

For this homework, read and work through Lesson 2 in your text ProEngineer Tutorial. During this lesson, you will create a part with a single extruded protrusion. Before you launch Pro/E, make a new sub-directory in your personal directory titled "HW2". Be sure to set your working directory to HW2 immediately after launching Pro/E. When you create this part, your text suggests that you name it "block", but instead, you MUST name it *****-hw2 where the asterisks indicate your GW netID.

Since this is a tutorial homework, each individual must perform this work independently. You are encouraged, however, to discuss the assignment with other students as necessary. It is recommended that you study the Questions for Review and create some of the "Exercise" models at the end of the chapter. After completing the lesson, answer the following questions and bring this Credit Sheet with you to class on the due date.

Problem 1.

What is the purpose of the sketching reference?

Solution

Problem 2.

Summarize the implicit rules of sketcher.

Solution

Problem 3.

Where are your Pro/E files stored when they are saved?

Solution

Problem 4.

What is meant by a "blind protrusion"?

Solution

Problem 5.

When you look at a sketch, in which direction will a solid protrusion occur?

Solution