



Autodesk Inventor 2008: Feature Recognition

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Thanks to continuous research and development being conducted at Autodesk labs as well as product improvement suggestions made by AUGI members through the Wish List, there has always been a technology drive and plenty of improvements in every release of Autodesk Inventor® software. The latest release, Autodesk Inventor 2008, is no exception. In this article, I will focus on the Feature Recognition tool inside Autodesk Inventor 2008.

Inventor Feature Recognition really helps us when converting neutral 3D CAD data such as STEP, IGES, or even SAT into actual Inventor models, which is fully feature-centric. Before we go ahead with the tutorial, I recommend that you download the tool Feature Recognition for Autodesk Inventor 2008 from the [Autodesk Labs website](http://Labs.autodesk.com/utilities/feature_recognition). (http://Labs.autodesk.com/utilities/feature_recognition)

The key objectives of this tool are to find and create features such as hole, extrude, fillets, chamfers shell, sculpt, sweep, and revolve within the model. The intended design features can be achieved either automatically or by an accurate step-by-step procedure, as described below.

Step 1: Start Autodesk Inventor 2008 and select Part Modeling.

Step 2: Open a part file, which has any one of the following extensions: STEP, IGES, or SAT.

Step 3: If you don't have a part file with these extensions, I recommend that you create such a file. Here I will be creating a feature, shown in Figure 1.

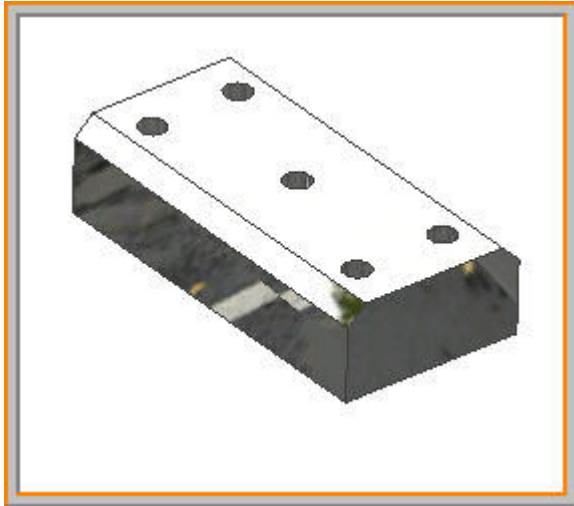


Figure 1: Part File

Step 4: Create a rectangle with sides 20mm x 40mm and extrude to a height of 10mm as shown in Figure 2.

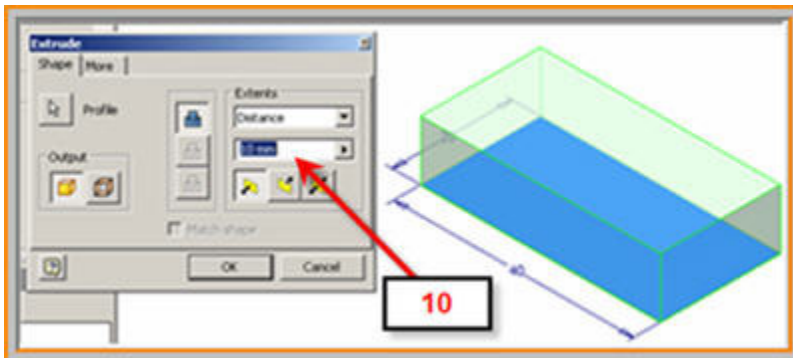


Figure 2: Extruding a Rectangle

Step 5: Insert five holes (diameter 3mm) using insert hole [H] feature option with dimensions as shown in Figure 3.

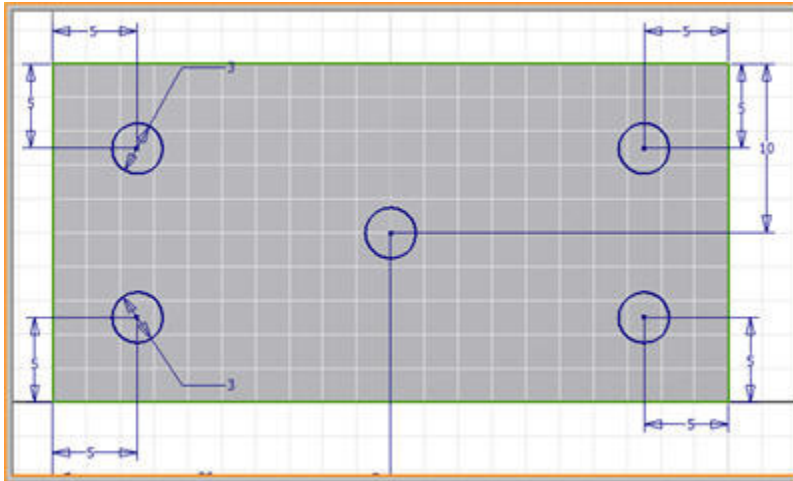


Figure 3: Dimensions

Step 6: Create a chamfer on both sides with dimension a 2 x 45° Chamfer as shown in Figure 4.

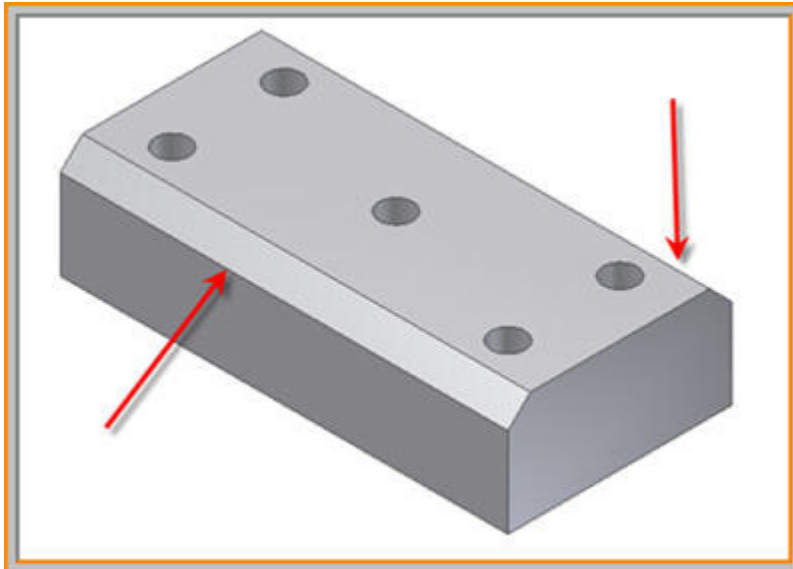


Figure 4: Chamfer

Step 7: Now save this part file as *.SAT (Save Copy As...) as shown in Figure 5.

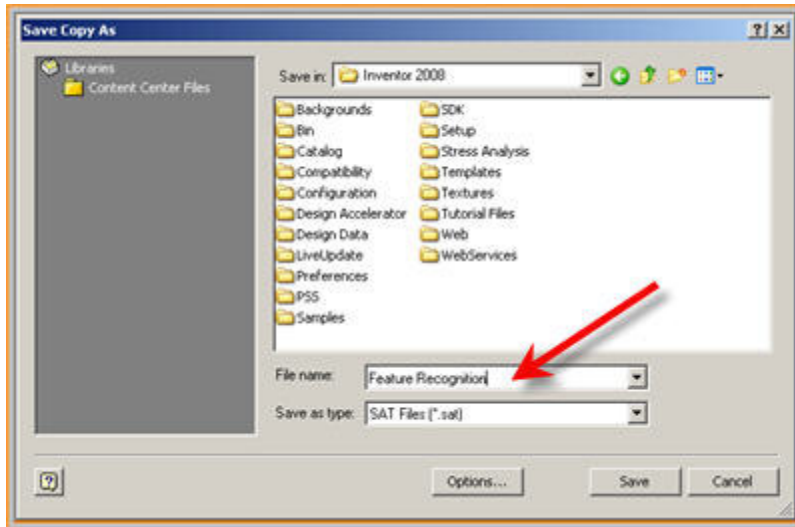


Figure 5: Save Copy As dialog box

Step 8: Now open the SAT file directly using the open dialog box and you will be prompted with the dialog box shown in Figure 6.

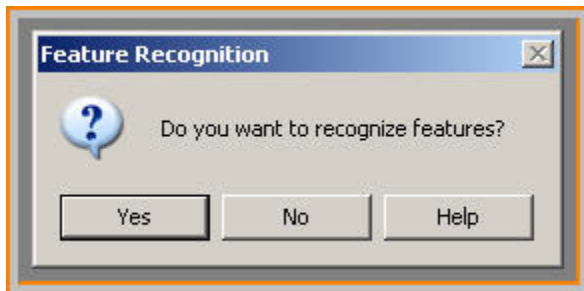


Figure 6: Feature Recognition dialog

Step 9: Now click on Yes and you will be taken into Automatic Feature Recognition module as shown in Figure 7. Click on Automatic Feature Recognition and you will be prompted via another window, shown in figure 8. Here you need to select the type of feature, which can be automatically recognized along with faces. Once all are recognized, they are individually displayed in the browser bar (Feature Recognition) as shown in Figure 9.

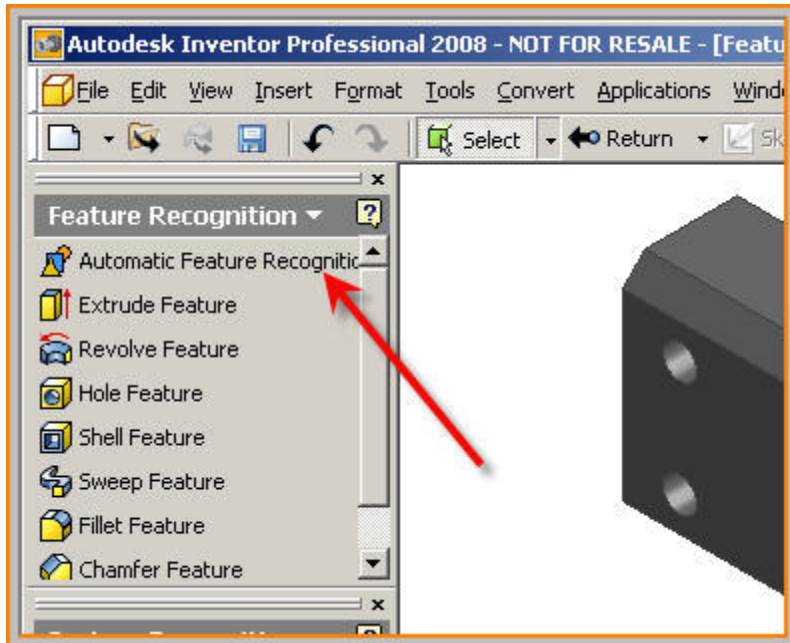


Figure 7

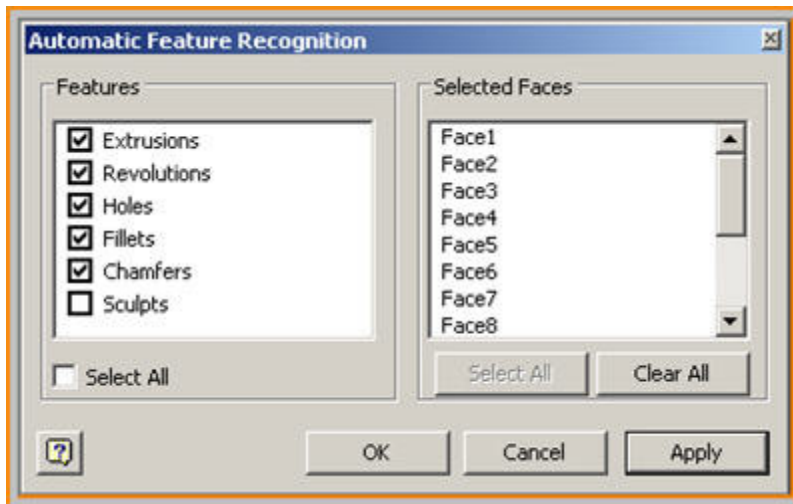


Figure 8

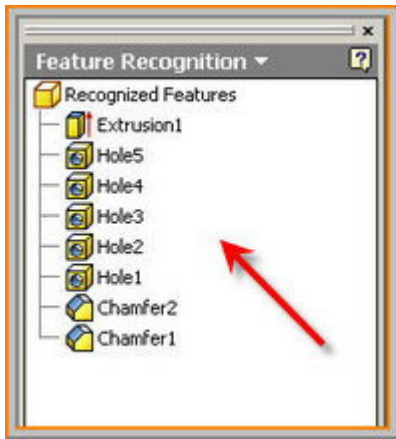


Figure 9

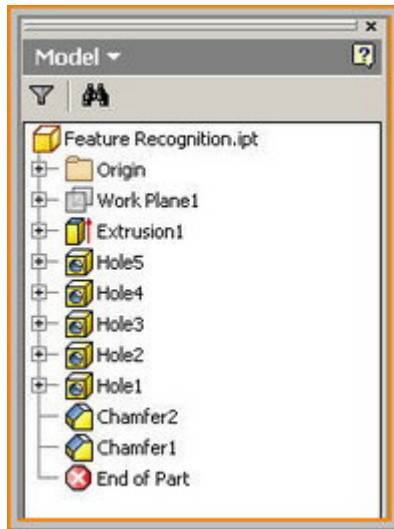


Figure 10

Step 10: Now click on Return in the standard tool bar and you will be prompted with: "There are some recognized features, do you want to create them?" Click on Yes and you will have the actual Inventor feature file. To justify this, try to change the extrusion height from 10 to 15 mm. If you are able to do it, then your part file is native Inventor format. Further change the color to Black Chrome. Figure 10 shows all of the operations. If you need to make some hole features as rectangular array, you can select those and execute them as a Rectangular Pattern, as shown in Figure 11.

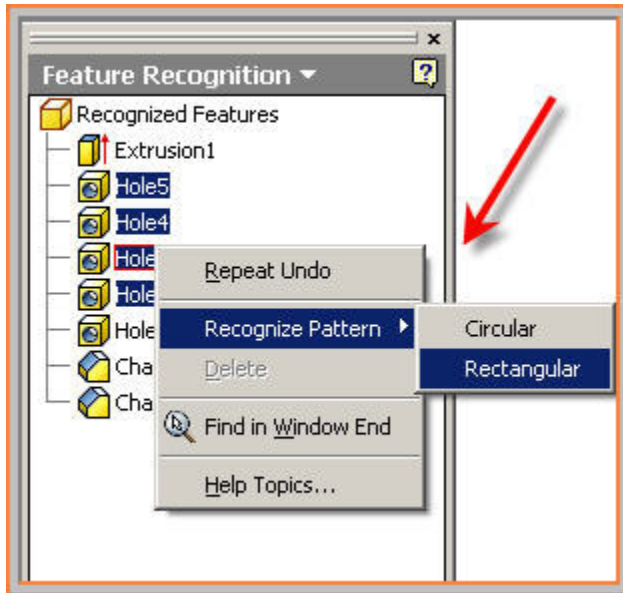


Figure 11

Note: If Automatic Feature Recognition does not work, it is safe to work on Interactive Feature Mapping.