

# Autodesk Manufacturing & Digital Prototyping Solutions

[Home](#)

[Blog Roll](#)

[Tips & Tricks](#)

[Digital Prototyping](#)

[Autodesk](#)

13/07/2011

## Autodesk Inventor: Quick Tip - Tolerances Part 3

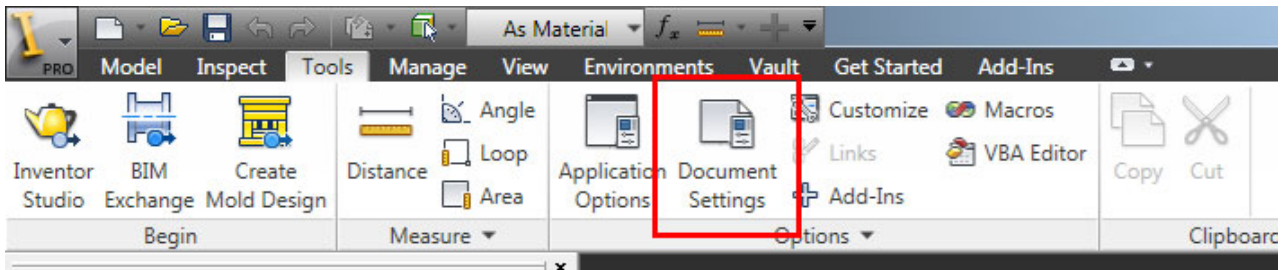
Just like my favourite movie Back to the Future, all good things come in threes and here's the third and final installment of using tolerances in your 3D models with [Autodesk Inventor](#).

So far we've had a look at [adding tolerances to 3D parts](#) so that the data can be re-used in the associative 2D drawings, but also how we can [re-evaluate the 3D model](#) based on the upper, lower and median tolerance to check things like fit and function.

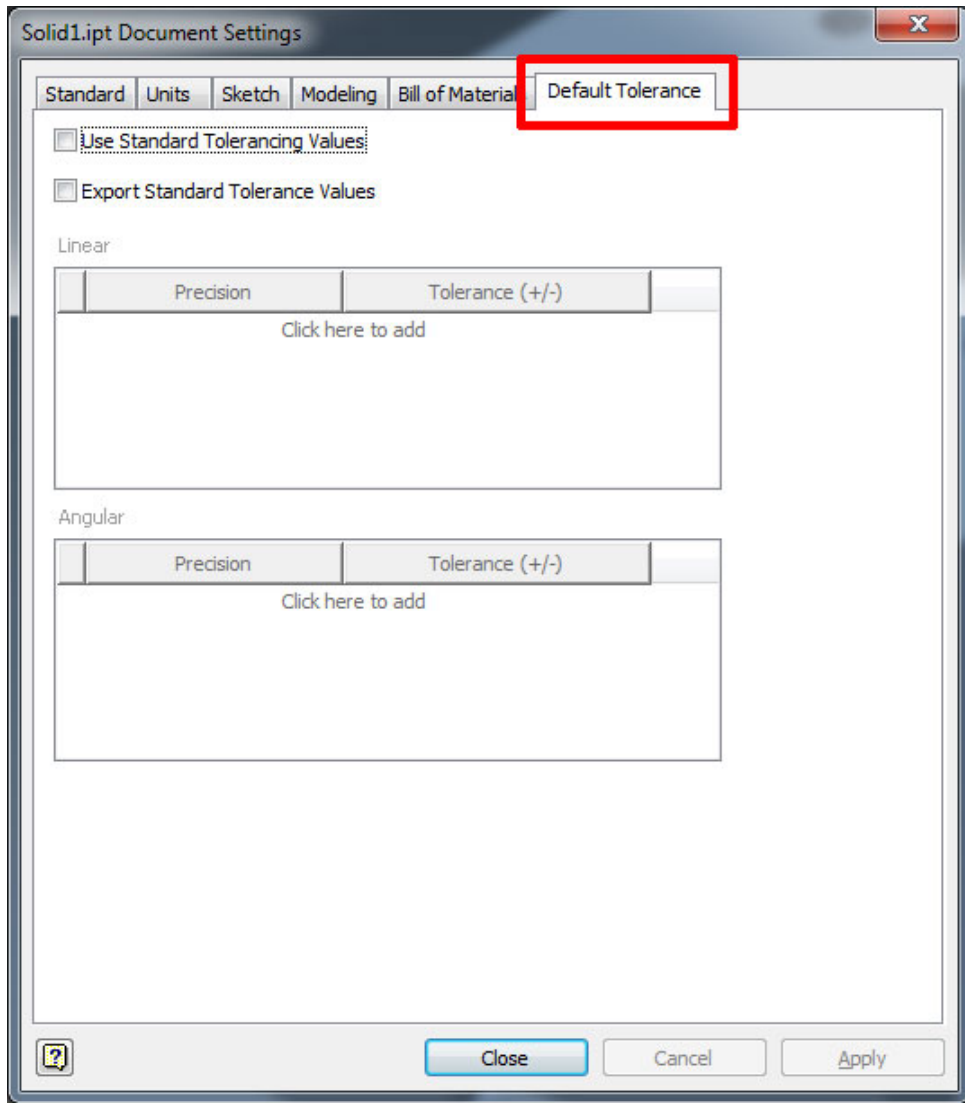
This final part of the trilogy looks at how we can set general tolerance values to comply with company\project standards.

Many engineers and manufacturers work to general tolerances that are set either in company standards or at the beginning of a project. These tolerances are applied to all dimensions so that manufacturing are using the same tolerance values across all parts and features. Autodesk Inventor allows users to set general tolerances within the Document Settings and apply these standard values to the 3D model automatically.

First, go to the Document Settings option on the Tools ribbon...

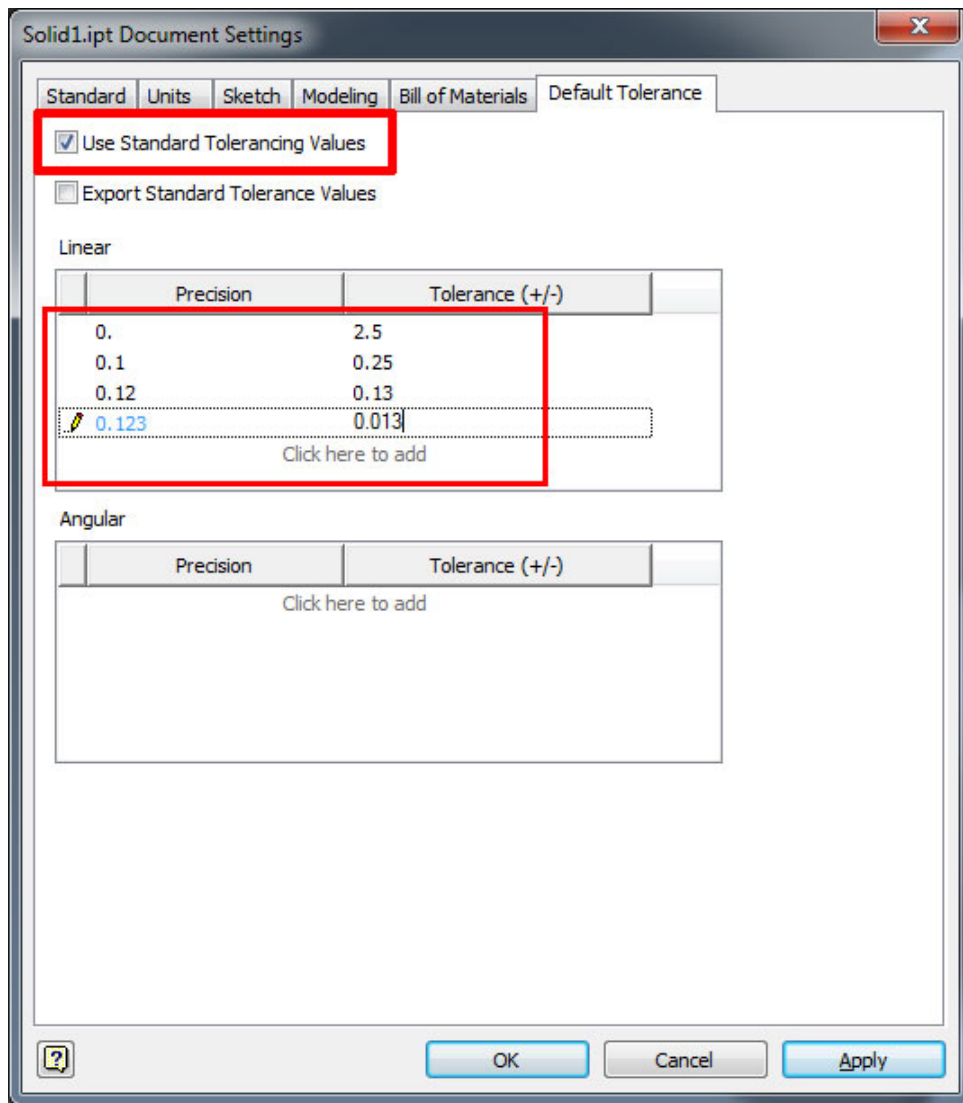


And then from there just select the Default Tolerance tab....



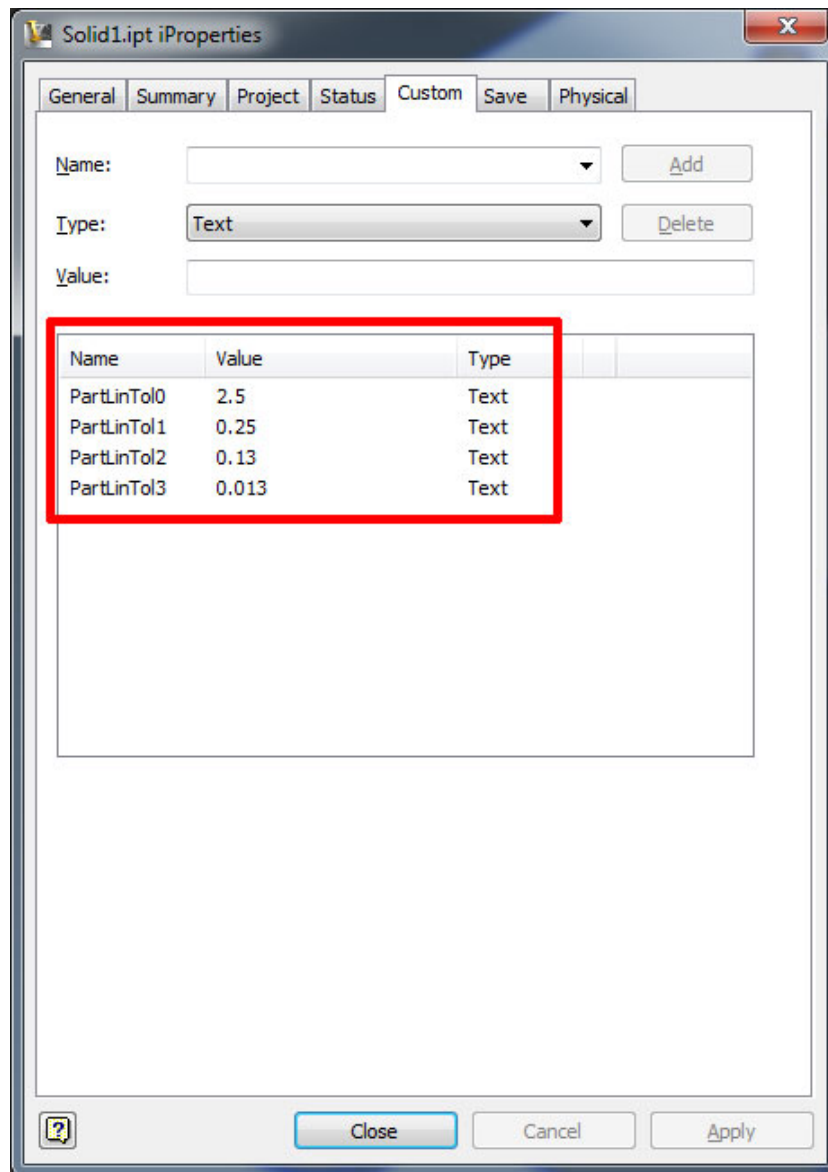
The Document Settings dialogue allows you set default settings for the current part (.IPT) document you are in. The General Tolerance tab allows you to apply the tolerance values for each precision as well as export these values to your document iProperties.

When you add a tick in the Use Standard Tolerancing Values option you will be able to add Linear and Angular tolerance values against a precision. Just select a precision from the Precision drop list and then type in the tolerance value you wish to apply to any dimensions using that precision:

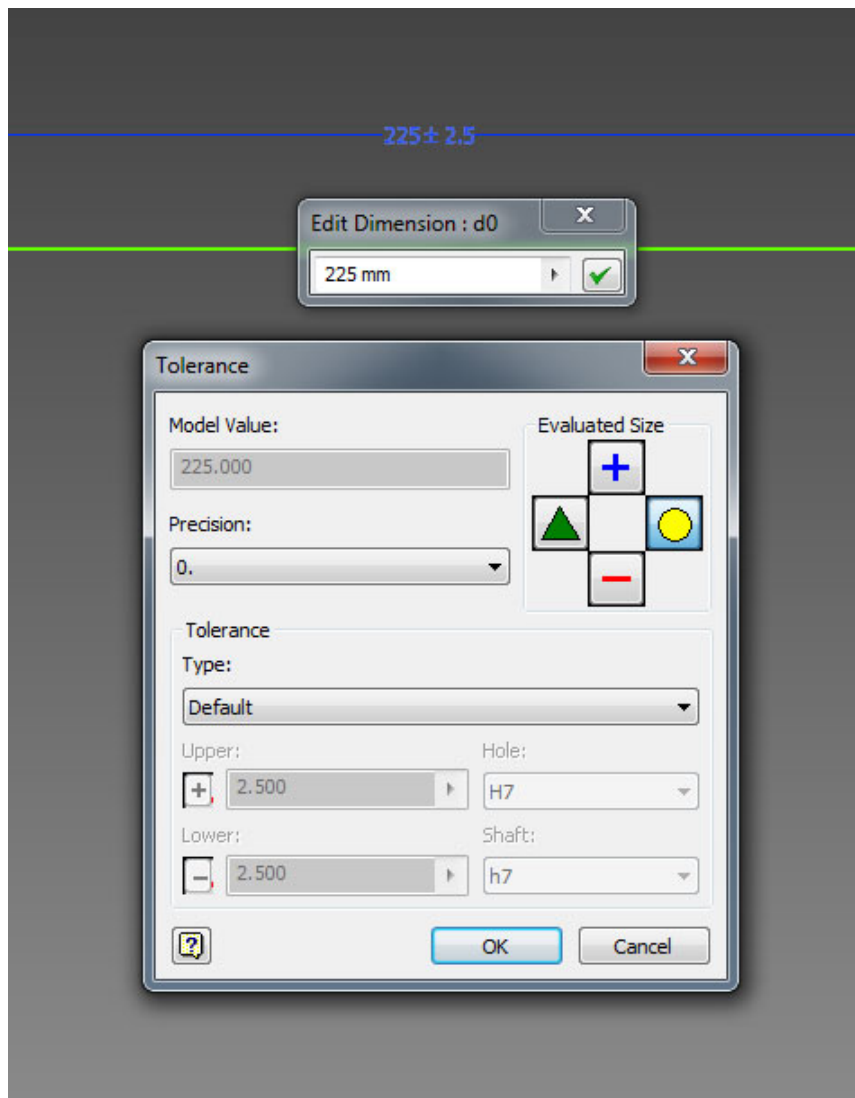


You don't have to apply a tolerance for all precision values, you can add just the ones you use.

If you add a tick in the Export Standard Tolerance Values option, the values you input as standard tolerances will be exported to the iProperties of the part file you are in. Keep reading to see how we can use these values in our drawing:

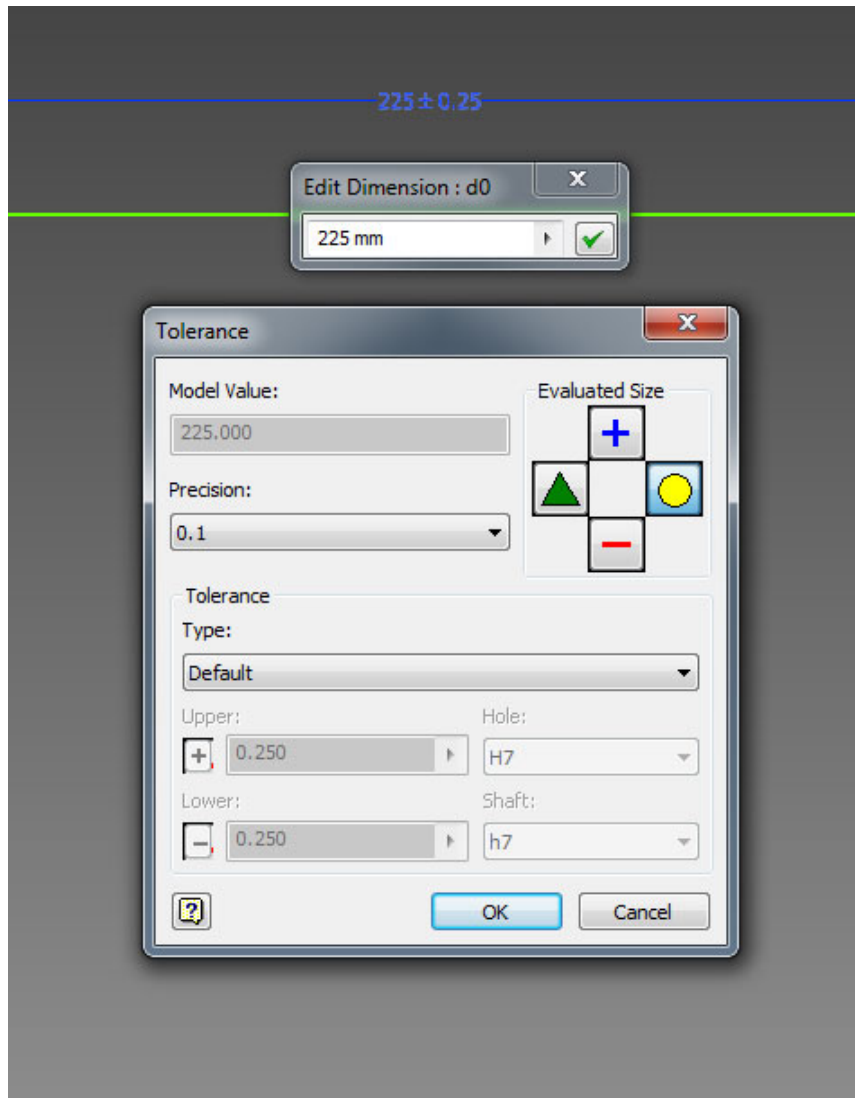


Once the standard tolerance values have been set you can use them in your 2D sketches to apply the tolerance value using the Tolerance... option in the Edit Dimension tool. Just select the precision and make sure the type is set to default and the standard tolerance for that precision will be applied:



You can see in the example above that by selecting a Precision of 0. and having the Type set to Default a tolerance of 2.5mm (upper & lower) has been automatically applied.

In the example below I have chosen a Precision of 0.1 meaning the tolerance has been set automatically to 0.25mm:



So now we have applied the tolerances to our 3D part, when we come to create the detail drawings how can we make sure that all of the tolerance values are automatically annotated?

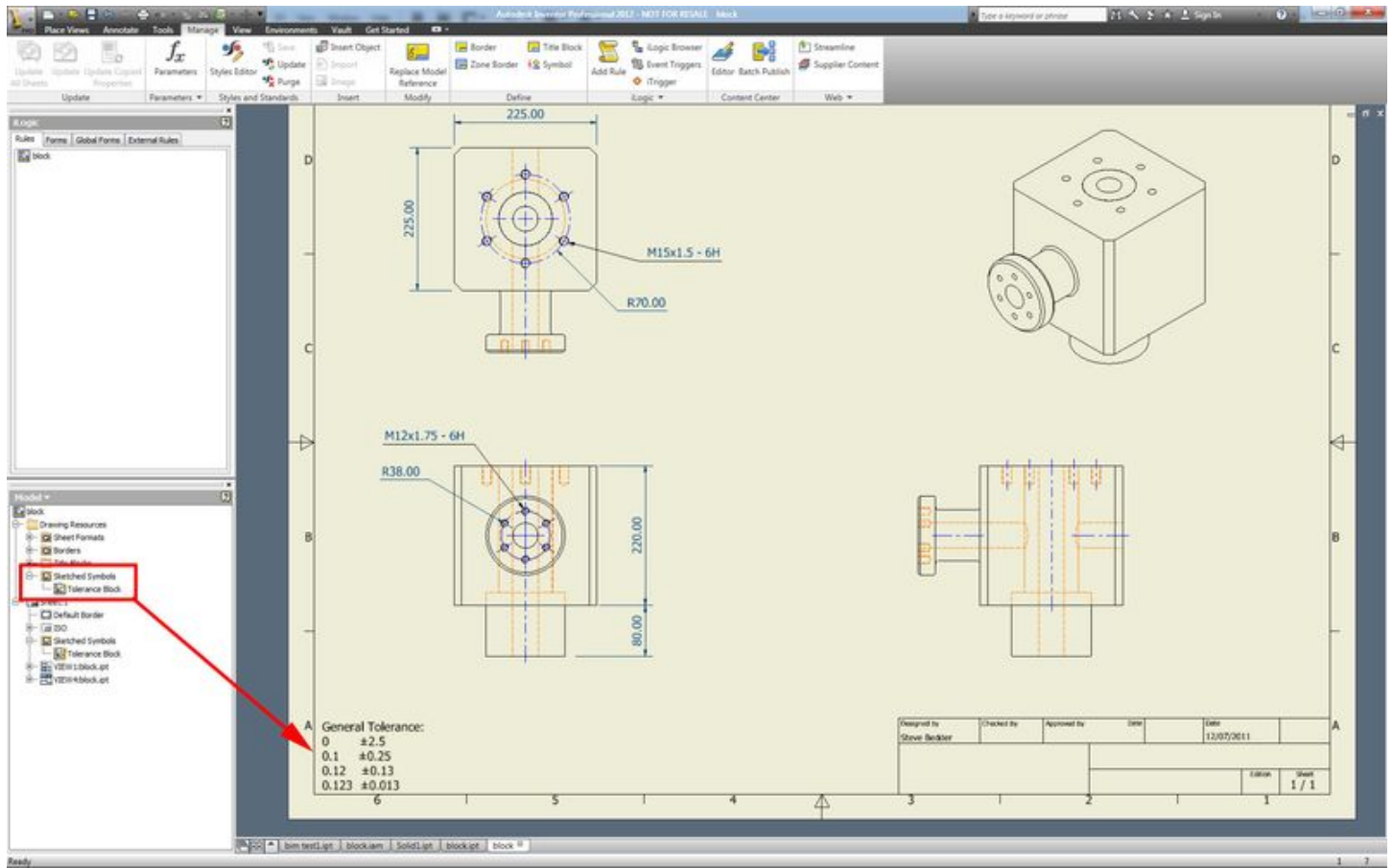
The fact that we selected to Export Standard Tolerance Values so that they are available in the part iProperties means we can re-use those values in the drawing. The way I do this is to create a Sketch Block which contains the following:

```

General Tolerance:
0      ±<PartLinTol0>
0.1    ±<PartLinTol1>
0.12   ±<PartLinTol2>
0.123  ±<PartLinTol3>

```

The <ParLinTolo> values are text values that are linked to the part document meaning the value will be pulled from the part documents iProperties. So whatever the standard tolerance values are in the part document, they will be automatically shown on the drawing. Having this as a sketch block in the 2D drawing means I can add it to any drawing, at any time to annotate the values automatically:

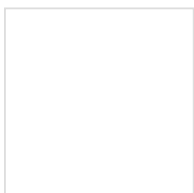


Adding the standard tolerance values and sketch block to your part template(s) will mean that every part and drawing you create will already contain the standard tolerance values and sketch blocks you need saving you having to apply them every time.

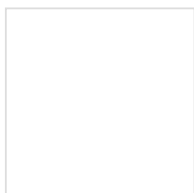
**You might also like:**



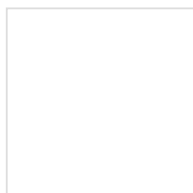
Autodesk Inventor: Quick Tip - Tolerances Part 2



Autodesk Inventor: Quick Tip - Tolerances



Autodesk Inventor: Quick Tip - Managing Parameters



Autodesk Inventor 2012 - Whats New? - Part 2

[Linkwithin](#)

Posted by [Steve Bedder](#) on 13/07/2011 in [Inventor, Tips & Tricks](#) | [Permalink](#)

Tweet | 0

| Like 12

**Comments**



[lance cockle](#) said...

I have included these in the template files and have included this in my title block for parts. It fills it in on part placement, automatically :)

How can I apply this to assembly drawings as well. I would like to specify tolerances there as well.

[Reply 21/09/2011 at 04:13 AM](#)



[Sunith Babu](#) said...

Good Post

[Reply 14/07/2011 at 12:48 AM](#)

Comments on this post are closed.

POWERED BY  TypePad

