Tutorial Material for

Pro/ENGINEER Wildfire 2.0
With Integrated Pro/MECHANICA

Tutorial 1: Basic Modeling

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Extrusions, Sketching, and Cuts

Introduction

In this session, you will make a simple Pro/E part, and practice the basic modeling technique: Extrusion, Sketching and Cuts. Extrusions are one of the most basic ways to design a part in Pro/E, and sketching is an important part of the design process.

Starting Pro/E and Creating new Part

1. Start Pro/E Wildfire.
2. Click menu: **File → New**. A dialog will pop up as shown in **Figure 1**.
3. Give a new name and/or description if desired.
4. Click **OK** button and Pro/E will create an empty solid part as shown in **Figure 2**.
5. You may want to take a look of the coordinate system in Pro/E

![Figure 1. “New” Dialog](image)
Sketch and Extrude

To make a part, the first step is to draw a sketch, then extrude it to form a solid part. Remember, there are number of ways to do one thing in Pro/E. Here only shows on method.

As shown in Figure 2:

1. Click the “Front” Reference plan. This is where you draw a sketch. You can also go to step 2 first, then chose the reference plane.
2. Then click the Sketch tool. All the information should be filled up (Figure 3)
3. Click Sketch button in Figure 3 if you want to accept all the inputs.
4. Now Pro/E is in the Sketch mode. This will allow you to sketch in the XY plane and extrude in the Z direction.
5. You can close the References window by click close button.
FYI: Insert and Sketch are the most useful tools. Some of them have a speed button in the Toolbar region.

Figure 3. Start a sketch: Options

Figure 4. Start a sketch
6. Click menu **Sketch \(ightarrow\) Options**, check **Grid** and **Snap to Grid** options. Click the Green Check button: ![Green Check button](Figure 5)

7. The **Sketch** menu can be floating beside main Pro/E window (Menu Manager) by un-check the menu item: **Sketch \(\rightarrow\) Intent Manager**, you can choose either way you want to work with. **Intent Manager** is the default setup for Pro/E. **I will use Intent Manager**.

8. Choose the **Line Tool** \(\rightarrow\) to make a sketch. To choose the **Line** tools, you can directly click the **Line Tool** in the **Sketcher Tools** region, or click menu **Sketch \(\rightarrow\) Line \(\rightarrow\) Line.** (If you choose to use the Menu Manager, you need to choose Line from the Geometry, and use corresponding tools to draw a line)

![Figure 5. Sketcher Preferences and Menu Manager](image-url)
9. Make a similar sketch using **Line Tool**. When you are sketch, Pro/E will display all the information in real-time.

10. Placing a new point by clicking **Left mouse button**, end it by clicking **middle mouse button**. Sketch as shown in **Figure 6**. If you make a mistake, you can click the **Arrow** to end the **Line** tool, then chose the line and delete it. (Ctrl + Z to undo the last step. Or Click the undo symbol in the toolbar)

11. **Zoom in / out** using the Wheel on the mouse or by moving the mouse up and down while holding CTRL and middle mouse button.

12. **Move the sketch** on the screen by moving the mouse while holding the **middle mouse button** and the **Shift** key.

13. The dimensions will be properly displayed. If you want to move the annotation of a dimension, drag the annotation and move it.
14. If you want to change the dimension, you can drag the Lines or change the numbers by double click on the dimension itself. (Figure 7)

15. After making the sketch, click Checkmark (Figure 8) to finish the sketch.
16. **Rotate the sketch** by moving the mouse around while **Holding middle mouse button down**. Now you have a sketch in the 3D space (**Figure 9**).

17. As shown in **Figure 9**, select **Sketch 1** first, then Click the **Extrude tool**.
18. Type in **200** into the extrusion depth box on the dashboard. *(Figure 10)*
19. Click check button to finish the extrusion. You will have a 3D part.
20. The **Model Tree** can be used to manipulate features you have already created.
   Right click on the part you wish to alter, and select corresponding commands from the list.
21. Save the part by click **File \rightarrow Save**, or click the disk icon in the tool bar region. Click OK button.

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**Creating Cuts**

Creating cuts is somehow similar to the extrusion, even the procedure is similar. The difference is that cut will remove the material instead of building up the part. In this part, a sketch will be first made on surface of the part, then choose extrusion tools to make the cut.

1. Open the **View manager**, *(Figure 11)*, Click **Orient** page if that is not the default.
2. **Double click** the **Default orientation**, as shown in **Figure 11**.
3. Select the surface as shown in **Figure 11**.
4. Click the **Sketch tool**.
5. Accept the default values, click **Sketch** button.
6. Use **Line tool** to draw the profile of cutting. To see the profile better, change the view style to **hidden line wireframe**.

7. Finish the sketch by click the Checkmark.

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**Figure 11. Choose the sketch plane**

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**Figure 12. Make a Sketch on selected surface**
8. As shown in Figure 13(1 to 4), Choose the Sketch 2 from Model Tree, click Extrude tool, then click material removal button on the dashboard, change the cutting direction if necessary, make it towards the block. Type the cutting depth if necessary.

9. Click the Checkmark to make the cut.

10. Rotate the part to view the cutting result.

11. Save your part file.

More modeling tutorials can be found on the internet. Here is one link:

http://www.me.cmu.edu/academics/courses/NSF_Edu_Proj/Wildfire_short_course/tutorials.htm

Note: Pro/E Wildfire 2.0 is slightly different then the tutorial materials from the above link, but they are still very useful if you want to learn more about Pro/E.
Practice Problem

1. Create the part shown in Figure of Problem 1 using an extruded protrusion.

   Figure of Problem 1. Dimensions

2. Add a cut to the side as shown in Figure of Problem 2.

   Figure of Problem 2. Dimensions