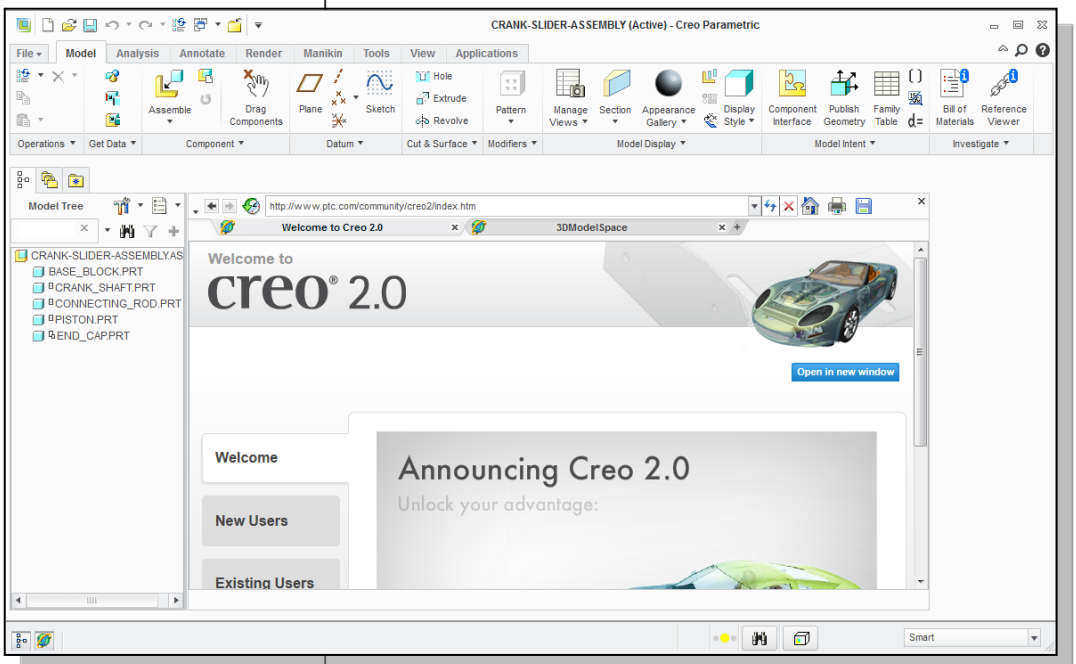
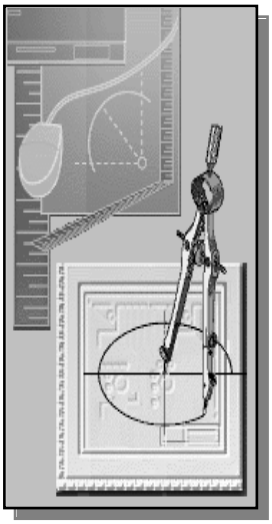
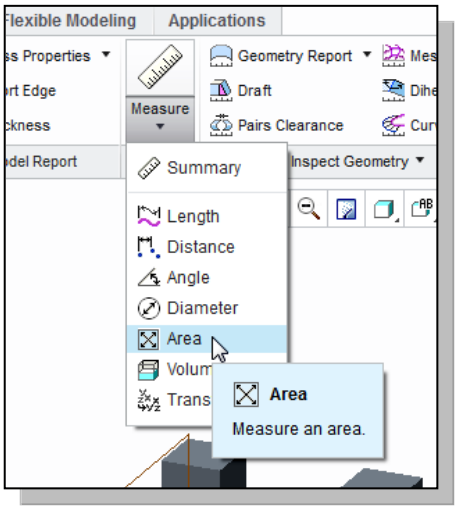
Updating to Creo Parametric 2.0





The information in this booklet is provided for users who have Creo Parametric 2.0.
Page numbers and step numbers correspond to the printed book.

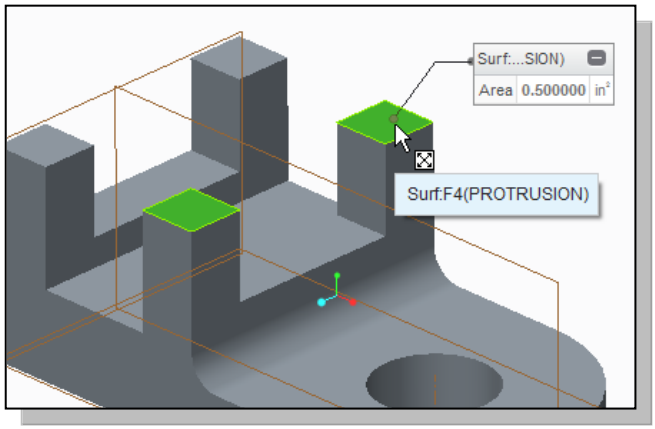
Page 3-28: Using the Measure Tools

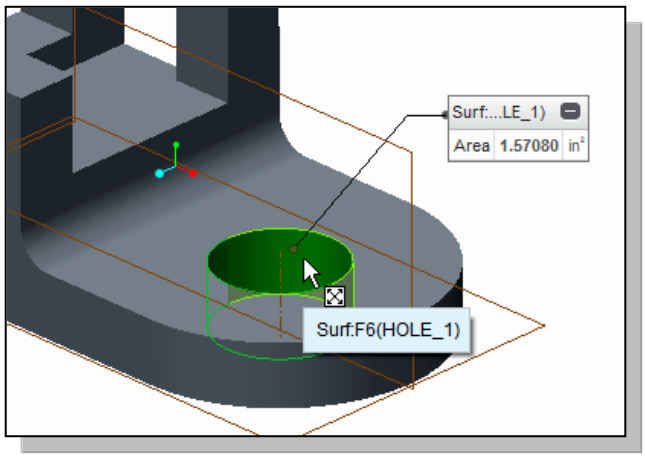


2. In the *Measure* toolbar, click the **Area** option as shown.

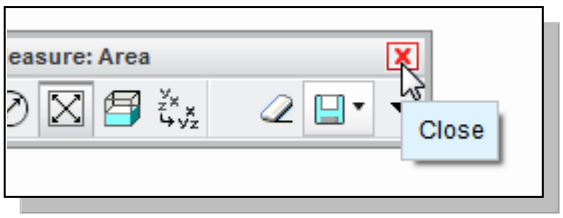
   - Note that **other** measurement options are also available in the pull-down list.

3. Click on one of the small flat surfaces of the model as shown, and notice the adjacent surface is also highlighted; the highlighted surface area is calculated as ***0.5***.





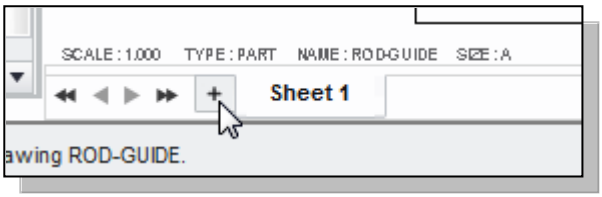
4. Click on the **cylindrical surface**, on the inside of the hole; the surface area is calculated as shown.



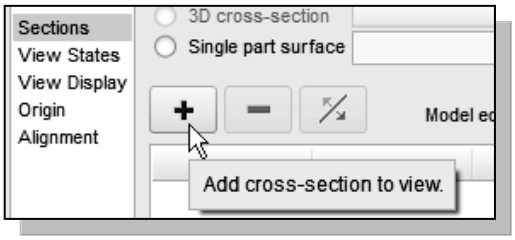
5. Click **Close** to end the Measure: Area analysis option.

Page 6-35: Adding a new Sheet in the Existing Drawing

2. Click on the **Add New Sheet** icon, located next to the **Sheet 1** tab near the bottom of the graphics area, to add a new sheet.



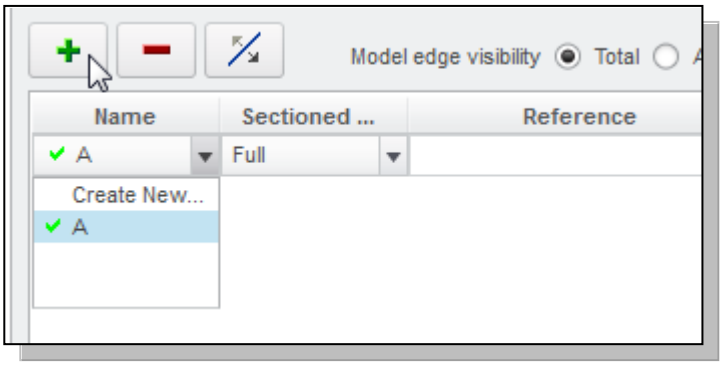
Page 7-18: Adding a Projected Sectional View



8. Click the [**+**] icon to **Add cross-section to view**.

9. In the *Section Views* list, a **Full Section A** is automatically created by default.

- Note that additional section views can be created using the **Create New** option.

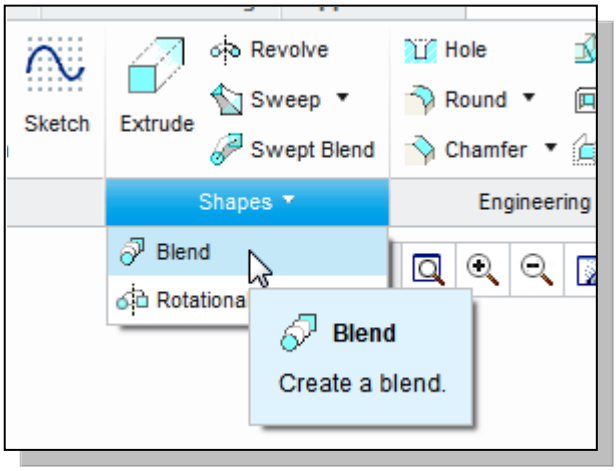


10. We will use the default settings and click **Apply** to accept the settings.

11. Proceed to Step 13 on page 7-19.

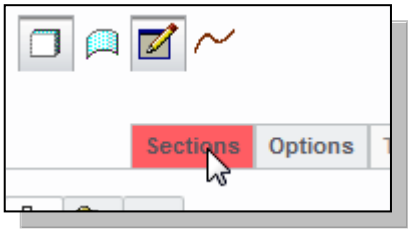
Page 8-9: Create a Blend Feature

❖ A *blend* feature is a series of two-dimensional sections that are joined together at the edges with transitional surfaces to form a continuous solid.



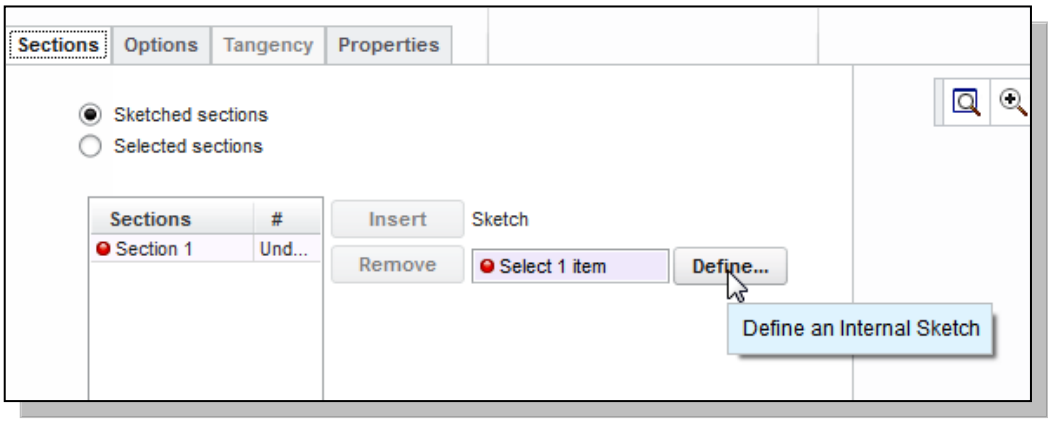
1. In the **Shapes** toolbar, select

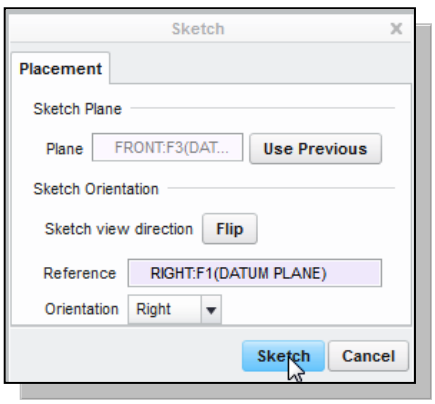
   Shapes→Blend



2. In the Dashboard, click on the **Sections** tab to display the available options.

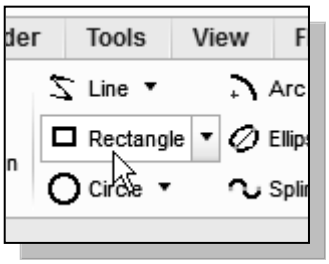
3. In the **Sections** tab, select **Define** to start a new sketch.





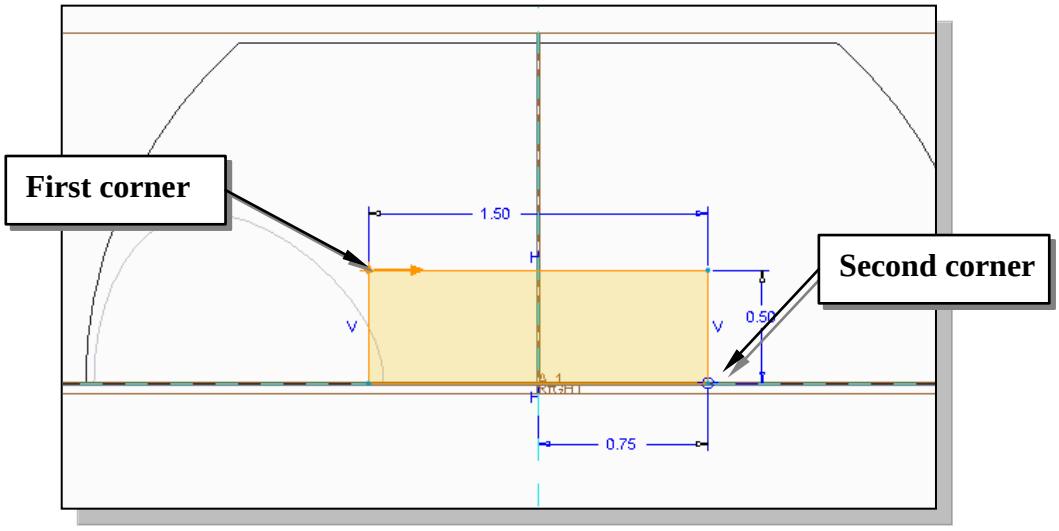
4. Pick datum plane **FRONT** as the sketching plane.

5. Click **Sketch** to accept the **RIGHT** plane to be used as the **Right side** orientation reference as shown.

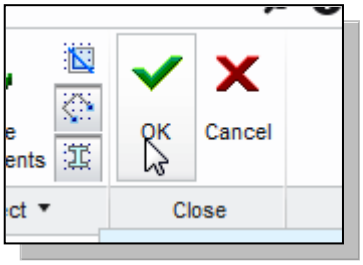


6. In the *Sketching* toolbar, click the **Rectangle** icon with the left-mouse-button.

7. Create a rectangle as shown; place the second corner of the rectangle on the horizontal axis.

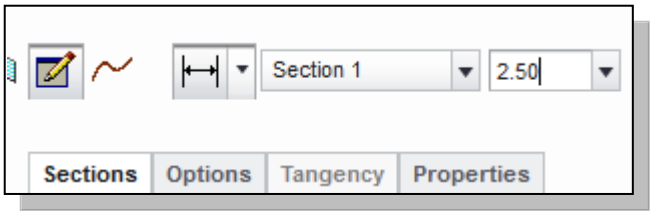


8. Use **Dimension** and **Modify** in the *Sketching* toolbar to adjust the size of the rectangle to **1.50** x **.50** and location to **0.75** as shown.



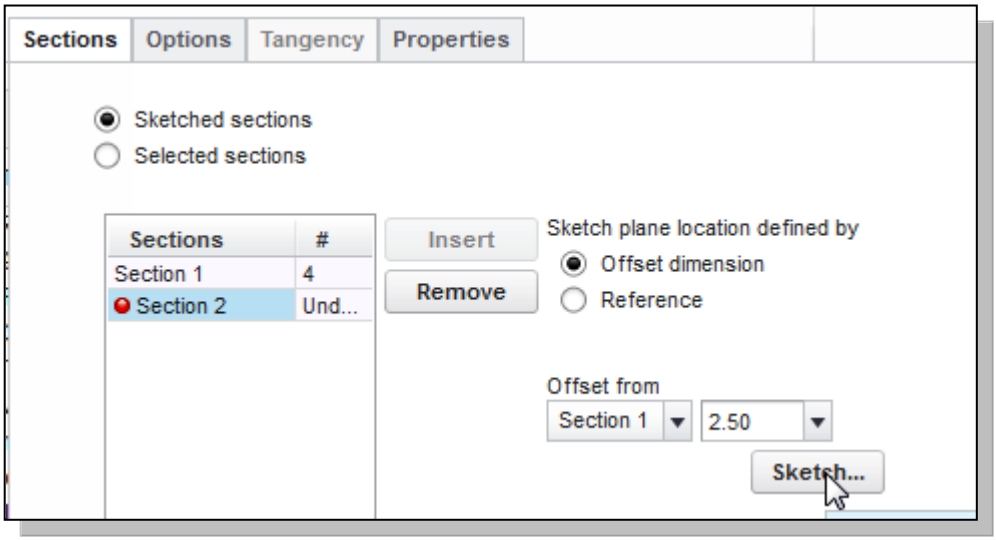
9. Click **OK** to accept the current sketch as the first section for the *Blend* feature.

- We have completed the first 2D section. We will next create the second 2D section. A *blend* feature is **a series of two-dimensional sections** that are joined together.

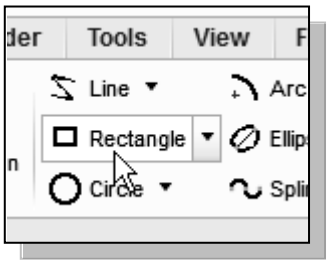


10. In the dashboard area, enter **2.5** as the offset distance for the second 2D section.

11. In the **Sections** tab, select **Sketch** to start a new sketch.

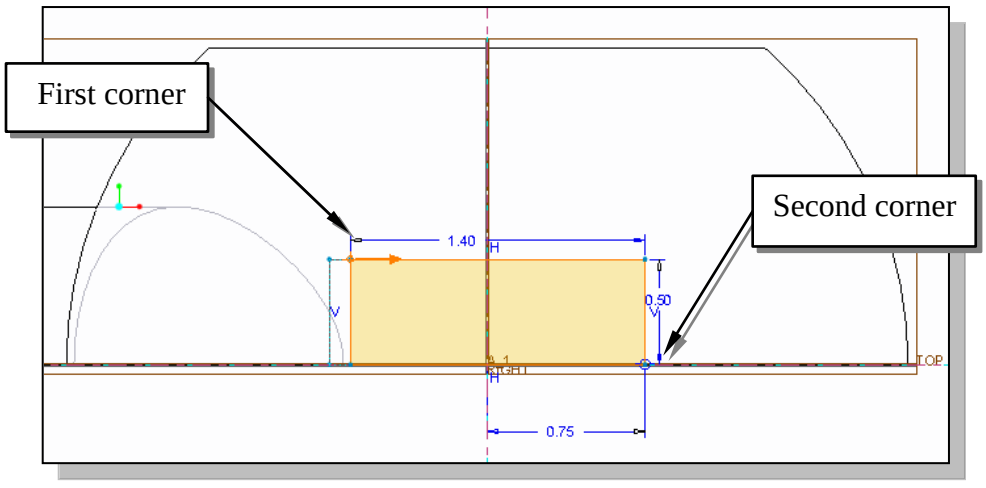


- The color of the previously sketched rectangle turned gray signifying that *Creo Parametric Sketcher* is ready to create the second 2D section.

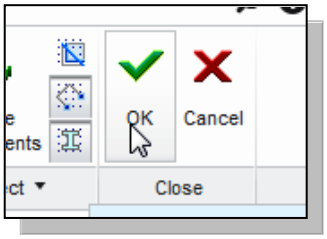


12. In the *Sketching* toolbar, select **Rectangle**.

13. Place the first corner of the rectangle aligned to the left of the origin and align the second corner to the horizontal axis as shown. The arrowhead indicates the direction of the blend. By placing the rectangle the same way as we did the first section, the blend will be aligned at the corresponding corners.

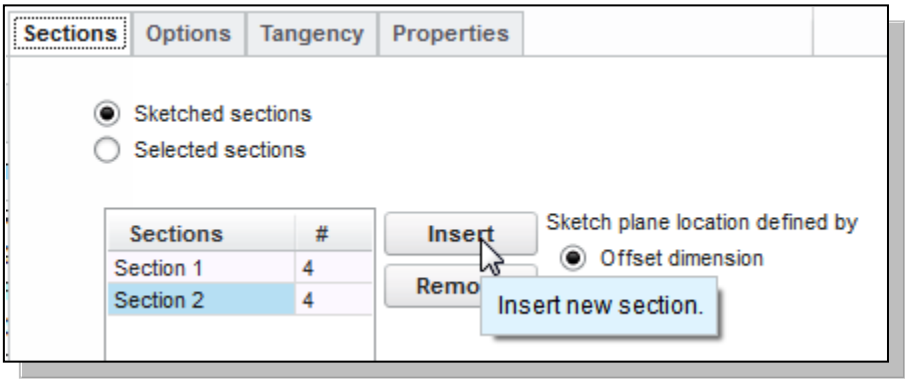


14. On your own, create and modify the three dimensions as shown.

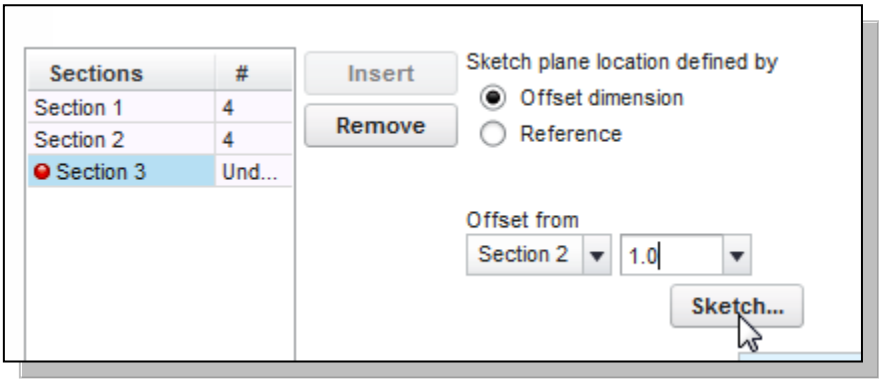


15. Click **OK** to accept the current sketch as the second section for the *Blend* feature.

16. In the **Sections** tab, click **Insert** to add another section to the *Blend* feature.

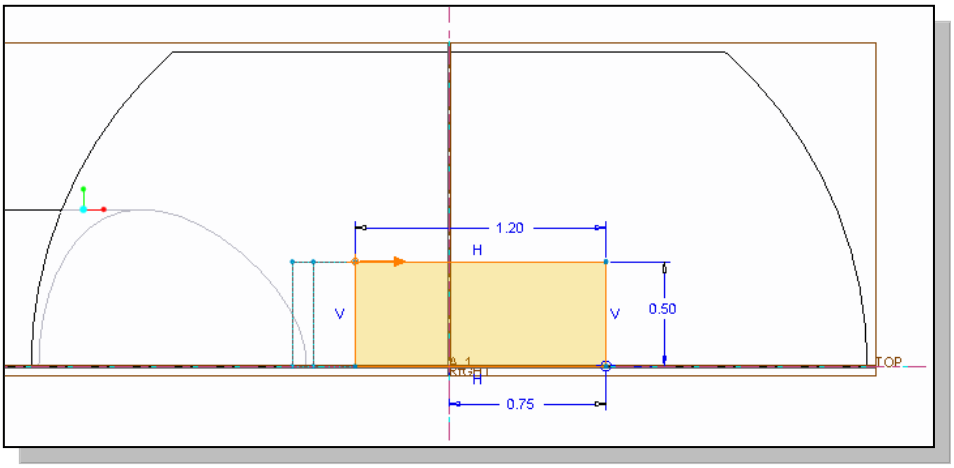


17. Enter **1.0** as the offset distance for the next 2D section.

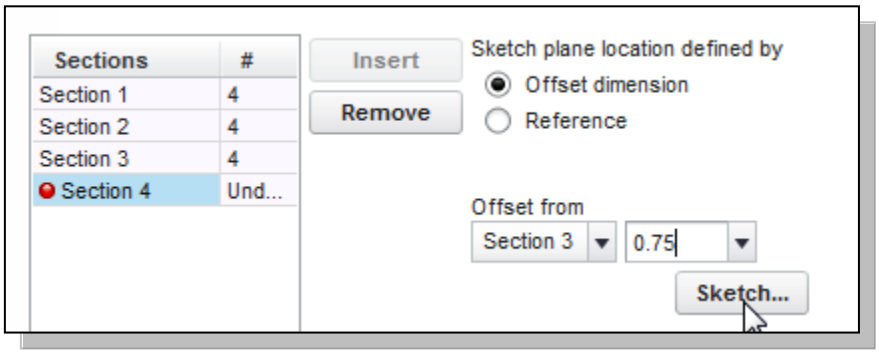


18. Click **Sketch** to enter the *2D Sketcher* mode.

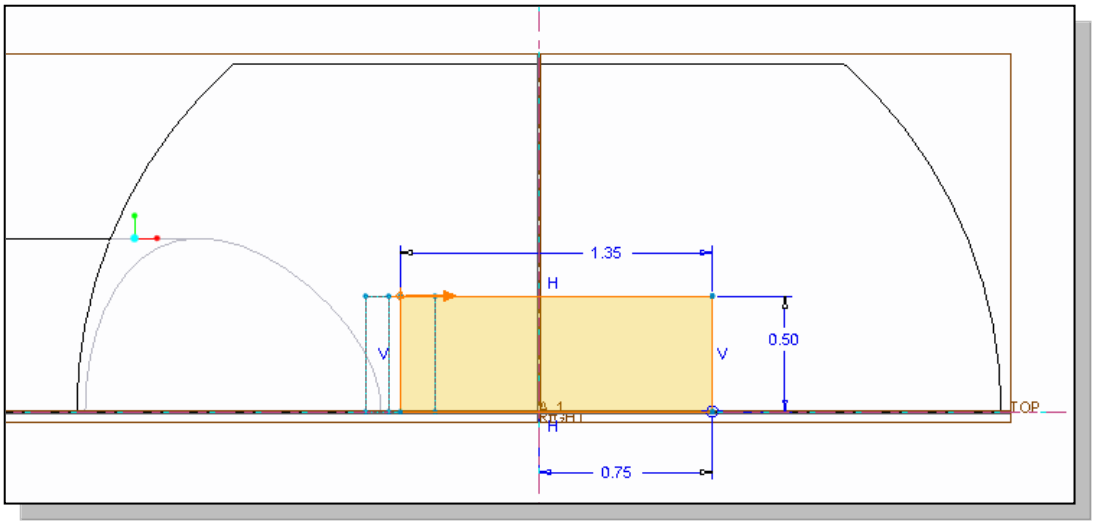
19. On your own, create the 2D section as shown.

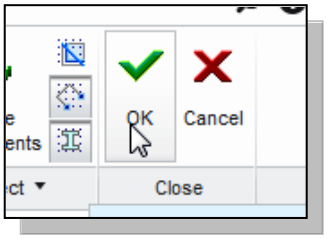


20. On your own, use the **Insert** option to add another 2D section with offset distance set to **0.75** as shown.

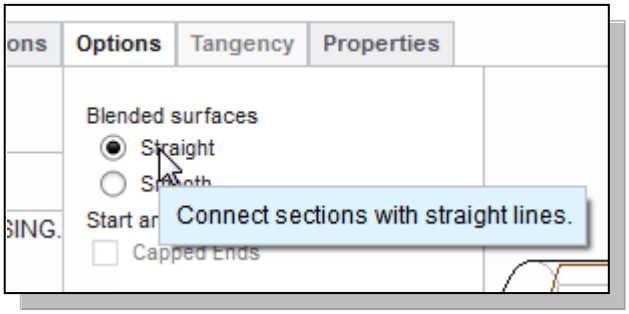


21. On your own, create the 2D section as shown.



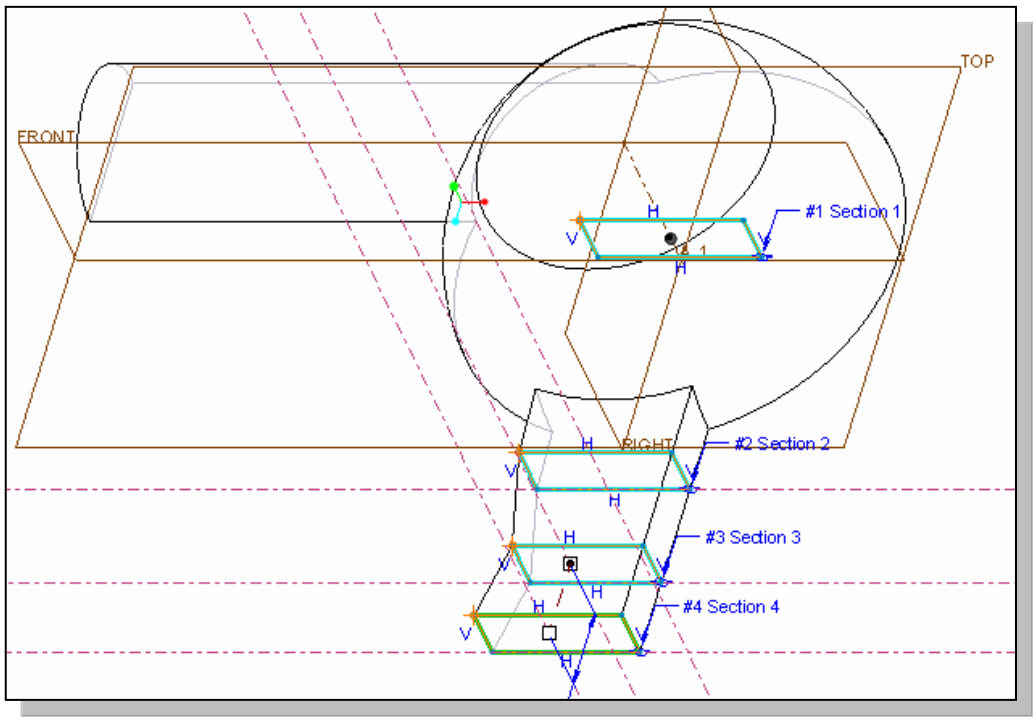


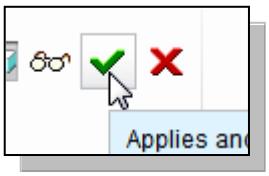
22. Click **OK** to accept the current sketch as the last section for the *Blend* feature.



23. In the *Options* tab, set the ***Blended Surfaces*** option to **Straight** as shown.

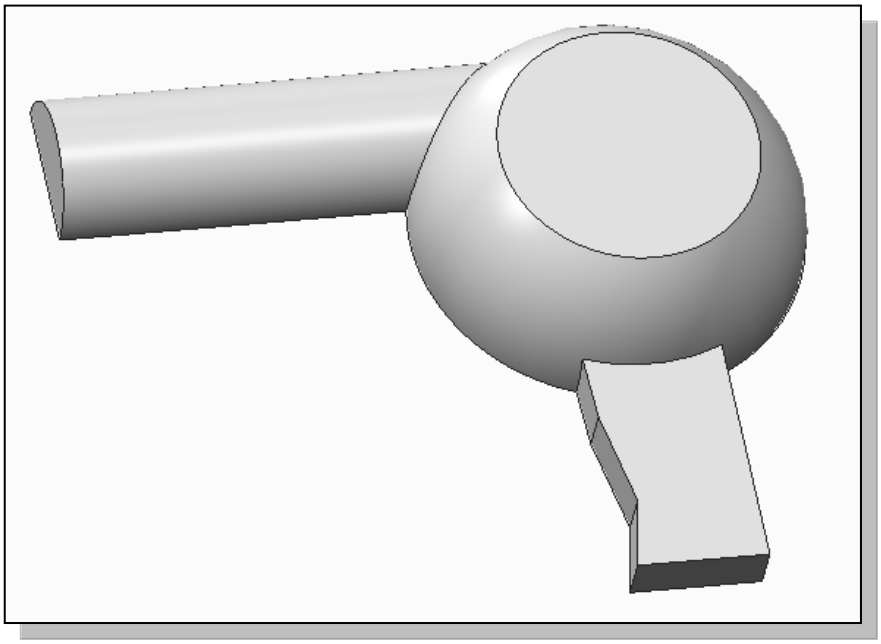
- Note that only ***one*** loop per subsection is allowed for the **Blend** operation.



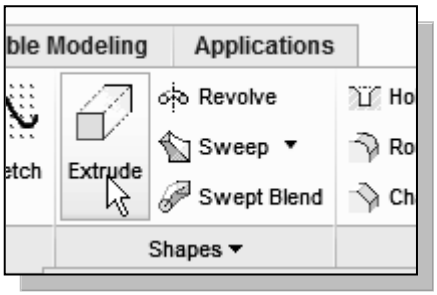


24. Click **OK** to create the solid feature.

25. On your own, use the *Dynamic Viewing* functions to view the completed 3D solid feature.



Page 9-4: Creating the Base Feature



1. In the *Shapes* toolbar, select the **Extrude** tool option as shown.