



Autodesk Inventor Tutorials

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Introduction to iFeatures (aka Design Elements)

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In addition to this tutorial I suggest you check out a similar article by Neil Munro located at <http://pointa.autodesk.com/local/enu/portal/Articles/article.jsp?articleId=1692>

In this lesson we will discuss iFeatures (which were known in former releases of Inventor as Design Elements). iFeatures are "predrawn" shapes that you can place in a part model to speed up modeling. For example if your company uses a specific shape in a part very often, rather than drawing the shape over for each iteration of the part you can make it an iFeature. iFeatures also have the ability to be controlled parametrically to change size and shape. Finally iFeatures can be used alone in a part model or as additional features on an existing part model.

Let's first look at a simple iFeature. Let's say my company uses metal channel stock for a lot of applications. Rather than have to create this U shape for each part I want to speed up the process by making it an iFeature.

First open a new .ipt and start a sketch. Draw the shape shown below. Be sure to notice that we define the thickness (0.25") only once and use parametrics to define the other two dimensions.

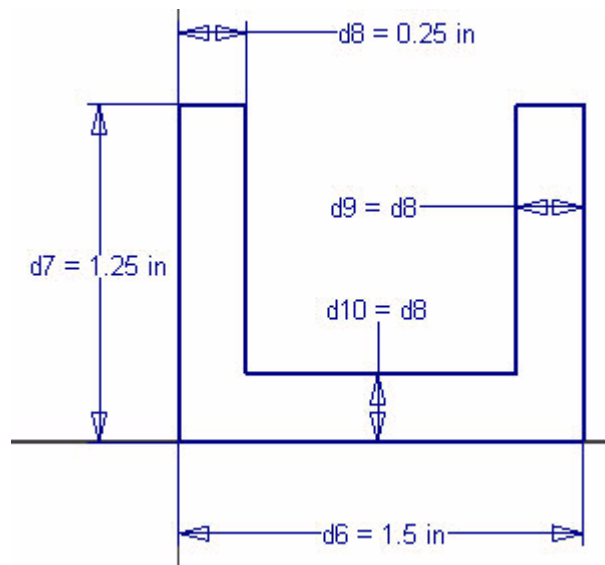


Figure 1 - Basic Sketch

Next extrude the part to some length. The length is unimportant at this point unless this is a feature that you do not want to change. You should have something like Figure 2. (Remember that your dimension #s may differ from the ones shown here)

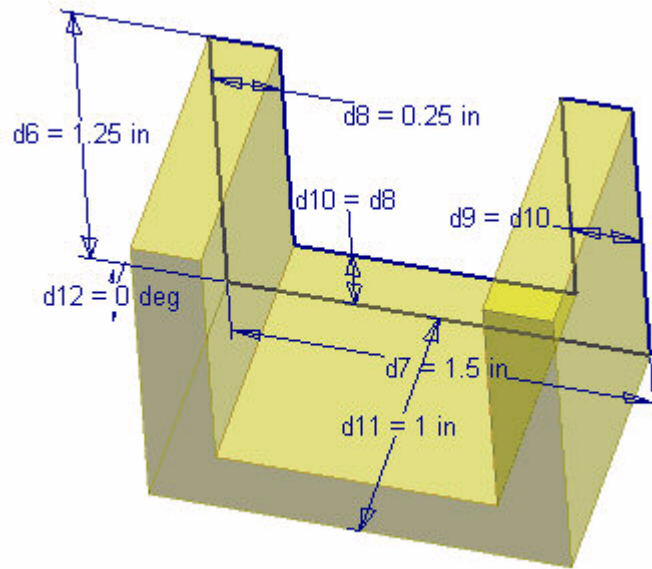


Figure 2 - Basic Extrusion

Next click on the Create iFeature button in the features toolbar (or in the panel bar).



Figure 3 - Location of the Create iFeature Icon

You will be presented with a dialogue box like the one in Figure 4. This is similar to the iPart Author dialogue box. At first you will not have anything in the left hand side of the box. Click on the extrusion you just created in the browser and it will appear in the left hand side of the dialogue box.

Select the parameters you want to be able to vary in the left side and move them to the right via the arrow buttons. Be sure to put them in the order in which you want them presented to the user. In our example we have chose to vary the length, width, height and thickness of our part. (see Figure 5)

You can now rename the "d" parameters to something more intuitive. We have changed the "d" parameters to Width, Length, Height and Thickness. (see Figure 6)

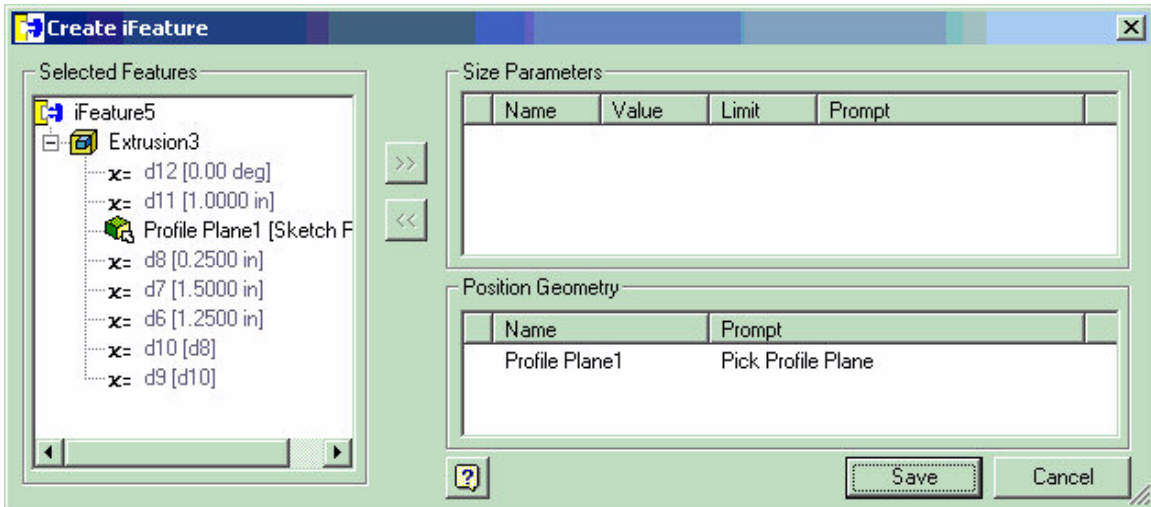


Figure 4 - Create iFeature Dialogue

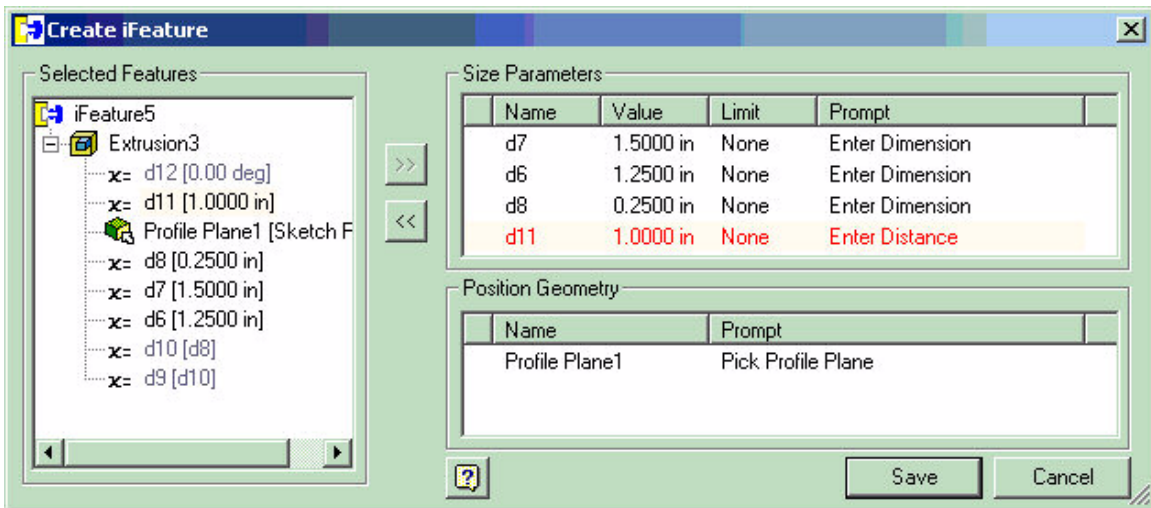


Figure 5 - Selecting Parameters in Dialogue

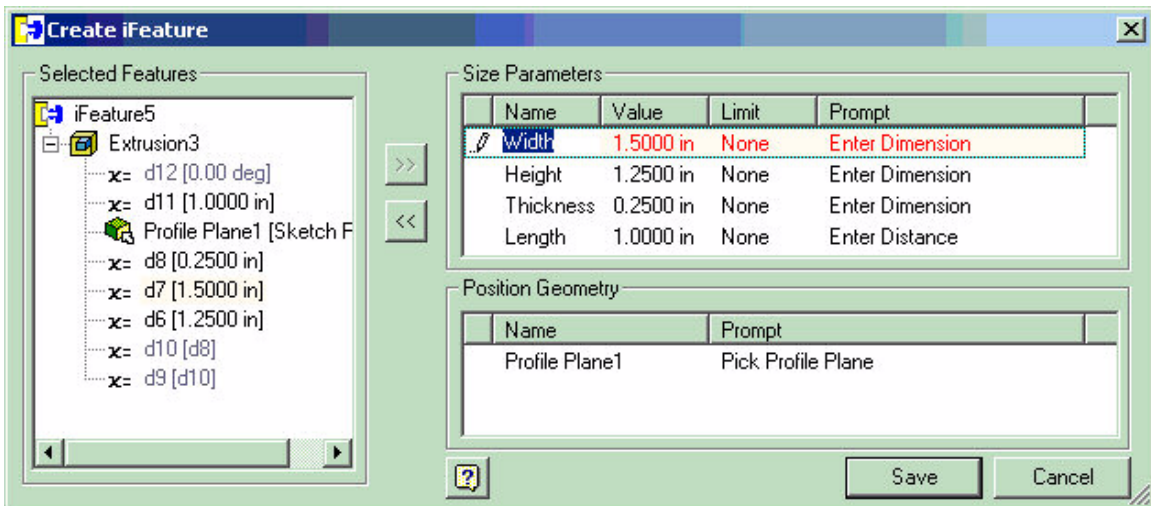


Figure 6 - Renaming Parameters

You may also change the prompt in the far right side of the dialogue box.

Now it's time to set the limits of our iFeature. If you want the user to be able to dimension this part in every possible fashion you leave all of the Limit values set to None. In our example we want to be able to constrain the user to only vary the dimensions within some limits or from a list of predefined values. Choose the first parameter and select List from the Limit pull down.

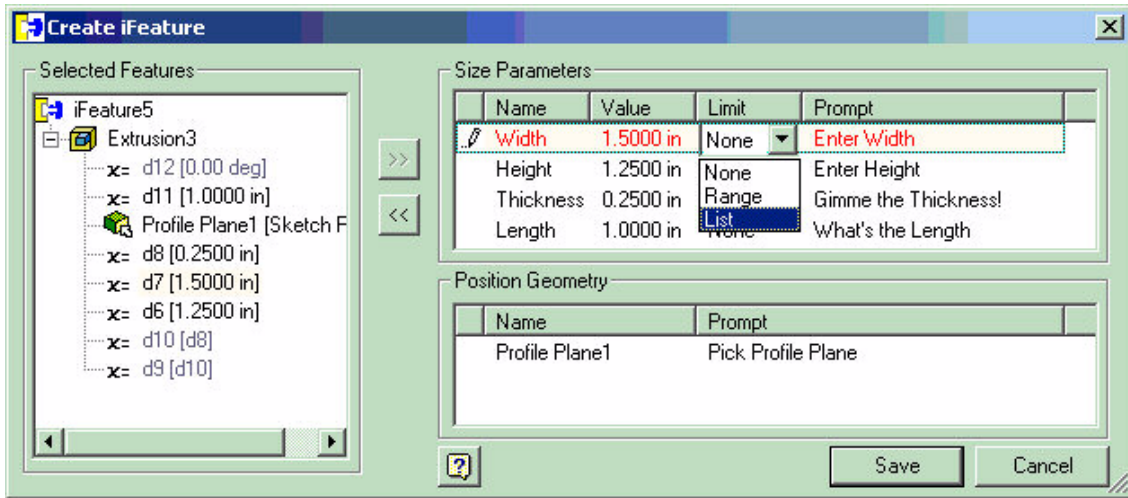


Figure 7 - Setting Parameter Limits

You will be presented with a new dialogue box that allows you to enter the list of values that are valid for this parameter. You may also set the default value for this parameter. (see Figure 8)

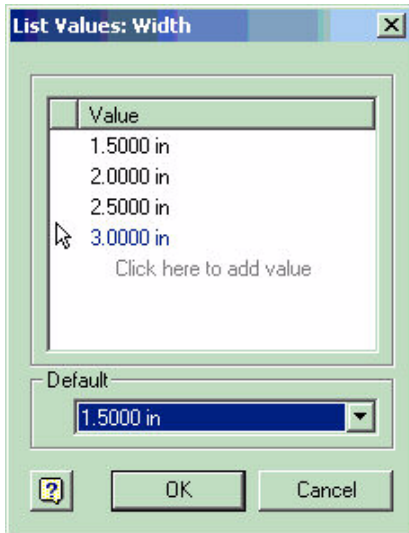


Figure 8 - Setting List Values

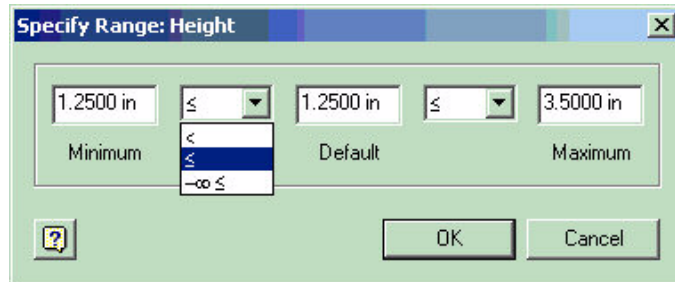


Figure 9 - Setting Range Values

Return to the iFeature Creation box and set the limits of the next parameter to Range. (see Figure 9)

Here you may specify the range of values that are valid for this parameter. Take note of the infinity signs. These allow you to include all values above or below a set value.

In our example we set the Thickness to a range as well but leave the Length unrestricted. This is the finished iFeature Creation table

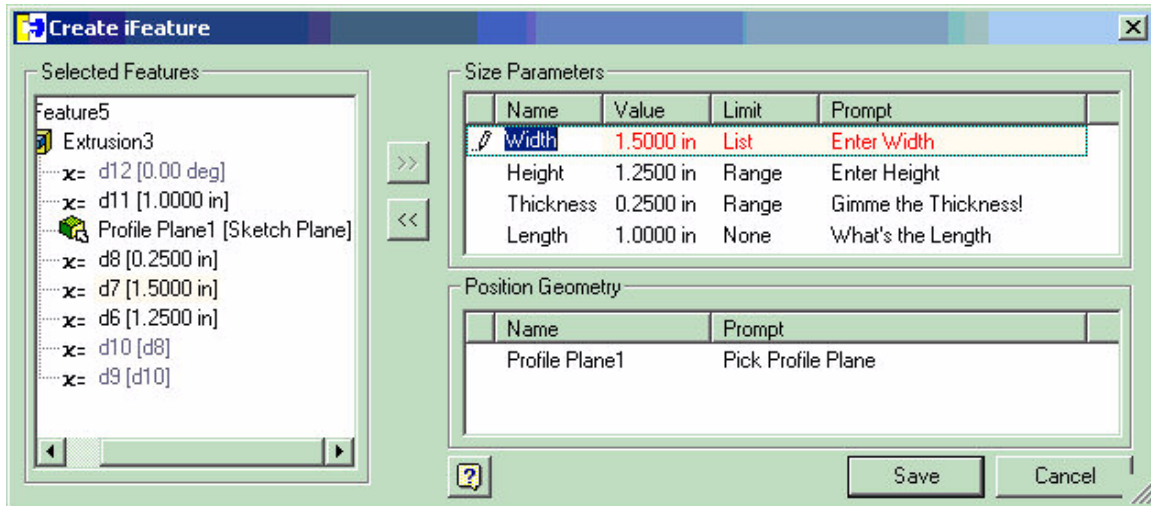


Figure 10 - Completed iFeature Table

Before you can save you must also choose a profile plane that the feature is based on. Click on the Profile Plane text in the lower dialoure box and choose the original sketch plane in the browser. This will allow you to place the design element on a workplane in addition to a existing features.

Choose save and save the iFeature in the

C:\Program Files\Autodesk\Inventor 5\Catalog\

directory (or to wherever you have set the Design Data Path in Application Options)

You can now either save or disregard the part file. You can disregard it as the data has been saved in the iFeature file and the part file is no longer needed.

You can now start a new part file and insert you new iFeature. The illustration below shows the list of values that were imposed on the Width parameter

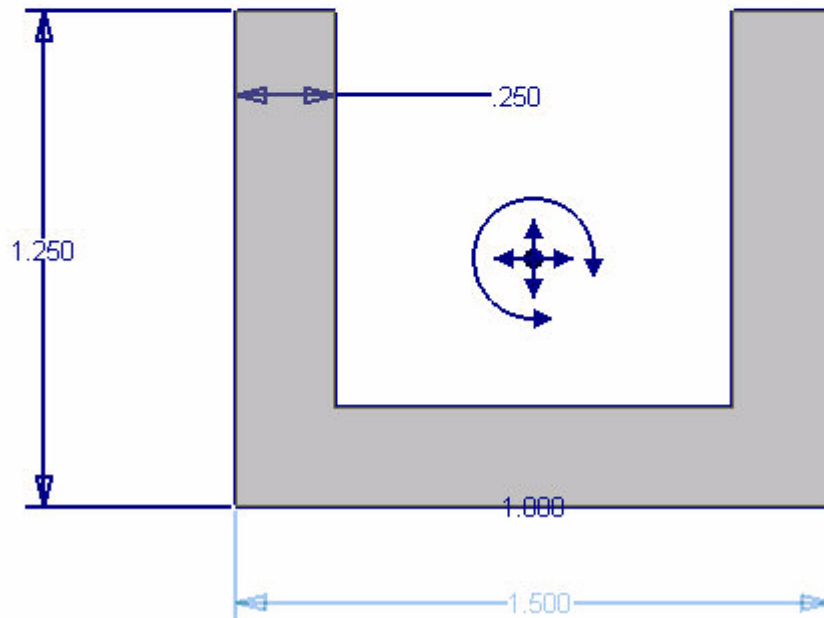
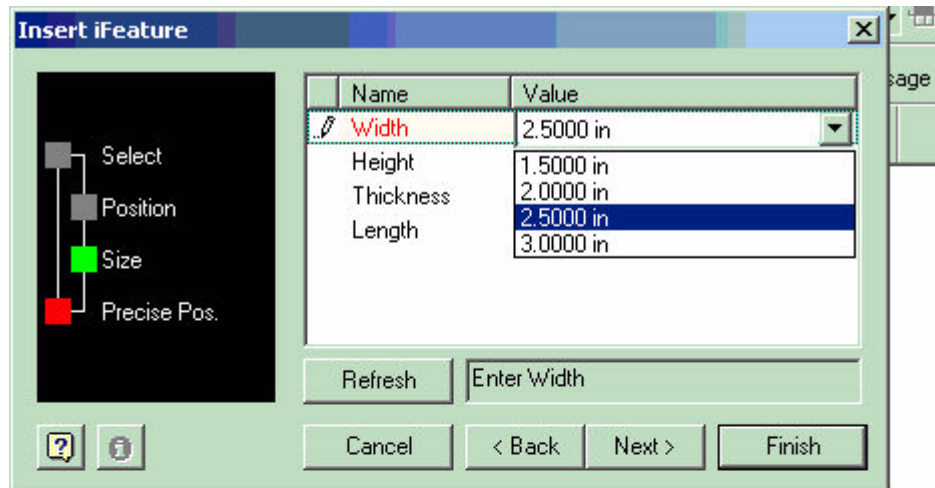


Figure 11 - Inserting iFeature

After selecting other values for the remaining parameters you should have a completed iFeature.

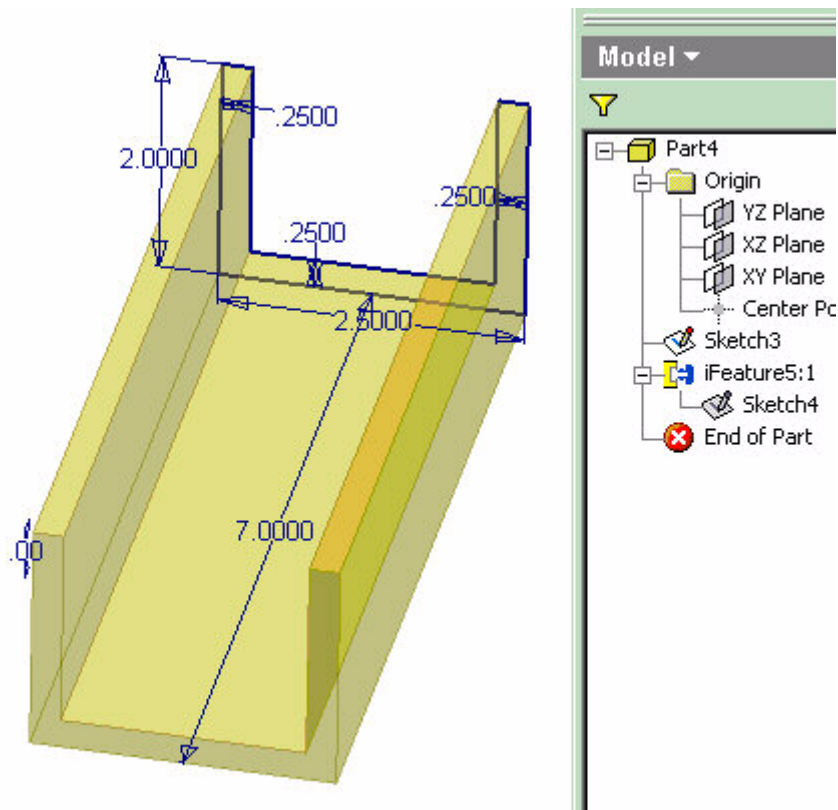


Figure 12 - Inserted iFeature

iFeatures may also be used to remove material. (Like the punch tools in the sheet metal mode). To do this simple start your iFeature part with an extruded block. Cut the shape you desire out of the block. When you select the features to be included in your iFeature simply deselect the basic block with which you began. When you insert this iFeature it will then cut into the existing geometry.

When using a “cut” iFeature take note of the way you terminate the “cut”. When creating the part you must determine if you will always want the cut to go through all of the feature or only to a certain distance. This will determine if you need to include the depth of the cut as a parameter in your iFeature.