Custom Parameter Formatting and Expressions in Autodesk Inventor

When preparing Production Documentation, it has always been an interesting task to set up your values in the Parts List the way you would want your downstream departments to reference your parts and assemblies.

1

PARTS LIST				
ITEM	QTY	PART NUMBER	STOCK NUMBER	S & P Number
1	1	1234-c	50.000 mm x 50.000 mm x 0.500 m	102650.000 ul
2	1	1234-b	50.000 mm x 50.000 mm x 0.750 m	102650.000 ul
3	1	1234-a	50.000 mm x 50.000 mm x 1.000 m	102650.000 ul
4	1	1234-a	50 mm x 50.000 mm x 1.000 m	102650.000 ul
5	1	1234-d	50.000 mm x 50.000 mm x 1.250 m	102650.000 ul

D

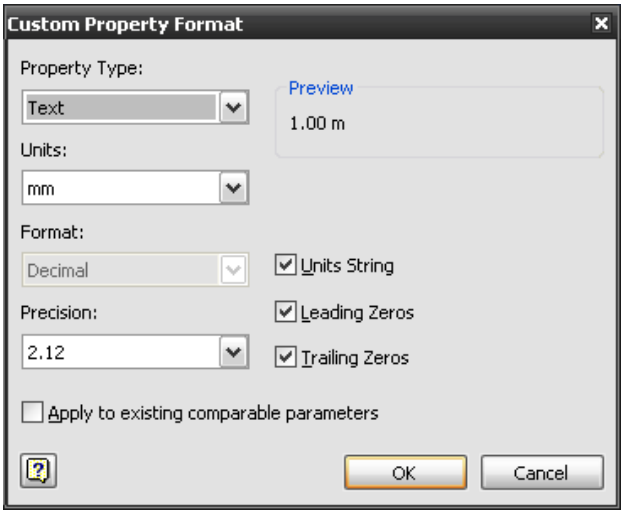
Now with Inventor 2009 it becomes easier than ever to setup your Documentation with Custom Property Formatting in the Parameters Table and Custom iProperty Expressions. With this new functionality you can set custom formats to better control units and precision. This will lend greater accuracy and conformity in your initial designs as well as your revisions to those designs.

1

PARTS LIST				
ITEM	QTY	PART NUMBER	STOCK NUMBER	S & P Number
1	1	1234-c	50 mm x 50 mm x 0.50 m	102650
2	1	1234-b	50 mm x 50 mm x 0.75 m	102650
3	1	1234-a	50 mm x 50 mm x 1.00 m	102650
4	1	1234-a	50 mm x 50 mm x 1.00 m	102650
5	1	1234-d	50 mm x 50 mm x 1.25 m	102650

D

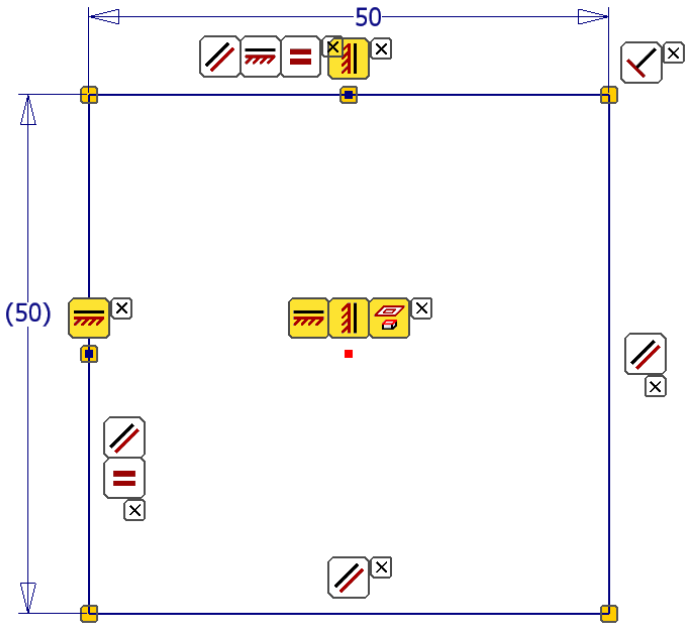
The purpose of this whitepaper is to introduce you to the options as well as their downstream significance in your designs.



Parameters 101

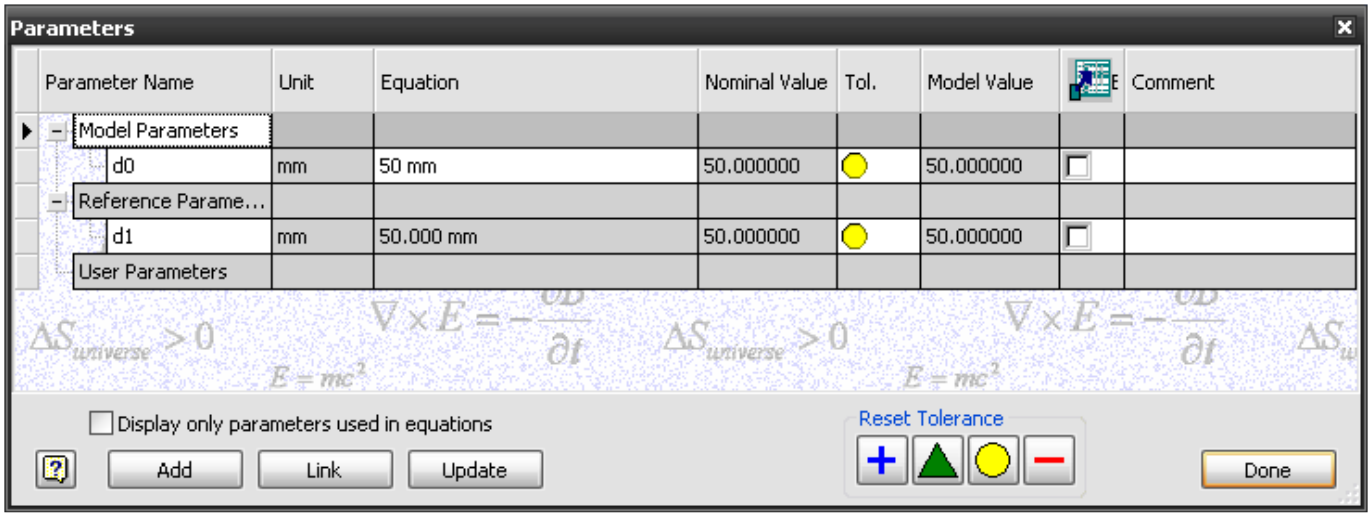
I have been using Parameters for a number of years, but here is a quick recap of different parameters and how to use them. If you already know what a Parameter is then skip to the next section.

Below you can see a very simple square sketch made up from a standard metric template. The origin is projected and constrained horizontally and vertically to the midpoints of the square's sides. The sides are also constrained equally to each other thus requiring only one dimension to become fully constrained. I placed a driven dimension on the left side for reference later.



Tip: Notice my constraint icons and annotations are larger, this is due to a scale change in my application options under the General Tab and Sketch Tab.

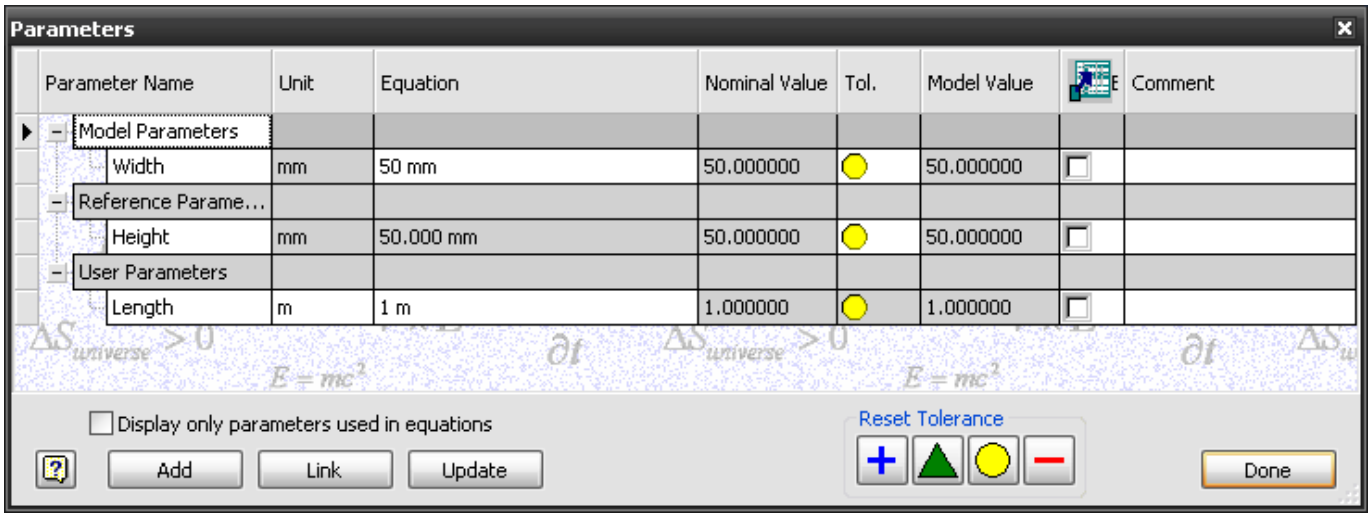
Looking at the f_x Parameters... you can see d0 is the 50 mm dimension across the top of the sketch as it is listed as a Model Parameter. The Reference Parameter is the 50 mm along the side of the sketch created as the driven dimension.



Inside the Parameters box you can create your own Parameters as well known as User Parameters by using the Add button. Existing Model and Reference Parameters can also be renamed for clarity.

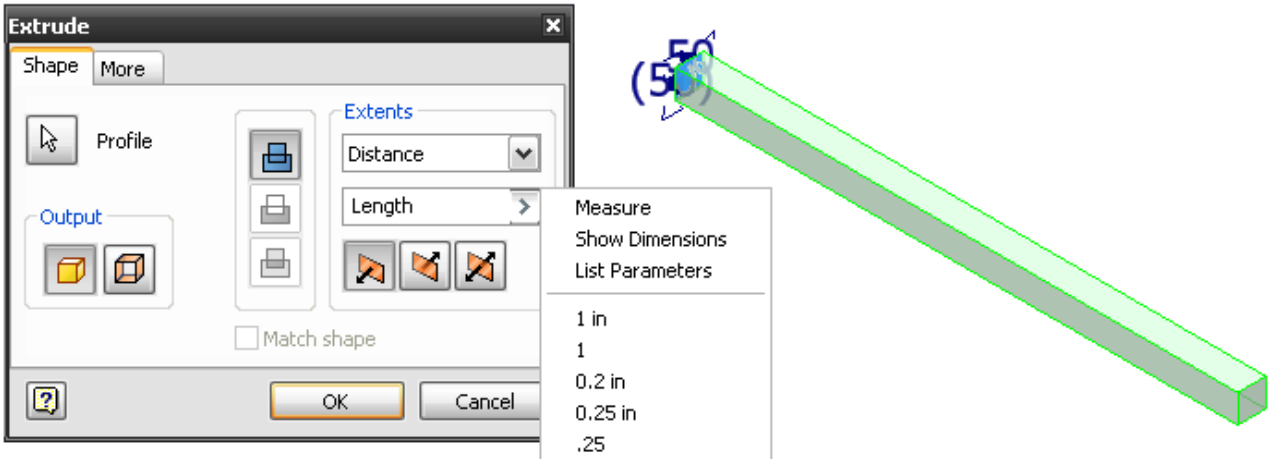
Remember, when renaming Parameters:

- Do not start with a number
- Do not use duplicate names
- Do not use spaces.



Change your Dimension Display (right click in the graphics window when not in a command) and change your display to Expression or Name and click Update to see the changes to your Parameter names or Equations.

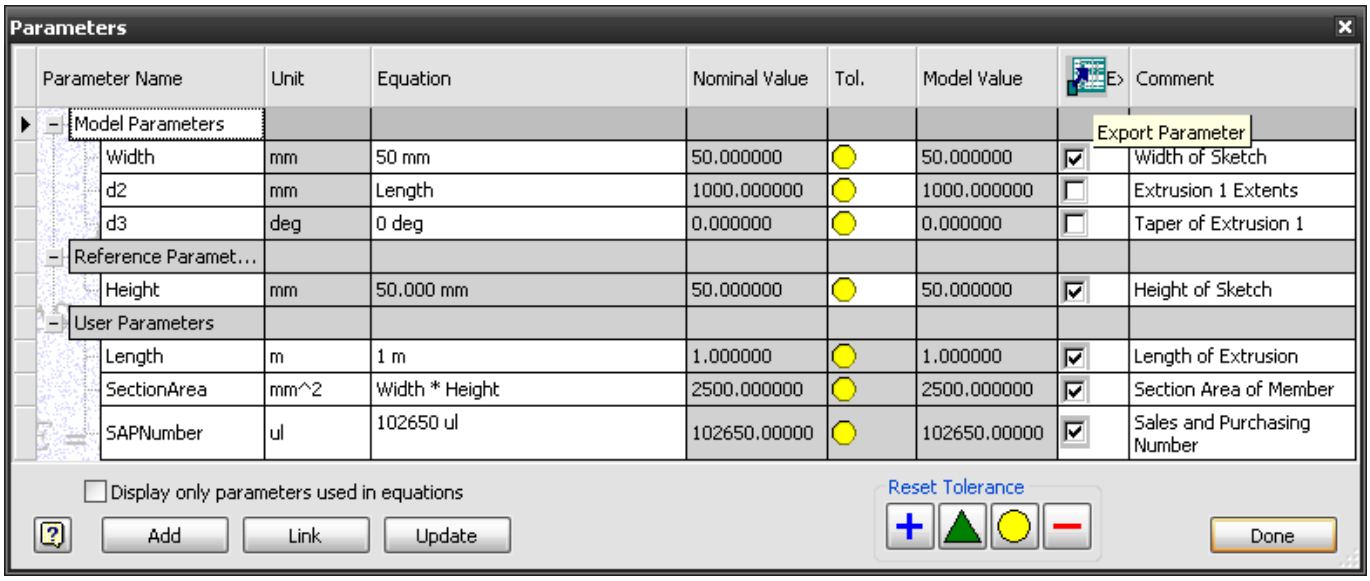
We can also use these Parameters in our Part Modeling tools as well. Whenever you see a right facing pull down arrow, click on it for more options and one of those options is to List Parameters. The Parameters that show up for selection are any that have unique names (not d0, d1, d2, etc).



You can also link or embed parameters created from an Excel spreadsheet (.xls) or other Inventor files (ipt, iam) right into the current file. With Excel, while linking is an attractive approach, I sooner embed and keep the data with the file rather than separated. Embedding seems to work smoother with file management systems like Vault.

Custom Parameter Formatting

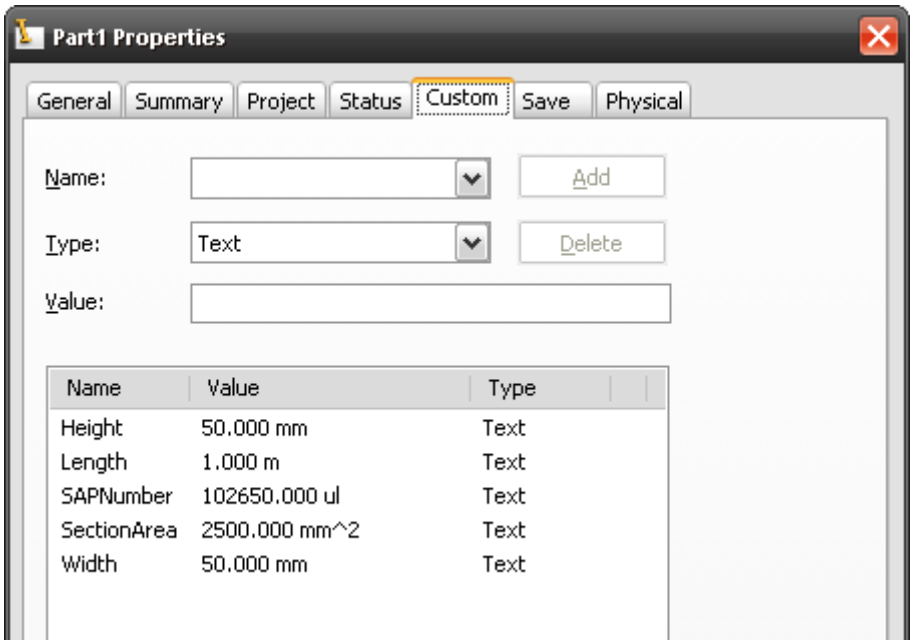
The first step in creating the Custom Formatted Parameters is to export them. The column labeled Export Parameter has a check box for each Parameter to make it exportable.



Once a Parameter is exported it can be used in:

- iProperty Expressions
- Part Lists
- Bill of Materials

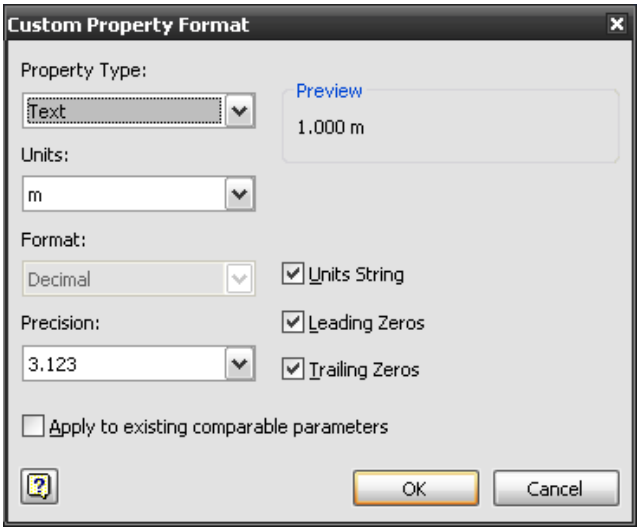
This process places Parameters into the Custom tab of the iProperties of the file so that it can be used in these various ways.

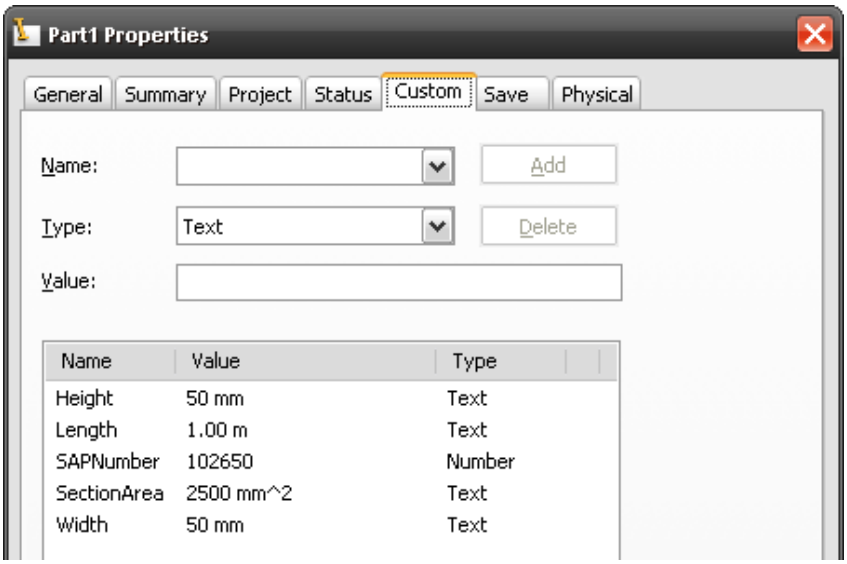


Once the Parameters are marked for exportation, a new right click option becomes available. First select (left click) the Parameter to work on, then right click and choose Custom Property Format.

Custom Property Format Dialog Box

- Property Type – Text or Number
- Units – Choose a Unit type based on parameter default
- Format
  - Inch and Feet can be decimal / fractional
  - Degree units cab be decimal / deg-min-sec
  - Any other type box is grayed out
- Precision – different choices depending on Format type
- Units String – display units after value
- Leading Zeros – display leading zeros
- Trailing Zeros – display trailing zeros
- Apply to existing comparable parameters
  - All similar parameters (Lengths, Volumes, etc) are all formatted based on one change

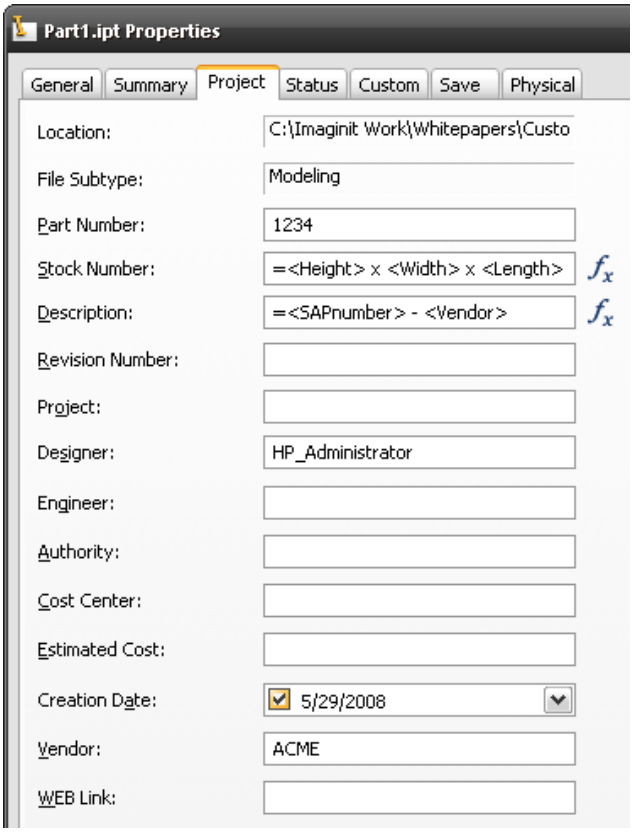




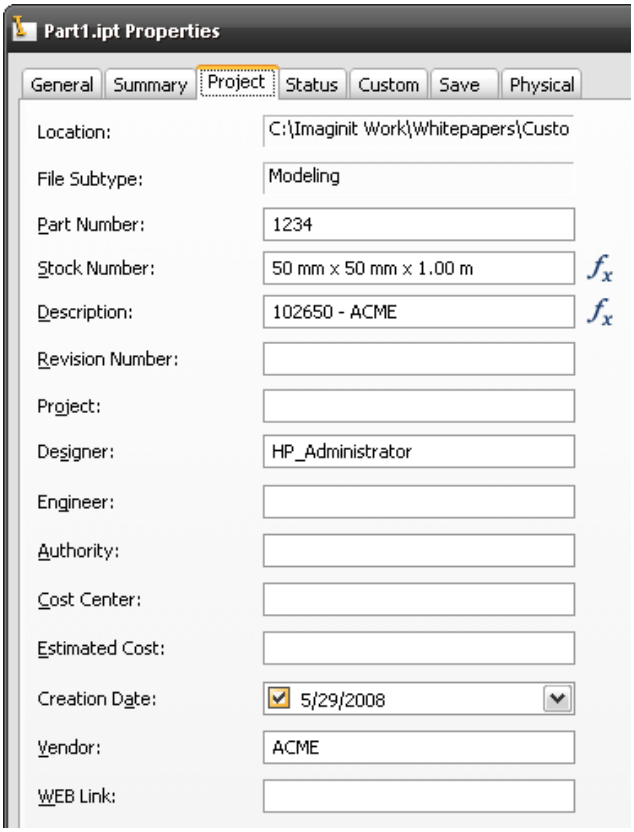
After a few changes the Custom tab of iProperties looks a little different, but there is no visual change to the Parameters box. It appears that this could all be done through the iProperties box naturally, but in this manner the user can create equations with their parameters and easily linked the dimensions from their sketches to the appropriate downstream items.

Using iProperty Expressions

Expressions used in iProperties are very powerful for unifying your designs' documentation. Expressions that are created in the iProperties dialog box can contain any combination of the iProperties in the file. They are primarily created for text type placement and are not used as equations. When an iProperty is evaluated the value of the parameter is substituted for the name. This will work for any file with iProperties.



IProperty Expressions



iProperties Evaluated

To create the Expressions, simply click in the iProperty box you wish to create an Expression in and follow these guidelines:

- Start the Expression with an "=" sign
- Surround your iProperty to reference with the less than (<) and greater (>) than symbols
- Use separators between your reference iProperties (spaces, x, - , _, &, etc)
- Hit Enter to evaluate the Expression
- Hover over the Expression to check the formula used
- Right click and choose Edit Expression to modify your formula

Using Parameters in the Parts List and Bill of Material

The process of knowing which parameters to export for use comes with first hand knowledge of your design. A user may know which parameters they want exported before the first sketch is drawn. In that scenario an intelligent template or start part with predefined parameters, export options, and parameter formatting is normally used. If a user is designing from scratch the known parameters may be ever evolving and added throughout the design.

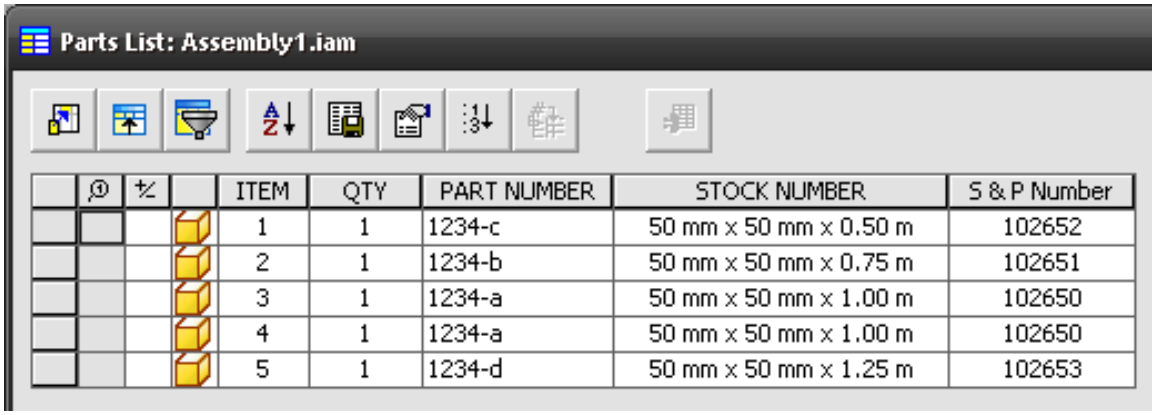
In the Parts List add a column that has an expression in it to streamline your documentation. Create templates for your files that utilize these parameters and expressions. Develop Part List styles that incorporate your design intent for them.

To add a column created from Exported Parameters and Custom iProperties:

- Edit the Part List by double clicking on it
- Select the Column Chooser 
- Select the New Property button 

- Click the text to add an iProperty
- Add the exact name of the iProperty
- Reorder the column as necessary in your Part List 

- Right click on the Heading of the Column and choose Format Column
- Rename the Heading as necessary

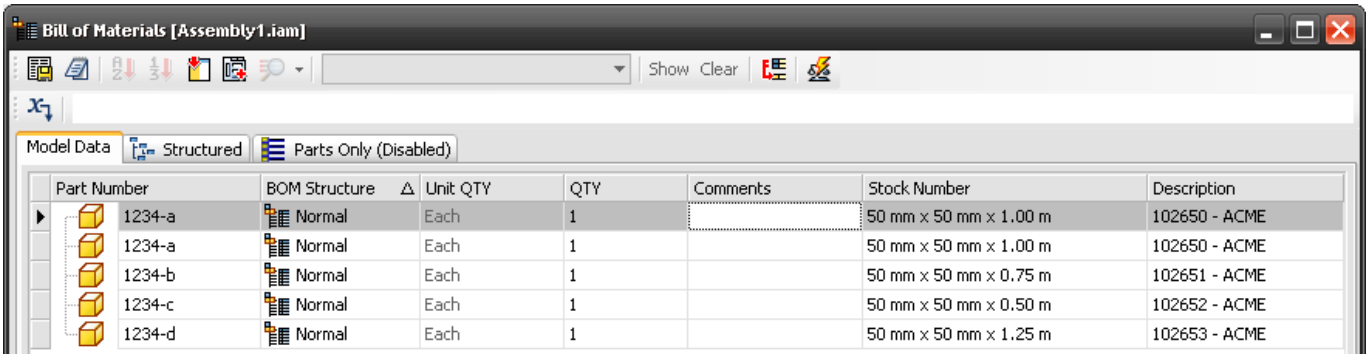


ITEM	QTY	PART NUMBER	STOCK NUMBER	S & P Number
1	1	1234-c	50 mm x 50 mm x 0.50 m	102652
2	1	1234-b	50 mm x 50 mm x 0.75 m	102651
3	1	1234-a	50 mm x 50 mm x 1.00 m	102650
4	1	1234-a	50 mm x 50 mm x 1.00 m	102650
5	1	1234-d	50 mm x 50 mm x 1.25 m	102653

Follow these tips to maximize your documentation of your designs up front

- Create templates (IAM, IPT, or IDW) with:
  - Exportable Parameters
  - iProperty Expressions
- Create Part Lists Styles in IDWs
  - Use Group settings
  - Format your columns

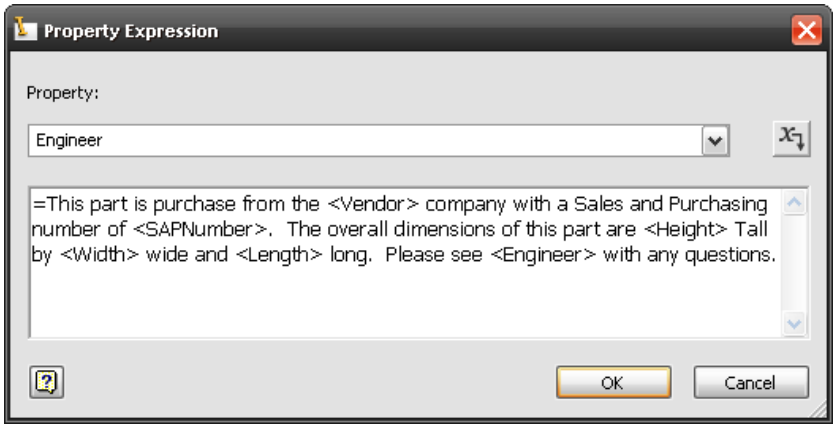
In the Bill of Material you can control expressions when the design gets to the documentation phase. Right click on the Parts List and choose Bill of Materials. Click on a column and choose the Create Expression button to formulate an Expression.

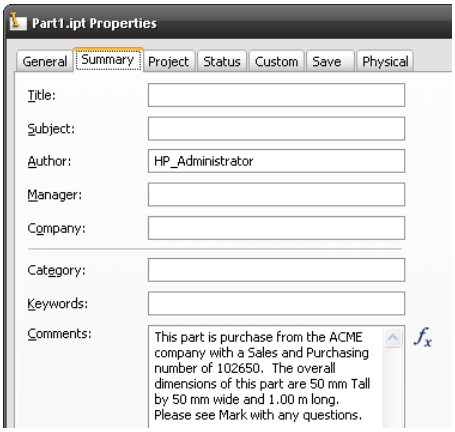


Part Number	BOM Structure	Unit QTY	QTY	Comments	Stock Number	Description
1234-a	Normal	Each	1		50 mm x 50 mm x 1.00 m	102650 - ACME
1234-a	Normal	Each	1		50 mm x 50 mm x 1.00 m	102650 - ACME
1234-b	Normal	Each	1		50 mm x 50 mm x 0.75 m	102651 - ACME
1234-c	Normal	Each	1		50 mm x 50 mm x 0.50 m	102652 - ACME
1234-d	Normal	Each	1		50 mm x 50 mm x 1.25 m	102653 - ACME

*Bill of Material editing will change Number type values to Text type values

In this box use text started with an "=" sign and type in text and use the Property pull down list and the insert Property button to aid in the process for accuracy. After one expression is evaluated copy and paste the expression to the rest of the cells desired so the formula carries through. After saving the drawing or assembly (both have access to the Bill of Material) the changes will be made in the part files.





About the Author: Mark Flayler

Mark is an Application Engineer with IMAGINiT Technologies, specializing in manufacturing environments. He has implemented the Autodesk Manufacturing products with several industries including the blow/injection molding, automotive, and custom machinery markets. Inventor has been a profound augmentation in his abilities allowing him to bring 3D digital prototyping to the forefront of the industries with which he has interacted. He has extensive experience and a comprehensive understanding of the technical, practical business and human dimensions of implementation. He is an effective and skillful communicator, consulting with his clients to help achieve their business objectives. Mark is an ATC certified instructor and he provides training, support, and implementation. Mark is PSE and ATC certified in AutoCAD®, AutoCAD® Mechanical, AutoCAD® Electrical, Autodesk® Data Management, and Autodesk® Inventor™.

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