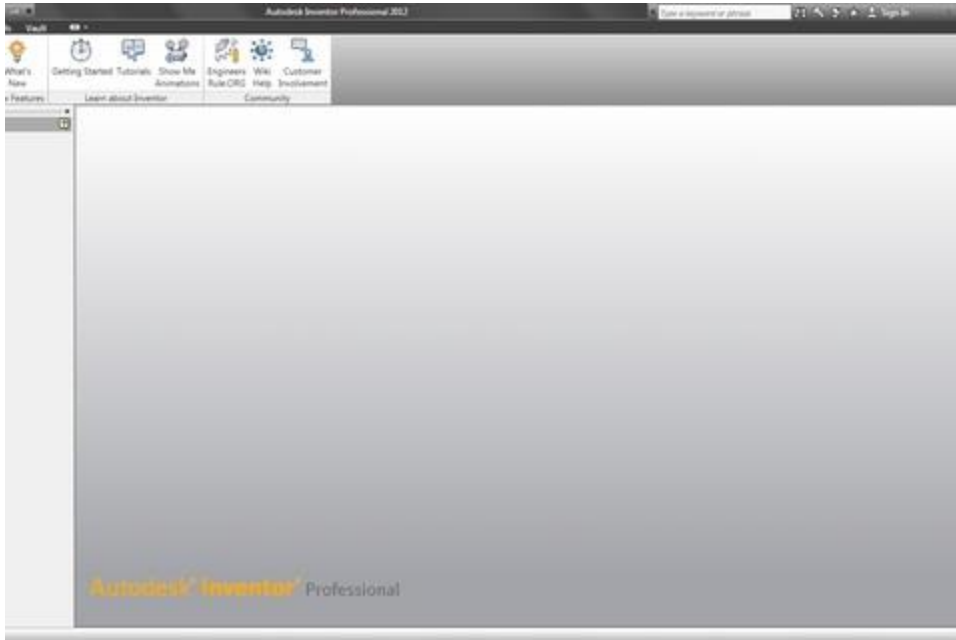


INTRODUCTION TO AUTODESK INVENTOR

ADV CADD 40S

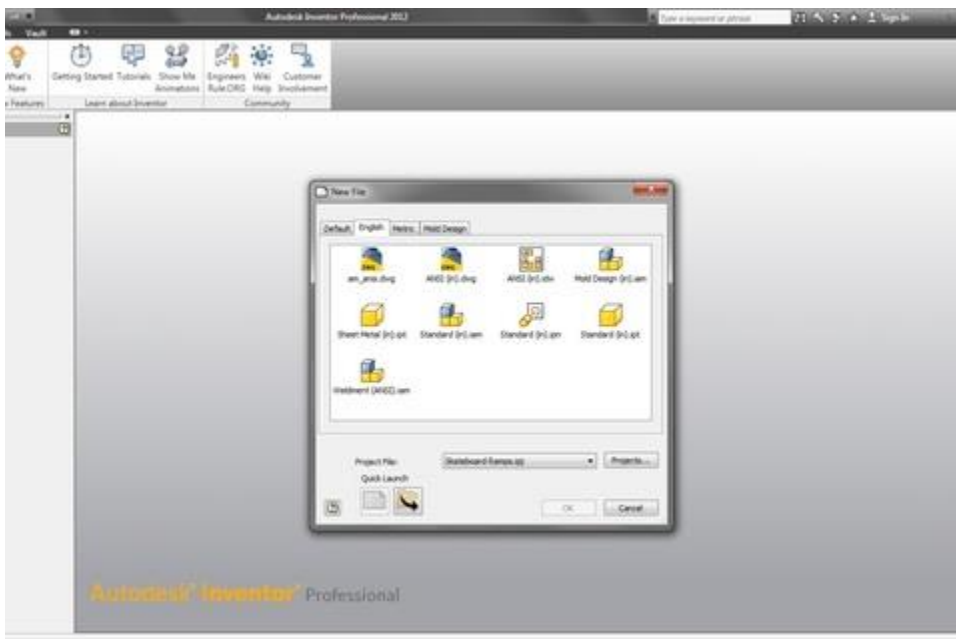
1. STEP 1

Step 1: When you first open Inventor you will see this start-up screen. Click on new to start a new part file.



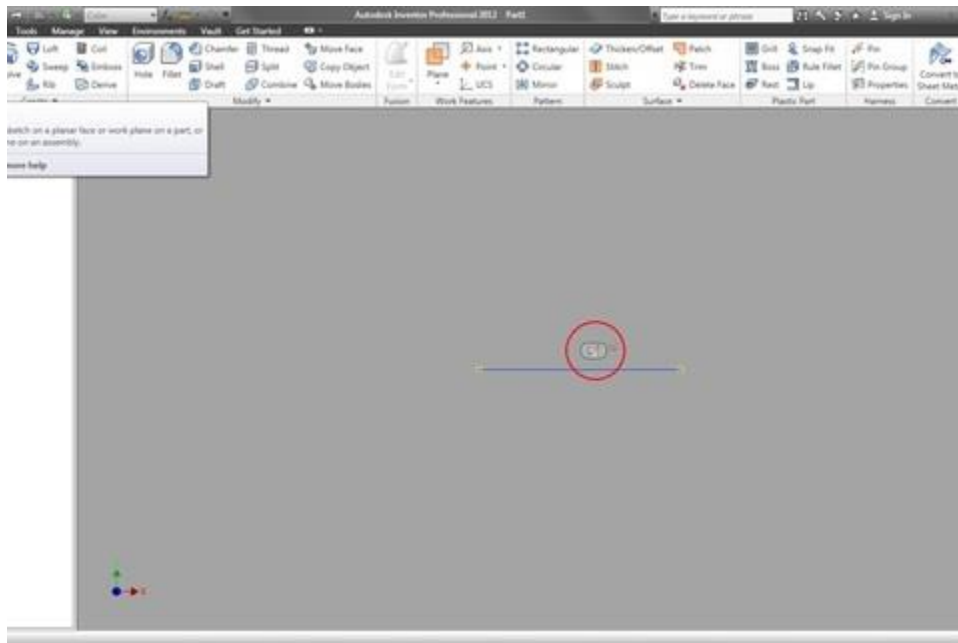
2. STEP 2

Step 2: You will notice that there are MANY options to choose from. You can create an AutoCAD .dwg file in both inch and metric, default setups for weldment and sheetmetal parts, and your basic part and assembly files. There is even an option to set up a specific path for a project folder. This does come in handy when you are dealing with multiple projects. For this tutorial I will choose the Standard(in).ipt which will create a part file in inches.



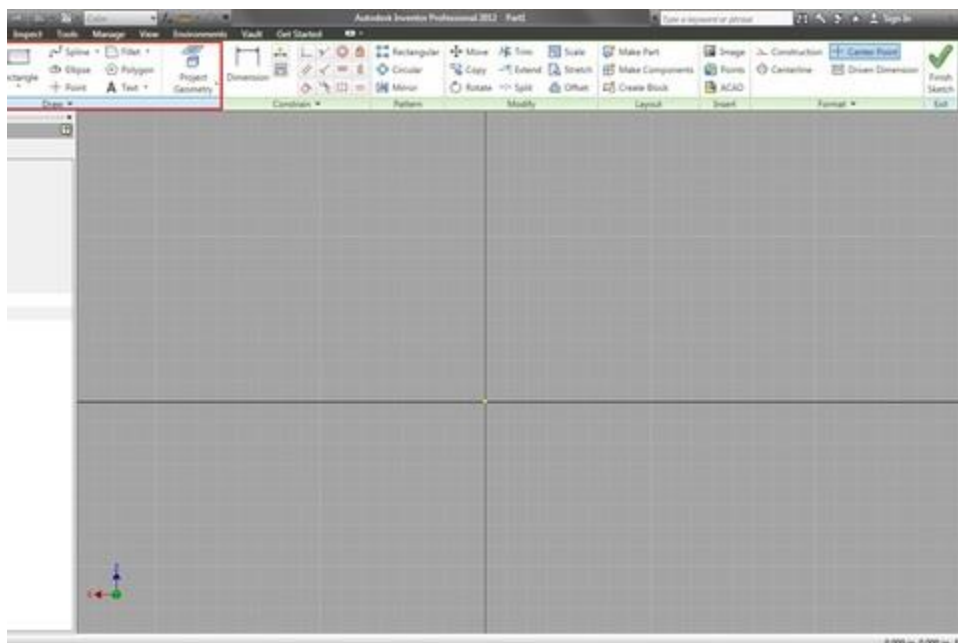
3. STEP 3

Step 3: Next we will create a sketch on a plane. The 2d sketch tool is located at the top left portion of the toolbar. You will notice that if you select a plane (xz plane selected) a quick sketch button will pop up. That will work as well to start a new sketch. The sketch section of the toolbar is self-explanatory. There are tools like spline, rectangle, circle, etc.; just like any other CAD program.



4. STEP 4

Step 4: Take notice of your new sketch showing up as "sketch1" in the bar to the left. This is where you can edit parent and child features to change the part. If that sounds complicated, don't worry. I will show you how to do that in another tutorial. You will also notice that the viewport has changed views to the plane that you chose to create the sketch. You can choose the view with the tools to the upper-right section of the viewport. Autodesk's classic wheel and somewhat new "view cube", as I call it, will let you snap to certain views; top, front, left, right, top-right corner, left-right corner, isometric, etc.

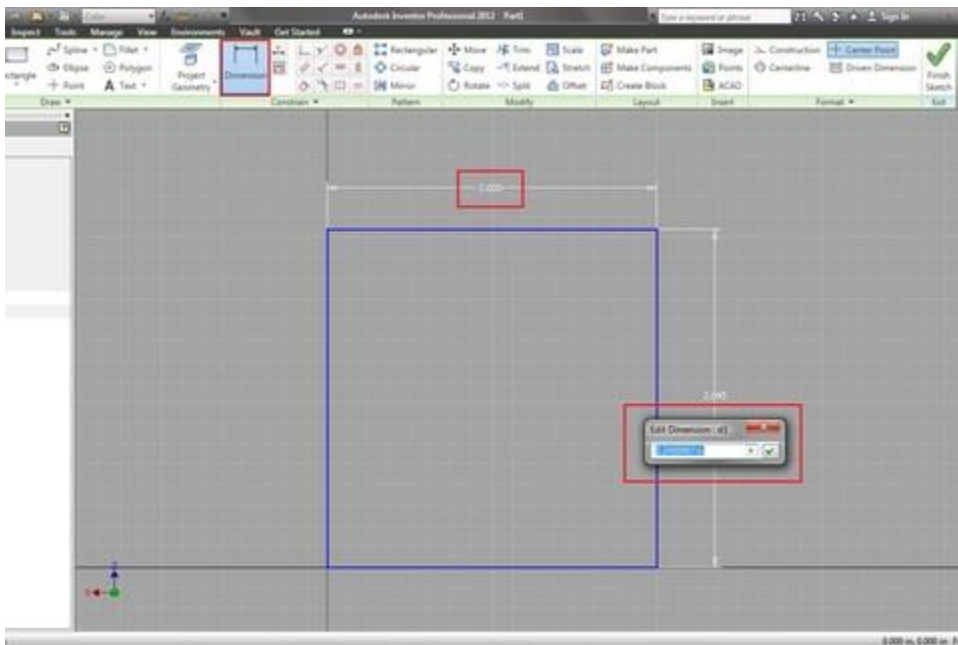


5. STEP 5

Step 5: Click on the rectangle tool in the sketch section of the tool bar. You now have to pick two points to create the rectangle. I have selected the origin (where all 3 axis intersect). Drag the second point out to approximately two inches and click to create the rectangle. This does not have to be exactly two inches, and I will show you why.

6. STEP 6

Step 6: We will now give our sketch dimensional constraints. To the right of the sketch section of the tool bar will be the dimensioning and constraint tools. I will cover constraints in the next Inventor tutorial. Right now we are simply going to give our sketch depth and width dimensions (remember we are on the xz plane) Click the dimension tool. You can create a dimension by selecting the whole line, the end points of a line, or by selecting two parallel lines. The placement of your dimensions will determine where "auto-dimensions" are placed and which are stated when creating a 2d drawing of your finished part. When you place the dimension a small window will pop up where you can edit the dimension. Change both dimensions to two inches (2.000)



9. STEP 9

Step 9: There you have it! We have created a 2in by 2in by 2in cube. My next tutorials will go into constraints as well as extrude cut (which, obviously would be the opposite of extrude: cutting away material instead of creating it)

