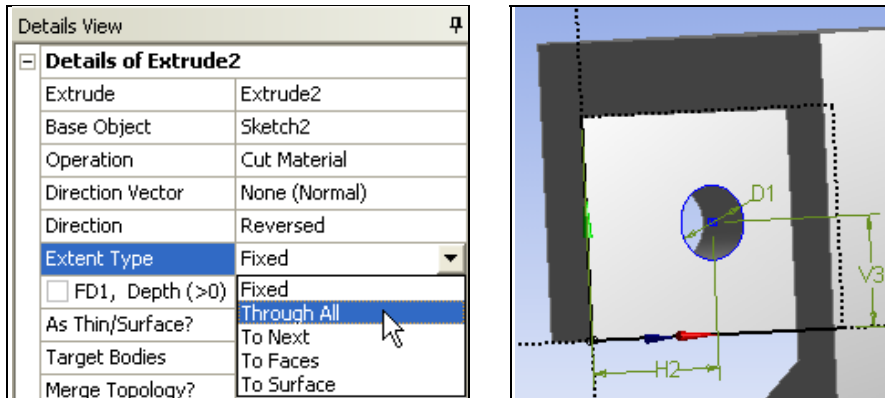
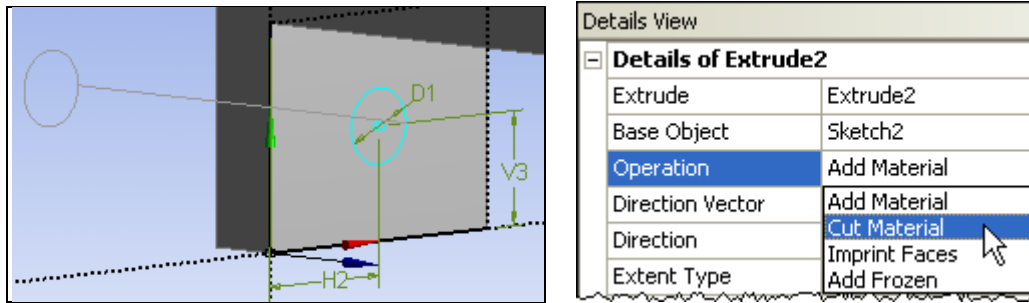
21. Add the new dimensions with **Dimensions**→**Diameter**, **Dimensions**→**Vertical**, **Dimensions**→**Horizontal**, and provide the desired values via **Details View**→**Dimensions** for D1, H2, and V3.



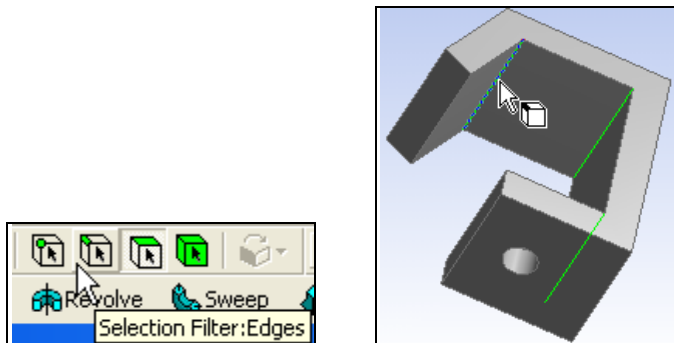
22. To form the cut select the Sketch_name and then Extrude



23. A default extrude length appears. Specify a cut with **Details View**→**Operation** and right click on **Add Material** and change it to **Cut Material**. Use **Details View**→**Extent Type**→**Through All**→**Generate**.



24. For the fillets, pick the **Select Edge** filter icon and pick the three interior edges.



25. Form and dimension the fillets **Create**→**Fixed Radius Blend** followed by **Details View**→**Details of FBlend** to set the radius to 3mm and **Apply** to see the fillets appear and finish the part.

