

**Autodesk Inventor Tutorials** 

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## Introduction to iFeatures (aka Design Elements) Latest Revision: 5/06/02

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

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 is 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)

Create iFeature	Size Parameters		×
IFeature5         Image: Features5         Image: Features5	Name Value      Name Value      Position Geometry	Limit Prompt	
	Profile Plane1	Prompt Pick Profile Plane	Cancel

Figure 4 - Create iFeature Dialogue

🔁 Create iFeature						×
Selected Features	1	- Size Paramete	ers			
🔁 iFeature5		Name	Value	Limit	Prompt	
Extrusion3		d7	1.5000 in	None	Enter Dimension	ST 74
<b>x</b> = d12 [0.00 deg]	>>	d6	1.2500 in	None	Enter Dimension	
		d8	0.2500 in	None	Enter Dimension	
🕂 🦓 Profile Plane1 [Sketch F		d11	1.0000 in	None	Enter Distance	
<b>x</b> = d8 (0.2500 in) <b>x</b> = d7 (1.5000 in)		- Position Geon	netry			¥
<b>x</b> = d6 [1.2500 in]		Name		Prompt		
<b>x</b> = d10 [d8] <b>x</b> = d9 [d10]		Profile Pl	ane1	Pick Pro	file Plane	
		2			Save	Cancel

Figure 5 - Selecting Parameters in Dialogue

C	Create iFeature							×	1
Ĩ	Selected Features		- Size	e Parameters				1	
	🔁 iFeature5			Name	Value	Limit	Prompt		
	Extrusion3	1	J	Width	1.5000 in	None	Enter Dimension		
	<b>x</b> = d12 [0.00 deg]	>>	1.000	Height	1.2500 in	None	Enter Dimension		
	<b>x</b> = d11 [1.0000 in]			Thickness	0.2500 in	None	Enter Dimension		
	🔩 Profile Plane1 [Sketch F	<u> </u>		Length	1.0000 in	None	Enter Distance		
	<b>x</b> = d8 (0.2500 in) <b>x</b> = d7 [1.5000 in]		-Pos	ition Geome	ry				
	<b>x</b> = d6 [1.2500 in]			Name	3	Prompt			
	<b>x</b> = d10(d8) <b>x</b> = d9(d10)			Profile Plan	e1	Pick Profi	e Plane		
			2				Save	Cancel	111

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.

Selected Feature	Ē	Size	Parameters			_	<u></u>
<ul> <li>iFeature5</li> <li>iFeature</li></ul>	~	Posi	Name Width Height Thickness Length tion Geomet Name Profile Plan	Value 1.5000 in 1.2500 in 0.2500 in 1.0000 in ry e1	Limit None Range List Prompt Pick Profile	Prompt Enter Width Enter Height Gimme the Thickr What's the Length Plane	ness!
	(	2				Save	Cancel

**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)

List Values: Width	Specify Range: Height
Value           1.5000 in           2.0000 in           2.5000 in           3.0000 in           Click here to add value	1.2500 in       ≤       1.2500 in       ≤       3.5000 in         Minimum       ≤       Default       Maximum        co ≤       0K       Cancel
	Figure 9 - Setting Range Values
Default 1.5000 in	
OK Cancel	

Figure 8 - Setting List 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

ature5			Name	Value	Limit	Prompt
Extrusion3	The second	J	Width	1.5000 in	List	Enter Width
<b>x=</b> d12 [0.00 deg]	>>		Height	1.2500 in	Range	Enter Height
<b>x</b> = d11 [1.0000 in]			Thickness	0.2500 in	Range	Gimme the Thickness!
😪 Profile Plane1 [Sketch Plane]			Length	1.0000 in	None	What's the Length
·x= ds [0.2500 in] ·x= d7 [1.5000 in] ·x= d6 [1.2500 in]		-Pos	ition Geomet	ry	Derect	
<b>x</b> = d10 [d8]			Drofile Dise	-1	Prompt Dial: Drafi	la Plana
<b>x</b> = d9[d10]			FIUIIIE FIAN	ei		

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

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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.