Autodesk Inventor Tutorials

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Tips & Tricks
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This section is intended to be a grab bag of tips and tracks for working with iParts. Many of them have come from readers like you who have found a neat way to overcome a problem. Some are from Autodesk designers, some from me. Take the all with a grain of salt, as I have not verified them all. Check back as the document will be updated from time to time.

1. "Error" Messages

This tip comes from Tom Sturtevant of Autodesk and concerns the "Active part does not match the table values" message.

I think the point of that "error" is generally misunderstood – in fact it's not an error, but an opportunity. Inventor assumes that the factory will be in sync with the default row in the table. It supports this in two ways:

1. When you edit the default row values in the table, the values in the default row are applied to the model. (This is handled slightly differently for edits via Excel or via Inventor's iPart Author dialog.)
2. When you modify values in the part, you have the option to "update the table before continuing".

This enables a workflow in which you edit the table by editing the factory part. Like this:
  * create a factory part
  * using iPart Author command add appropriate columns. Add all the rows you want, but only modify the KEY values. Exit iPart Author command.
  * in the browser, switch to a different default row.
  * edit the part so it correctly represents the current default row.
  * switch to another default row – it will ask "... Do you wish to update the table before continuing?" answer Yes!
  * repeat for all rows

This is probably more trouble than it is worth for many factories, but at some level of complexity it is easier to set a row configuration through Inventor editing than through table edits. For large tables you can do this for a few selected rows then use excel for the bulk replication. If you are modifying color or material for each row, you don't have to worry about spelling errors. Of course it's not quite a complete solution. I don't think it works for threads (yet). It won't work for file names. It should work for parameters, properties, feature suppressions, iMate values, and color and material styles.

Thanks to Tom for this tip! -Sean

2. Multiple colors in iParts

Someone wrote me the other day asking about multiple colors in iParts. For example he had a fitting but wanted the end of the fitting to change color for English or metric. This is possible but it takes a bit of forethought.
You need to make the features that will change color the default color. In other words don't change their color via the properties dialogue.
Next go back to the parts that will remain a constant color and using the RMB on the features in the browser, select properties and change them to whatever color you want. The premise here is that the iPart author can only control the "base" color, not the one that can be set via properties. It's kind of backwards from the way you would normally draw parts (make the non-changing parts the base color) but it works.

Click **here** to download the iPart shown below. In this example we set the base color to Green and vary the color of the cylinder by the iPart table. Changing the diameter of the cylinder changes the cylinder's color.
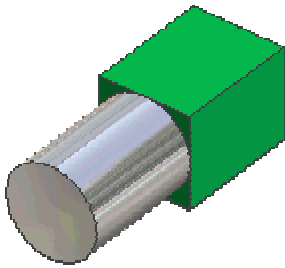
3. Reduce Typing in Excel

Typing in information in Excel can be repetitive. If for example you have a column for Threads1:Designation and another for Length and you want to make the filename a combination of the thread size and length you can use the Excel "concatenate" function. This allows you to combine the information from multiple cells into one cell. See the Excel help for more information.

4. 3D Text in Inventor

As of version 5.3 Invcentor does not have a built in capability to extrude or emborss text in a model. There are two ways to achieve this effect. One is to make your text in AutoCAD and use the Express tool "Explode Text" to obtain the text as lines. You can then import this sketch into the model and extude or cut it. A much more legant way is to use Charles Bliss' 3D Text Macro. You can find it at **his site**. Read the installation instructions and have a blast with it. It's really a great addition to IV.

4. Strange Characters in iPart Children

A recent post in the NG reminded me that I did not address some aspects in my iParts tutorials (since updated). If you notice that some of your iPart children have strange characters in the filename: e.g. HexBolt1(_FS)4-20.ipt or IPNA9(_BS)24.ipt then you have illegal characters in your filename column. Illegal characters include: / \ * ? $ and a few others. (I'm attempting to compile a complete list. If you find others please email me). While these characters ARE legal in the description, title, part number etc... fields they are not legal for filenames.

5. Make detail prints of each iPart Variant

This is a much asked question and the best way we have found so far is to create an assembly and insert one of each of the iPart variants into the assembly. Now make details from the iPart children that were created. Not the most elegant solution but one that works. Some of the VB gurus are working on a way to automate this process. More news as it develops.

6. Is My Video Card Supported / Which Driver Should I Use

Visit **http://support.autodesk.com/inventor-graphic-cards/** for all your answers

7. Adaptively is Grayed Out on the Part I Want to be Adaptive

This is likely due to having used the part as adaptive in another assembly. An adaptive part cannot be used in two different assemblies. If you no longer want it to be adaptive in the first assembly and want it to be adaptive in the second (current one) open the adaptive part and select *Tools/Document Options/Modeling* Tab. Uncheck the "Adaptively used in assembly" checkbox. Save the file ands you should now be able to make it adaptive in the second assembly.

8. How do I prevent parts from showing in the BOM (Parts List)

This is useful when you have to create some existing feature or customer supplied parts to model around but you don't want them to show up in the BOM. In the assembly, RMB on the part and select the *Properties/Occurrence* tab. Check the "Reference" checkbox. No changes will be visible in the assembly. When you place a view od the assembly in the IDW the parts will likely be "phantomed" out. You can RMB on the view and select Edit View. In this section you can control the display of reference parts (Normal, Phantom or Hidden). Regardless of the display the parts will not be displayed in the BOM.

8. Make Model Rotate Automatically

More of a parlor trick than anything else but I've found it useful for presentation purposes. Choose the orbit command (don't hold down F4) then as you orbit the model

hold down the SHIFT key. Now simultaneously release the SHIFT key and the mouse button. The model will continue to rotate at the speed and direction when you let go of the mouse button.

8. Save Model as a Graphics File

You can export IV assemblies, parts and drawings as BMP files. To do this select Save Copy As.. and use the file type pull down to select BMP. Now before saving the file choose Options. You will be presented with a dialogue box with a X and Y value. Enter a value (say 1000) for X and Y (they need to be the same or Y needs to be left as 0). Now save the file. Compare this to a file saved at 500 X 500. This controls the size of the IV file. The larger the number the larger the file size in MB and the better clarity that is generated. Use caution with values above 4000 as the file size can grow to enormous levels. You can now (and should) save this BMP as a JPG or GIF file to reduce the file size.

9. What's New Dialogue Getting Old Real Fast

IV 5.3 has a problem on some installation that causes the "What's New" dialogue box to appear constantly. The remedy for this is to go to *Tools/Add-Ins* and search for DSS Popup Monitor. Uncheck both boxes. The What's New problem should now be OLD news.

10. IV 5.3 and Volo View Install Problems

Before installing IV 5.3 be sure to uninstall Volo View Express from your machine. The installation of 5.3 requires Volo View Express 2 and it will not successfully install over the older version. Also be aware that there are two different versions of VVE on each CD of the AIS. The one you want to install is the one of the AIS #1 CD (the one with IV, not MDT on it)

11. Get a Hold on Those Bolt Circles

This tip comes from Quinn Zander and address a simple way to construct bolt circles that are easily changeable based on horz. and vert. dims.

```
"Using a Pattern in a part for bolt patterns has many advantages. The
spacing can be easily controlled with parameters, and by using the
Associative Pattern tool in an assembly, fasteners can be placed
parametrically into the pattern semi-automatically.

Let's take our rectangular face that needs a 4-bolt pattern.

On the sketch we drop a hole-point someplace in the lower right corner
of the sketch.

Start the Line tool (we are not extruding anything so using a Normal
linetype is acceptable and saves 1 step over choosing Construction) and
sketch a horizontal line between the midpoint of the vertical sides of
```

the rectangular face, and sketch a vertical line midpoint between the
top and bottom edges making a "centered" pair of lines.

Now look up the bolt pattern measurements in the book you are
referencing and start dimensioning the Hole Point.

To dimension the hole point, select one of those lines that was just
drawn, and the point, RMB->Linear Diameter and plug the value. Do the
same for the other dimension. Hit "H" on the keyboard (directly from
the
sketch) and drop the hole.

Go to Parameters and rename the 2 **d** values to **bp_horiz** and **bp_vert**
for bolt pattern horizontal and bolt pattern vertical.

Start the Rectangular Pattern tool, select the hole and defining edges
for direction, but when plugging the value for the spacing, click the
rollout arrow and select "List Parameters".

bp_horiz and **bp_vert** will show up in the box for selection.

No thinking, always centered, easily changed, no dividing up the
measurements from the book (even if they are specified in abnormal
units
like cm).

Simple and fast."

12. Stop Picking on Me..

When in detailing (.idw) mode things can often get crowded which leads to picking items
that you don't want to pick. That's when filters come in handy! In an .idw choose the
select pulldown and choose Filters. This will open a new dialogue that allows you to
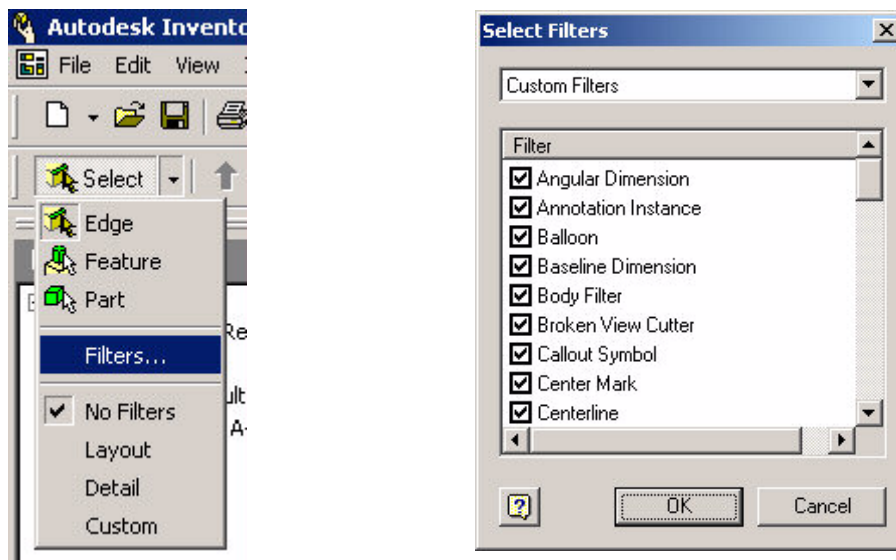
select what items are actually "selectable" in the .idw. Tired of always picking the view
box when you really wanted the detail circle? Uncheck "views" from the list. At the top

of the box are some predefined filter setups and an option to add your own. Just don't forget to recheck the boxes or you'll go nuts trying to get that view to move.

13. I'm getting Dizzy

To turn on Inventor's graphical statistics (aka the "spin test") change the following registry setting.

HKEY_CURRENT_USER\Software\Autodesk\Inventor\RegistryVersion5.1\System\Preferences\Scene Manager\Debug\Post Statistics = 1

This will place a small display in the bottom of the screen that lists statistics about your model.

14. Hole Centers? We Don't Need No Stinking Hole Centers

This tip comes from Kent Keller via the Inventor NG:

Holes can be placed on any point in the sketch (not just on hole centers). You can select the end of a line or the coiencident point of two lines that are joined. The hole feture command will not pick these points automatically but you can select them in the sketch. This can be useful for making bolt circles. Create a polygon with the number of edges equal to the number of holes oyu want then select the corner points of the polygon as your hole centers.

15. One the Surface it's Just a Symbol

This tip comes from Doug Dupont via the Inventor NG:

```
"I remember reading back a week ago that someone did not like the way
Surface Texture Symbols where inserted and having to enter text each
time. I found a better way. Enter your first symbol and them RMB and
copy it. Now RMB again and paste it as many times as you would like.
When pasting it might go to the wrong side of the line or dimension.
All you need to do it drag it over to the proper side. This is a time
saving if you add a lot of surface symbols and don't like to type."
```

16. Dictation for Inventor

This tip comes from Lars Sandberg from the Inventor NG and concerns the use of the transcripting function in IV. This used to be an obvious switch in the Tools>Application Options dialogue but in later versions of IV it was hidden. Additional info on transcripting can be found at http://www.mymcad.com

```
"The key's below ensures that *.tf files are written to c:\temp
Transcription is some kind of real-time backup. If a crash occur you
can see that in the bottom of the *.tf file (look for newest
timestamp). Remove read-only attribute and edit that part
Away.
```

```
[HKEY_CURRENT_USER\Software\Autodesk\Inventor\RegistryVersion5.2\System
\Preferences\Transcript]
"C:\\Temp\\"
"TranscriptingOn"=dword:00000001
"TranscriptSkipErrors"=dword:00000001
"TranscriptStopOnError"=dword:00000000
"ShowCPUTimes"=dword:00000000
"AllowReplay"=dword:00000001
```

17. Squares and Cubes and Superscripts, Oh My!

This tip comes from Quinn Zander via the Inventor newsgroup and concerns how to get a superscript characters (2 for squared, 3 for cubed) in a text dialogue box.

```
In the text dialog, hold the ALT key down and type (with the other hand
on the number-pad, not the top row of number keys) 0179 for cubed and
0178 for squared; let go of the ALT button and finish typing.
```

You can insert other characters this way by going to *Start/Programs/Accessories/System Tools/Character Map* and selecting a character. Look in the bottom right hand corner and note the keystroke combination.

18. Possible Solution to the Imbedded Image Bug

From Bill Bogan et all at Autodesk….

Inserted image displays only an icon and filename
When you insert an image into a sketch, you may see only an icon with the filename below it. This can happen when the application associated with the file type does not a very good job as an OLE Server.

Microsoft Paint displays .bmp, .jpg, .jpeg, and .gif formats. If you have trouble displaying one of these file types, try one of these solutions:

❑ Remove MSPaint from your system and reinstall it.
or
❑ Or change the registry setting for the file type. The following example uses the .bmp file type:

```
Problem:
[HKEY_LOCAL_MACHINE\SOFTWARE\Classes\.bmp]
@="ImageComposer.bmp"


Change it to:
[HKEY_LOCAL_MACHINE\SOFTWARE\Classes\.bmp]
@="Paint.Picture"


 where @ is the 'Default' string name in regedit.
```

This sets Microsoft Paint as the default editor. To open an image with another editor, do the following:

1. Open Explorer
2. Select Tools>Folder Options>File Types Tab.
3. Find bmp in the list of file types.
4. Select Advanced.
5. Add a new action i.e. ("OpenBmp") and set it to your desired application (i. e. "C:\Program Files\Common Files\Microsoft Shared\PhotoEd\PHOTOED.EXE" "%1")
6. Set Action as Default

19. Making Threaded iParts a Bit Easier

David Zinn gave me a tip that cleans up your iPart tables on threaded iParts quite a bit.

If you create a custom property named ThreadX:Family (where X is 1,2,3 etc..) and give it a value of ANSI Unified Screw Threads (or ANSI Metric M Profile or NPT etc..) and a property named ThreadX:Class and set it's value to 2A (or 1A or 3A, whatever is correct for this thread) you do not have to include these values in each row of your iPart table. All that is needed is the ThreadX:Designation. Save a lot of clutter in the iPart table and/or Excel sheet.

Great tip David!

More to come… check back…

Have a nifty trick or tip? Email them to **sean@sdotson.com** and I'll include them here with a credit to you.