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Creating and Using Autodesk Inventor® iParts and iMates

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MA32-4 Inventor, especially Release 10 and the Professional version, includes a huge library of stock components such as gears, bearings, threaded fasteners, and so on. Even so, it is not possible for it to include your "stock" parts as it comes out of the box. Most companies have a series of component parts that are unique to their product line, but are used in various sizes and configurations depending on the particular model or variant being built. A simple example might be a basic shaft that is made in a variety of user-specified lengths. This presentation will show you how easy it is to set up an iPart factory. Now your designers will be able to quickly and easily pick a type of component from your library, then specify details such as size, capacity, and so on. The iPart factory will then generate the specific part. iParts can include iMates, so the iPart components "automatically" attach themselves properly as they are inserted into an assembly.

About the Speaker:

An AutoCAD software user since 1986, Bill was a product engineer and manager for Weiser Lock in Vancouver, Canada for 27 years. Bill has taught AutoCAD and mechanical design at the British Columbia Institute of Technology since 1996 and teaches Autodesk Inventor at the Institute’s Training Center. He has lectured on a wide range of subjects at Autodesk University since 1995. An active member of the Vancouver AutoCAD Users Society, he has written "The Learning Curve" column for CADalyst magazine since 1986, and writes about Autodesk Mechanical Desktop and Autodesk Inventor for Autodesk's Point A Toplines. He also writes for Inside AutoCAD Journal and Design Product News.

NOTE

This course is intended for intermediate users. I assume you have a working knowledge of the basic part & assembly modeling & constraining in Autodesk Inventor™.

Rumour has it that John Walker, one of the founders of Autodesk, once said "Never do anything twice". I'm pretty sure he never said it again.

Many products, especially machines, use standard component parts such as nuts, bolts, washers, bearings, structural steel shapes, and so on. We do not need to create models for these components over and over again because Inventor includes the Content Library, which is a collection of hundreds of thousands of these sorts of parts.

Complete though this collection is, until the Design Accelerator was added in release 10 there was a noticeable absence of certain common components such as gears, v-belt sheaves, roller chain sprockets, and so on. As a partial work-around, there are several sample iPart Factory components for release 9 and earlier, hidden within the c:\Program Files\Autodesk\..\Samples folder.

Many companies build a family of similar products, or use very similar but not identical components in a variety of products or even within one machine. For example, I once did some consulting work for a company that produces industrial air and hydraulic cylinders. Consider their air cylinders alone; they are available in bores ranging incrementally from 1 ½" to 12", with four different mounting foot styles, three valve port arrangements, three different rod end fittings, and in any customer-specified stroke length.

All the cylinders look pretty much alike except for size. Although there are only 12 different part numbers in an assembly, they had over 2500 component part drawings and it wasn't enough. Someone was always ordering a combination that had never been built before.

This is a perfect application for the iPart functionality that I alluded to earlier.

An iPart factory is a single part file that includes a table outlining the possible variations in the part. When you select an iPart factory for placement in an assembly, you see a series of key options that enable you to fully define which variation of the part you want to place. An iPart child matching your selections is generated (or reused if previously generated) and placed as a part occurrence in the assembly.

Let's work through a step-by-step example of generating a simple iPart factory for a dowel pin and then placing variations (iPart child) in an assembly.

1. Create a model containing all features that may be included in any of the iPart children. Our sample part is a dowel pin with a small chamfer on each end.

Figure 1: A simple dowel pin

2. Rename the part parameters (sketch dimensions, extrusion lengths, and so on) that you will manipulate to create the different iPart children. In our example I have renamed d0=Diameter and d1=Length.

Model Parameters

Parameter Name	Unit	Equation
Diameter	in	0.25 in
Length	in	0.75 in
d2	deg	0 deg
d3	in	0.010 in
d5	deg	20 deg

Figure 2: Renamed parameters

3. Start the iPart Author tool (Tools > Create iPart). Renamed model parameters and any user parameters are automatically added as parameters that can vary between iPart children. The current values of these parameters define the single version initially available.

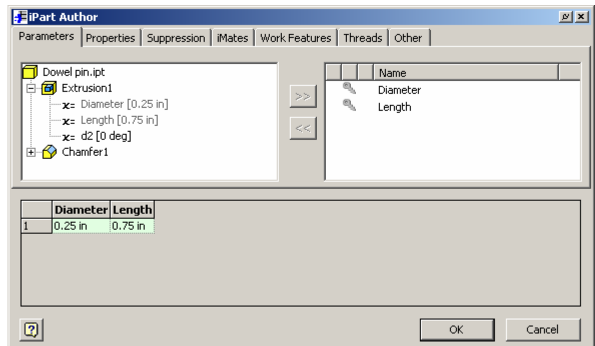

Figure 3: the iPart Author

4. Select critical parameters as keys for the iPart, as shown in the top portion of Figure 4. Right-click on a parameter name, select Key from the context menu that appears, and then select its priority.

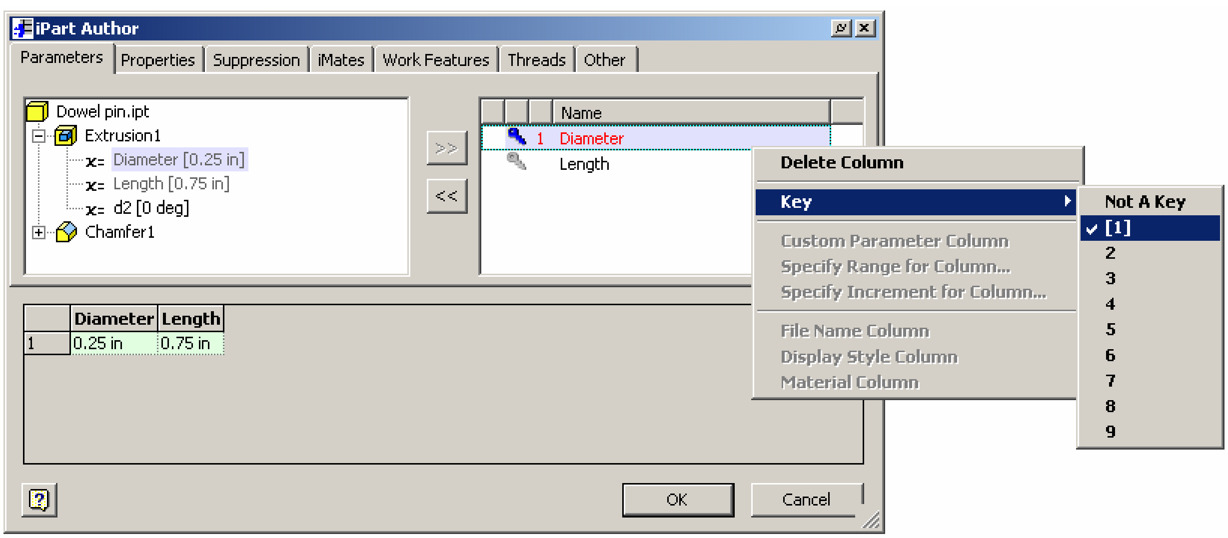

Figure 4: Setting key values

Key values are presented during iPart placement to provide a hierarchical selection of the iPart child. In this example, the diameter and length of the dowel are the only selections required to fully define the dowel. Diameter is set as the primary key, and length as the second key. When placing the dowel in an assembly, you first select diameter, thus limiting the length selection to those iPart children matching the selected diameter.

5. Add rows to define each iPart child. You can right-click the existing table entry in the lower portion of the iPart Author dialogue box and then select Insert Row, but this can be a little tedious. The easy way is to click OK in the iParts Author dialog, then right-click the new Table entry that has appeared in the Browser. Click on Edit Via Spreadsheet to edit the table in Microsoft® Excel to speed up row additions and create relationships between table cells.

   NOTE Microsoft® Excel must be installed on your computer for this to work.

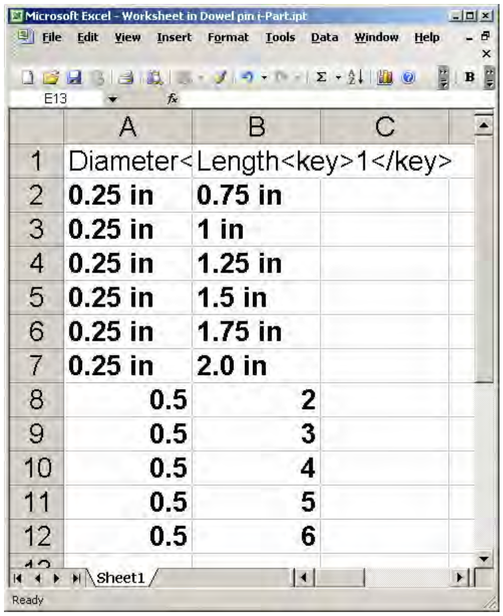

Figure 5: Adding part variants in a spreadsheet

Inventor stops reading the table at the first blank cell, so there cannot be empty cells within the desired range. In particular, note the repeated values in the first column. Values can be in text or numeric format, but each change of format or value is seen as a new group. For best results, be consistent.

When you are finished, close the spreadsheet simply by clicking on the X in the upper right corner. Your revisions will automatically be written back to the Inventor part file, and the Table entry in the Browser will update.

6. Save the iPart factory to a “suitable” folder. There will be more on this later, but for now we will use the current workspace folder. We’ll call or sample *Dowel Pin.ipt*

That’s it! We have now created an iPart factory containing 11 different dowel pins using just one model and a table. Now let’s look at using our iPart factory.

7. Create a new assembly file, or open an existing one.

8. Click on the Place Component tool, and then select your iPart factory from the Open File dialogue, just as you would place any regular existing part.

9. Now comes the difference from placing a normal component. An iPart factory brings up a dialog box showing the part variants that are available. Figure 6 shows all three tabs.

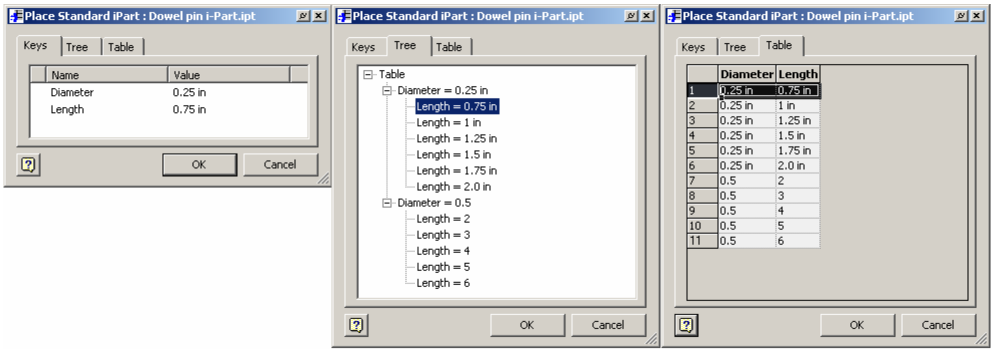

Figure 6: The three tabs of the Place Standard iPart dialogue box.

Let's look at each tab in turn.

a. Keys – this displays the key parameter names, and the default values. If you simply click OK at this point then Inventor revert to behaving almost like a normal component insertion sequence; you will be invited to select insertion points for one or more instances of the part.

   The slight difference is that the dialog box remains visible, so you can go back and select a different variant. To do this, you need to click on one of the other tabs.

b. Tree – This shows all possible variants that are available. Initially, the tree is collapsed so that only the first key values are visible, but you can click on the + sign beside them to expand them out. You can also stretch the dialog box to show more variants. Click on the one you want, click on OK, and place one or more instances of it.

   Once again, the dialog box remains visible so you can go back and select a different variant.

c. Table – this displays all possible variants in a spreadsheet format. Click on the one you want, click on OK, and place one or more instances of it.

   Once again, the dialog box remains visible so you can go back and select a different variant.

Each time, the software generates the selected iPart child, or reuses it if it already exists, and places an occurrence in the assembly.

Now, wasn't that a lot easier than modeling each variant as you needed it?

There are two kinds of change to an iPart…

Type 1: What if you decide to change the variant of an existing iPart child insertion within an assembly?

No problem. Each child placed in the assembly contains the iPart table. Click on the + sign beside any iPart child instance in the browser, then right-click on the Table entry that appears. Click on Change Component, and the Place Standard iPart dialog box of figure 6 reappears. Make your selection, click OK, and you are done. The one instance updates, and all existing assembly constraints remain connected.

Type 2: What if we change the basic definition of the iPart factory?

No problem. Simply open and edit it like any other Inventor part. For example, we can replace the chamfer at each end with a fillet. Note that we have to open the iPart factory directly. We cannot edit it from within the current assembly, as we can with normal parts.

You also want to be careful what you wish for; you might get it. If you change the basic definition of an iPart factory then <u>all</u> instances of <u>all</u> variants of <u>all</u> assemblies that contain <u>any</u> child of the iPart will update.

Plain and Fancy…

So far we have created a part wherein only the key values were varied. What if we want the chamfer size to change automatically for the group of larger-diameter parts?

No problem.

1. Open the iPart factory file.
2. Rename the chamfer size parameters. In our example (figure 2), d3=Chamfer_Size and d5=Chamfer_Angle.

3. Start the iPart Author tool (Tools > Create iPart).
4. Expand the Chamfer feature to display its parameter names.
5. Click on each of them in turn, and click on the >> button to copy them to the Name window. As you do so, a new column will be added to the table window for each parameter. This time, do not assign them key numbers.
6. Click OK.
7. Edit the spreadsheet table and change the chamfer size and angle as desired. Note that all of the pins of one diameter do not have to have the same chamfer sizes; you could enter varying chamfer sizes depending on length, or even totally randomly. Close the spreadsheet and save the part file.

Now when you insert an iPart child into an assembly you will still only be able to select between the Diameter and Length key values, but the chamfer sizes will automatically change to suit the particular iPart child you selected. Once again, any existing instances in any assembly will also update.

Obviously, you could have added the chamfer specifications when you originally created the iPart factory, rather than adding them later.

iSpy…

Okay, what if some variants don't have a chamfer?

No problem.

1. Start the iPart Author tool (Tools > Create iPart). Observe that the iPart Author dialog has a number of tabs. Click on the Suppression tab.
2. Note the list of features in the upper left window. Click on the Chamfer feature.
3. Click on the >> button. A new Chamfer1 column appears in the table, with the word Compute in every cell, as per figure 7.

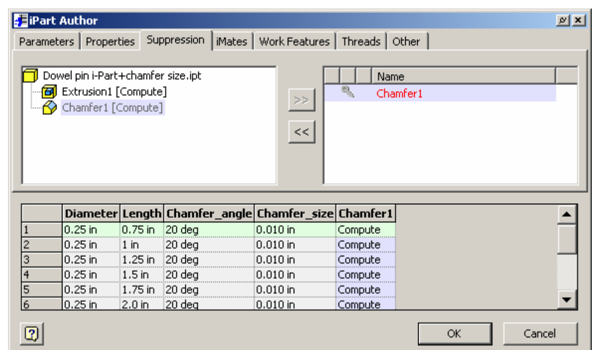

Figure 7: iPart Author Suppression tab.

4. If you do not want the Chamfer feature to appear in a particular iPart child variant then simply click on its cell and change the word Compute to Suppress. Repeat as necessary for other variants. If you prefer, you can also do this in the spreadsheet editor.

That's it! Whenever you insert an instance of a variant, the iPart factory will compute or suppress the feature as appropriate. You can have more than one suppression feature, with different combinations of Compute/Suppress. For example, some pins might have a chamfer while others have a fillet and still others have neither.

Notes To You…

1. All variations, whether it is by values or Compute/Suppress, must be "possible". You cannot have a hole larger than the part in which it is placed.
2. You can only suppress a full feature; you cannot suppress a face or edge that is created from a portion of a sketch.
3. If you suppress a feature, you might also suppress any other features that are dependent on its existence.

4. All other parametric relationships within a part are maintained. If another feature has a dimension related through a formula to a dimensioned defined from the iPart table, then it will change as the iPart value changes.
5. Observe the second tab on the iPart Author dialog box. iParts have access to <u>all</u> the File>iProperties available in a file. You can thus have iPart factories with options for raw material, source, costing, and so on.
6. The fifth and sixth tabs let us specify options for work features and thread specifications respectively, if any are present in our part. A single iPart factory could thus produce all possible Imperial and metric bolts, for example, as long as the table is big enough to include all standard cross-combination values.

There Are Two Kinds of iParts…

So far, all our examples have used specific, discrete values for the different variants. These are known as "Standard" iParts.

Okay, what if you have an aluminum extrusion that you want to cut off to varying user-specified lengths? You don't want to re-create the profile each time and then extrude it by different amounts.

No problem. Simply create a "Custom" iPart factory. Actually, there is one minor problem; the Inventor help facility is a little thin in this area.

Let's use a really simple example.

1. Create a new part consisting of a single feature, which in turn is defined by a circle. When we extrude it we will get a simple rod.
2. Rename the appropriate parameters: d0=Diameter and d1=Length.
3. Convert the part to an iPart factory.
4. Set the Diameter to be key #1. <u>Do not</u> assign a key sequence to Length.
5. In the table window, right–click on the Length column header and then click on Custom Parameter Column. All cells in this column (1 in our case) will turn blue to indicate it is a custom value.
6. Click on OK, then save the drawing.

We have now created a Custom iPart. The basic difference is that a Standard iPart has discrete values for each cell and therefore will only create a finite number of different parts. At insertion time, the user selects one of the possible combinations.

With a Custom part, on the other hand, the user gets to specify a unique value for one or more parameters at insertion time or during subsequent editing. Let's see this in action.

7. Insert the new iPart into an assembly.
8. This time the Place Custom iPart dialog box appears. Figure 8 shows its three tabs.

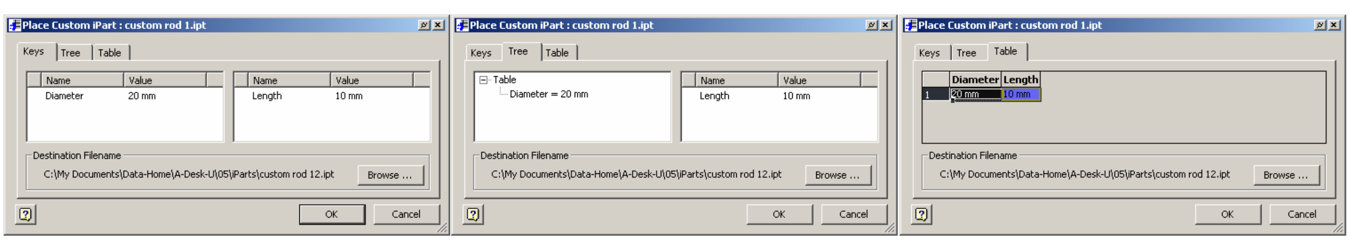

Figure 8: the 3 tabs of thePlace Custom iPart dialogue box

Note that the first two tabs now each have a new pane on the right, displaying the default length. The Table tab has a blue background in the Length cell. It is not quite obvious, but they are actually asking you to specify a custom length.

9. Click on the default Length value in either of the first two tabs, or double-click on the blue Length cell in the Table tab.
10. Enter any desired value.
11. Before going any further, note the Destination Filename section at the bottom. More on this later.
12. You can place as many instances of this length as you want.
13. To insert instances of a different length you must click on the Browse button in the Destination Filename section and then specify a new file name. Now you can insert instances of the new length.

Our sample showed a simple table with a single row. It is possible to mix and match, so there are several possibilities for custom iParts:

1. We can have one or more custom columns that apply to all rows. For example, our simple cylinder could be available in several pre-defined diameters but the length of any selected diameter is user-defined.
2. We can have individual custom cells. For example, diameter sizes 1, 2, and 3 are only available in a short list of standard lengths, but size 4 is available in any length.
3. We can apply upper and lower limits and/or stock increments to any custom cell or column. For example, size 5 is available in lengths of 10” to 200” in 5” increments. This could be done with many lines in a standard iPart, but is often easier in a custom one.
4. Custom cells or columns can be applied to any parameter, including calculate/suppress and any of the file iProperties.

The same, only different…

There is one other major difference between standard and custom iParts, and that is file locations.

Remember, every unique part in Inventor requires a separate file. This includes parts generated by iPart factories.

1. Standard iPart factories generate the file name automatically. It is a concatenation of the base part name plus the option values. The first time an iPart is generated from an iPart Factory, it creates a new folder just below the one containing the factory and saves the variant file there. All subsequent parts generated from it land in the same folder.
    a. If you or another user select the same combination again, Inventor simply re-uses the existing generated file to save time.
    b. If you revise the selection of an existing instance in an assembly to a combination that has not been used before, it generates a new part and links to it instead.
2. Custom iPart factories generate a default file name, being the base part name plus an incremental number. Each variant is stored in the same folder as the iPart factory.
    a. We can override the default name and folder if we want. Remember the browse button mentioned earlier?
    b. If you re-use an existing combination of values, Inventor generates another new part even though an identical one already exists. You are often better to copy an existing instance rather than generating a new one.

So which is better, standard or custom?

That depends:

Standard iPart factories are usually used when the parts it generates are indeed “standard”, such as threaded fasteners that may be used by multiple users within an organization, or company-specific parts that are nonetheless only available in discrete sizes. The factory and its generated children would normally live on a network drive that is included in the Project file library path.

Custom iPart factories are usually used when the specific children it generates are unique to a specific assembly. A typical example would be a “stock” extrusion shape that is cut to multiple “suitable” lengths. Assembly-specific factories often live in the same folder as the assembly that uses them. If several assemblies use the same basic factory, then it often lives on a network drive but the children are named so they live in the assembly’s folder.

Mating season…

Standard components are typically assembled using the same geometry and constraint type. For example, a bolt is typically assembled to a hole with a single Insert assembly constraint using the circular edge under the bolt head. iMates are predefined, named assembly constraint halves that are attached to key geometry. When a part with iMates is inserted into an assembly, it will look for other parts with the matching half constraint. If it finds a mate, it combines to form a single resolved constraint. Let’s look at a simple example.

1. Create a part, such as the simple clevis pin shown in figure 9.
2. Click Tools > Create iMate
3. A dialog box appears that is almost identical to the standard assembly constraint dialog, except it is called Create iMate, and it only has provision to select a single feature. For our example we will apply an Insert imate to the underside of the clevis pin head. Click Apply.

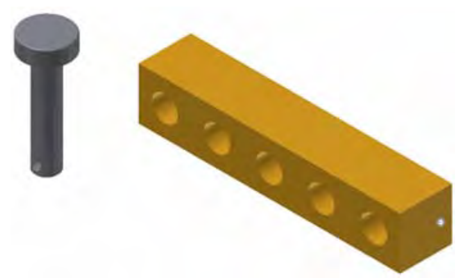

Figure 9: A clevis pin and a bar full of holes

4. The browser now has an iMates entry. Expand it out, and rename the iMate to Pin-A.
5. Save the part.
6. Create another part, such as the bar full of holes also shown in figure 10.
7. Create Insert iMates on all the holes.
8. Rename the iMate on some of the holes to Pin-A and some to Pin-B.
9. Save the part.
10. Create a new assembly.
11. Click the Place Component tool to place an instance of the bar in the assembly.
12. Click the Place Component tool again.
    a. If necessary, click on the Use iMate button in the lower left corner of the file dialogue box to make sure it shows a check mark.
    b. Select the clevis pin part. The clevis pin is added to your assembly. Like magic, it automatically finds a hole with an iMate whose properties exactly match those of the pin. It then breeds a single Insert constraint from the two half iMates, and zooms to show you this insertion.
13. Repeat the previous step until you run out of matching iMates. Remember, ours is named Pin-A. Each time it will find the next available match until it runs out.
14. When it runs out of Pin-A iMates it will simply revert to a standard component placement function and ignores the Pin-B iMates. The instance will be place, but it must be constrained manually.

That's it! iMates are as simple as that. Okay, now for some specific details.

1. I said earlier that the iMate halves must be an exact match. This includes type, name, and offset value.
2. iMates can be applied to any type of constraint, including the motion constraints.
3. A pair of iMates resolve into a single constraint which cannot be edited. It can only be deleted and replaced. The unresolved iMates become available for later use.
4. A part can have more than one iMate. Upon component placement it will resolve all that it can, and leave the others unresolved.
5. Each unresolved iMate has a special circular glyph on the part, but by default they don't show. Right-click on a part and select iMate Glyph Visibility to see them.
6. iMates can be applied to part features within an assembly. They can then be used when the assembly is placed as a sub-assembly within a larger assembly. Unresolved iMates on components within the sub-assembly do not carry forward, however.
7. Alternate components can have the same iMate specifications. For example, a machinery manufacturer might use a variety of motors and gearboxes. If all motors and gearboxes have NEMA C flange mountings, and if iMates are set up properly, then any motor will mate with any gearbox.
8. You may want to apply iMates semi-automatically. For example, you may have forgotten to turn on the Use iMate button, or you may have done so deliberately because you didn't loke its automatic choice. No problem:
    a. Right-click on one or more parts and then activate iMate Glyph Visibility.
    b. Press and hold the Alt key, then click and hold on an iMate glyph.
    c. Start dragging the glyph. When it turns green you can release the Alt key.
    d. Drag the part to the desired location. When the first glyph touches the second they will mate if they are compatible.
9. Updating the iMate definition in a part does not retroactively update any prior insertions. Once an iMate has paired it becomes a fairly normal constraint and does not refer back to the part definition files.

Professor Inventor in the library with a mouse…

(Much of the following was "borrowed" from an article by Neil Munro, but that's okay; we work together and regularly steal each other's stuff).

Autodesk Inventor software includes a library of standard fasteners, bearings, keys, and other mechanical parts. These components are available in a wide range of international standards, including ANSI, ISO, DIN, BSI, and JIS. The library browser is a separate pane in the assembly browser, as shown in Figure 10.

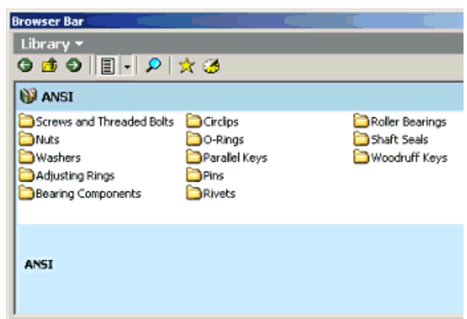

Figure 10: The Content Library

As with iPart factories, the software generates a selected component the first time you place it in an assembly. Standard library parts include appropriate iMates (see the iMates section of this guide for more information) attached to the geometry typically referenced when assembling the part. In addition, you can easily replace one or all occurrences of a library component with a different version.

Components are placed from the library catalogue using Autodesk's i-drop® technology. Once the size parameters of the part have been specified in the library browser, you click in the component image window to fill the i-drop symbol and then drag to the Autodesk Inventor graphics window to place an occurrence in the assembly (see Figure 11).

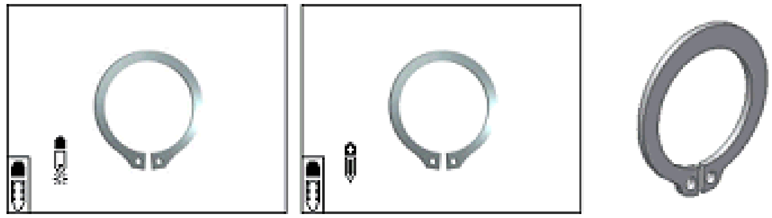

Figure 11: Using i-drop technology to insert a part

With i-drop technology you can drag content from other servers, including web-based servers, directly into your assemblies. Tools to add to existing libraries or create your own i-drop-based content libraries are not yet available. You must create iPart factories or use other methods for components that are not included in the supplied content library.

The content library also includes custom library parts, currently limited to common steel shapes such as channels, flanged beams, and angles. As with custom iParts, you specify the length of the custom library part during placement and provide a file name and location to store the custom part. Unlike standard library parts, custom library parts are editable and you can add features after placement (see Figure 12).

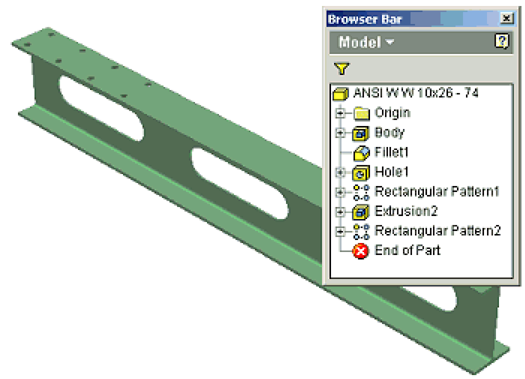

Figure 12: Custom iPart with additional features

Decisions, decisions...

So, how do you decide which method to use for standard part creation? None of the methods described is perfect, and a solution that uses all the creation tools is probably the answer for most companies. iParts are currently the most flexible solution, but generating iPart factories may require considerable work. The content library requires little setup, but the inability to add new content or edit the standard parts generated from the library may limit its usefulness for some companies. Reusing existing data and creating new non-iPart models will likely be a part of your standard component strategy, so take the time to create simple, efficient models.

Adding to your choices, Release 10 added the Design Accelerator. This is a full-scale topic in its own right, but basically it is an automatic component generator that is not based on sizes but on performance; for example, you simply specify the power, speed, ratio, and service factor. It does the design calculations and generates the parts for a v-belt drikve that meets your specifications.

As you can easily guess, iMates and iParts were a match made in heaven (or at least in Tualatin, Oregon). Regardless of the methods you use to create standard parts, consider adding iMates to all your standard components. All standard parts in the content library include appropriate iMates, iParts can include iMates, and standard parts and subassemblies can also contain iMate definitions.

Never forget the Prime Directive: never do anything twice.