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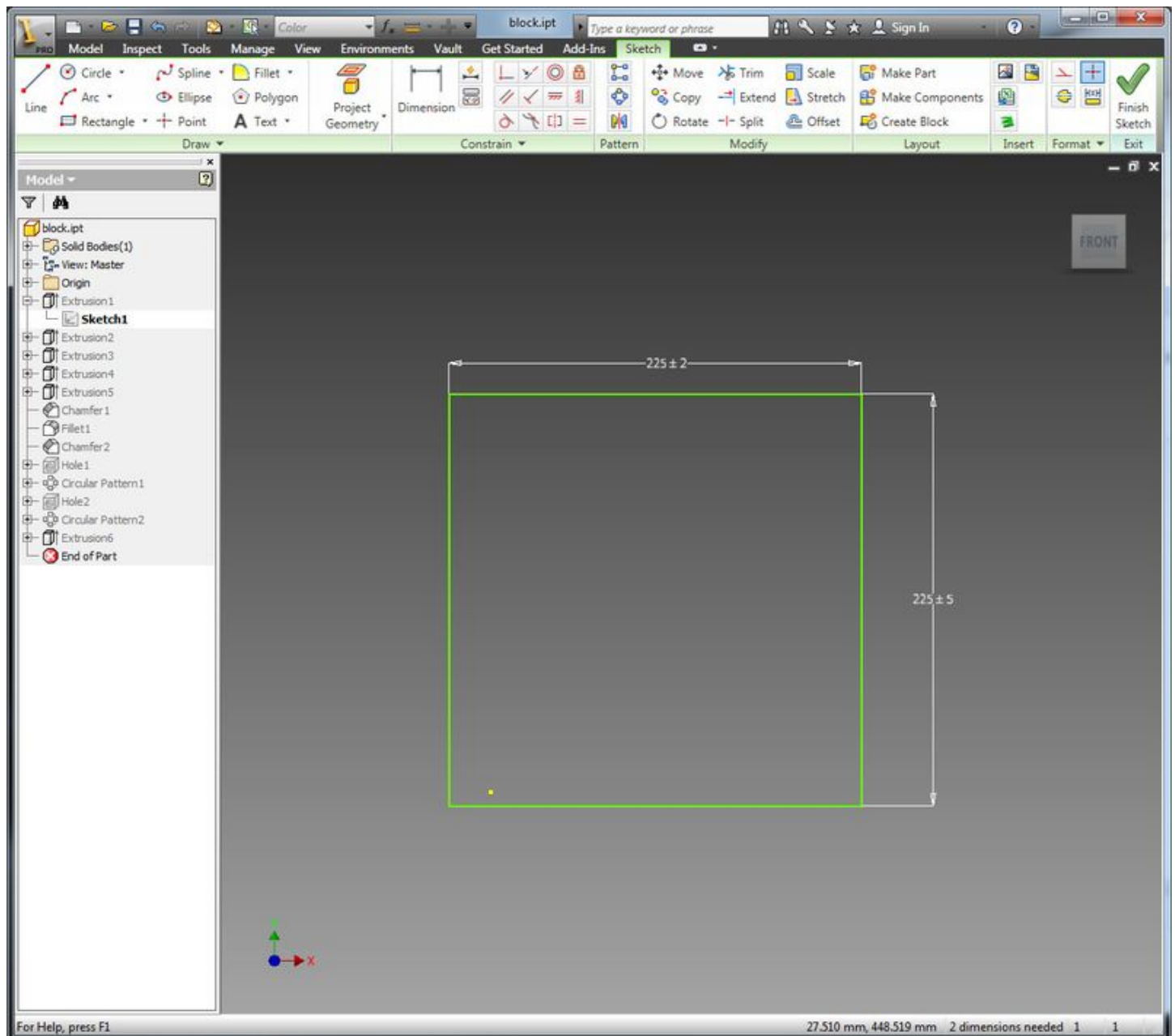
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08/07/2011

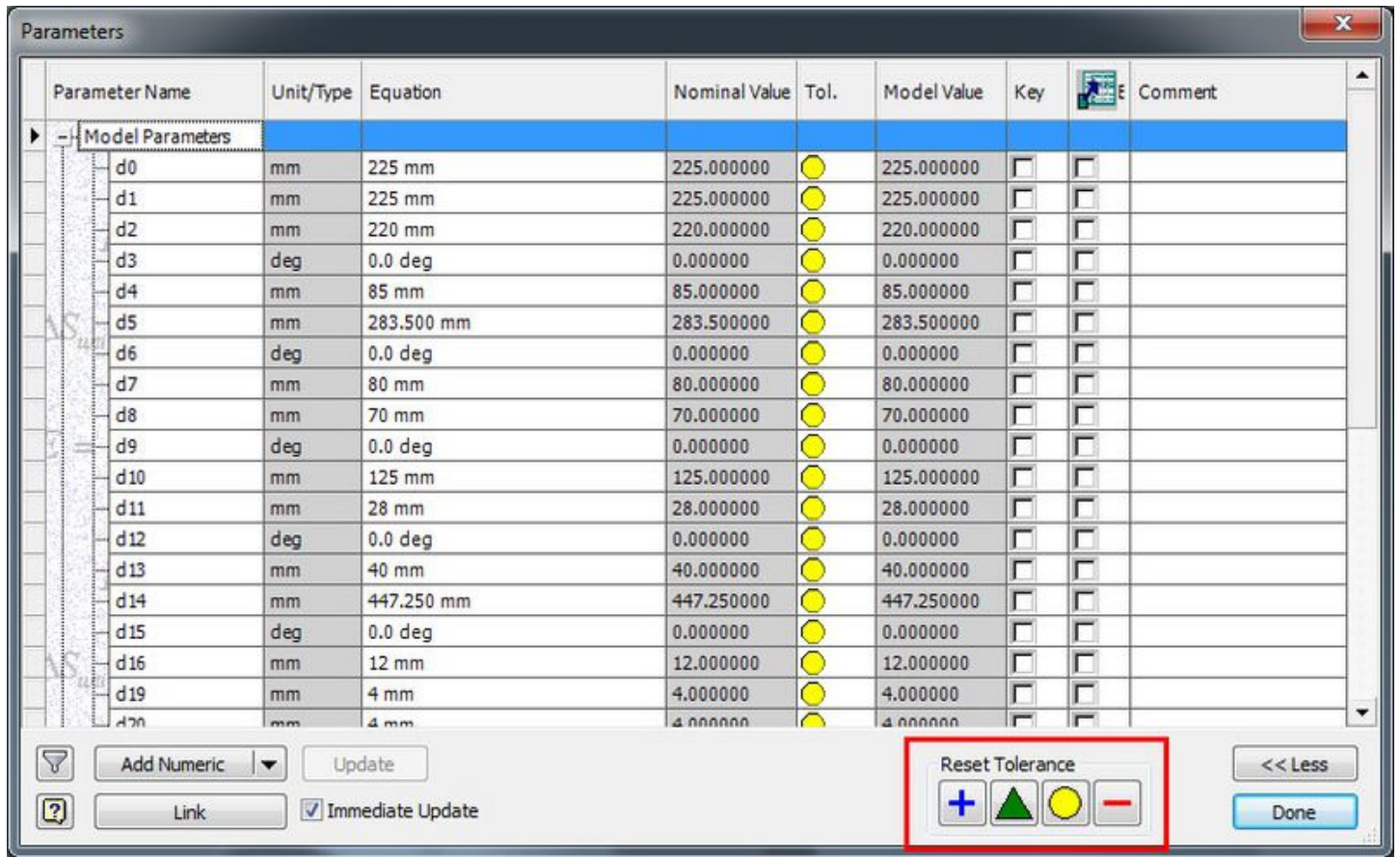
Autodesk Inventor: Quick Tip - Tolerances Part 2

On Wednesday I posted an Autodesk Inventor [Quick Tip all about adding tolerances to your 3D parts](#) so that you can automatically annotate your 2D drawings with the tolerance data. If you update the dimensions and tolerances in the 3D model - the tolerance data automatically updates on the drawing! Today I'm going focus on using the tolerance data in the 3D part(s) to re-evaluate the parts and see how manufacturing at the upper\lower tolerance can affect fit and function.

Using the same part we used on Wednesday I have edited the tolerance values in the 2D sketch so that we can see the effect of re-evaluating the model using the tools in Autodesk Inventor...



To update the 3D part so that it using the tolerance values to size the part, all you need to do is open the Parameters dialogue.



The parameters dialogue includes a Tol. column which as you see above shows a yellow circle for all dimensions. This means all dimensions are currently set to their nominal value e.g. the dimension value you entered into the sketch. However, we can re-evaluate all the dimensions in one go using the four options in the Reset Tolerance tools (from left to right):

- Upper Tolerance
- Median Tolerance
- Nominal Tolerance
- Lower Tolerance

Clicking on one of the options will set all of the dimensions in the parameters dialogue to the tolerance value:

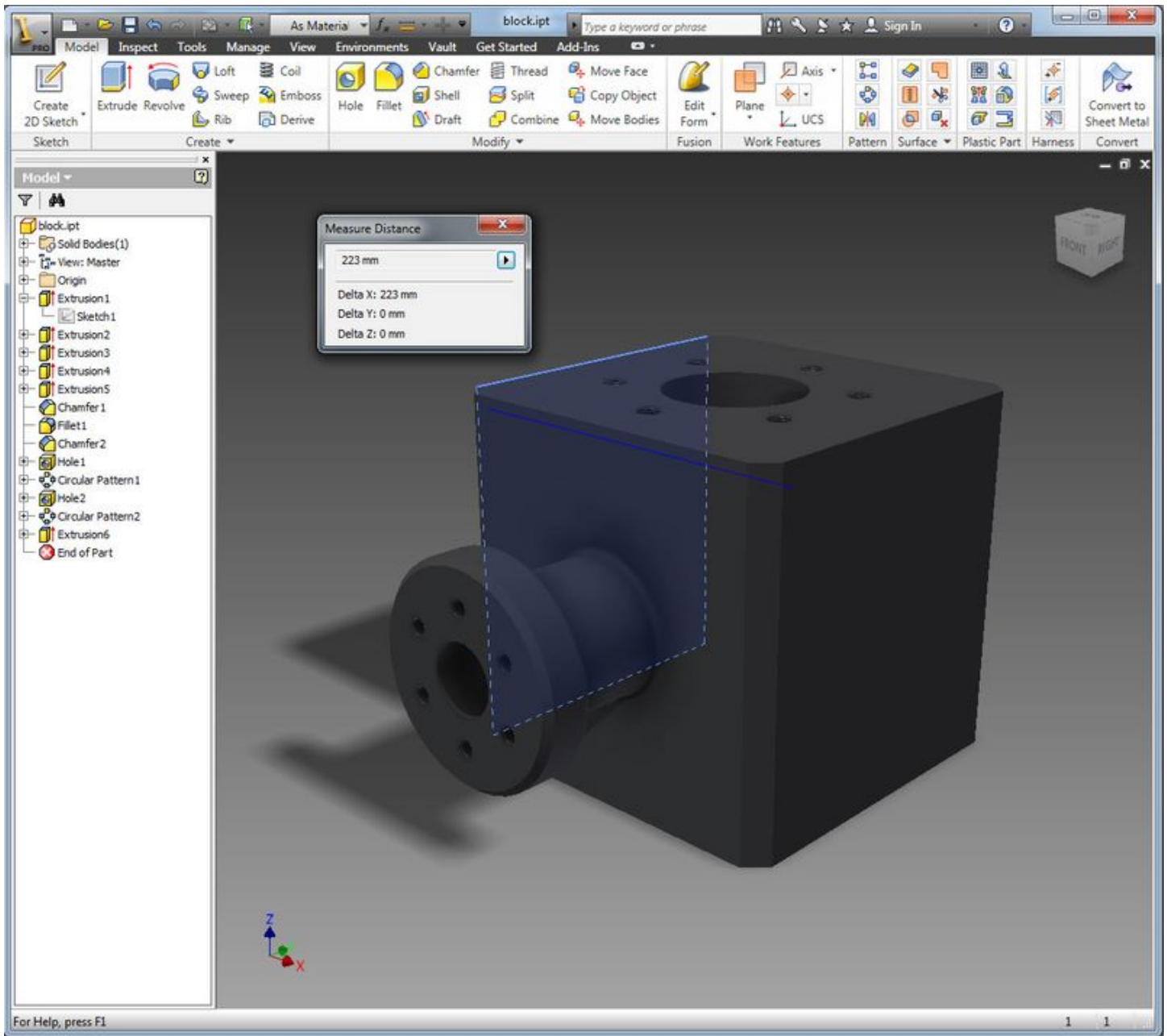
Parameter Name	Unit/Type	Equation	Nominal Value	Tol.	Model Value	Key	Comment
Model Parameters							
d0	mm	225 mm	225.000000	+	227.000000		
d1	mm	225 mm	225.000000	+	230.000000		
d2	mm	220 mm	220.000000	+	220.000000		
d3	deg	0.0 deg	0.000000	+	0.000000		
d4	mm	85 mm	85.000000	+	85.200000		
d5	mm	283.500 mm	283.500000	+	283.500000		
d6	deg	0.0 deg	0.000000	+	0.000000		
d7	mm	80 mm	80.000000	+	80.000000		
d8	mm	70 mm	70.000000	+	70.000000		
d9	deg	0.0 deg	0.000000	+	0.000000		
d10	mm	125 mm	125.000000	+	125.000000		
d11	mm	28 mm	28.000000	+	28.000000		
d12	deg	0.0 deg	0.000000	+	0.000000		
d13	mm	40 mm	40.000000	+	40.000000		
d14	mm	447.250 mm	447.250000	+	447.250000		
d15	deg	0.0 deg	0.000000	+	0.000000		
d16	mm	12 mm	12.000000	+	12.000000		
d19	mm	4 mm	4.000000	+	4.000000		
d20	mm	4 mm	4.000000	+	4.000000		

You can see in the example above, by clicking on the Upper Tolerance option, the Tol. column now shows all the dimensions with a + meaning they are all now at their upper tolerance value. The highlighted values are the two dimensions I've applied tolerances to and if you look at the model value you can see that they are different to the nominal value - they now include the addition of the upper tolerance values I applied in the 2D sketch.

If I select the Lower Tolerance option, you can see in the screen shot below, all of the dimension values update to the lower tolerance values I applied in the 2D sketch:

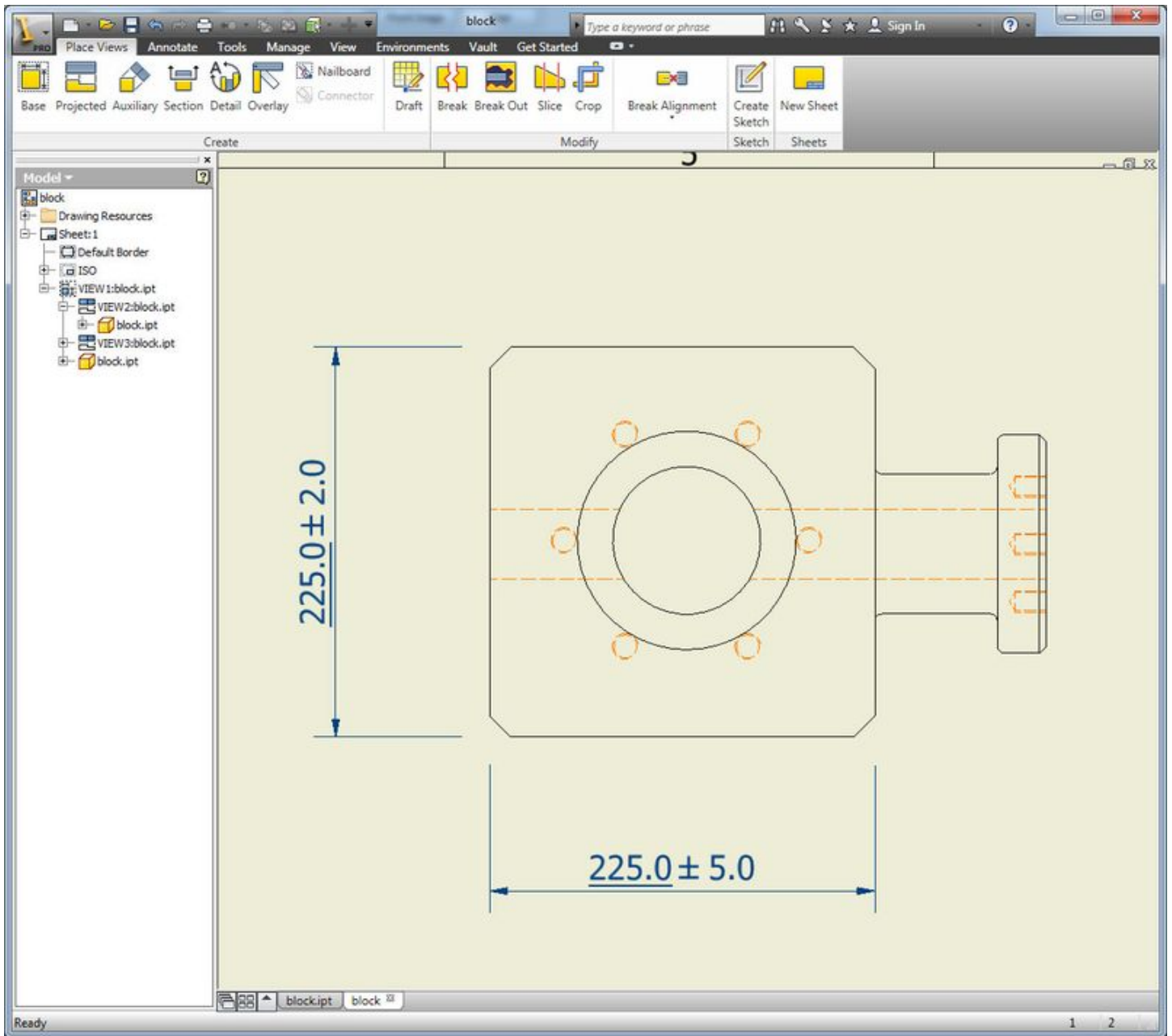
Parameter Name	Unit/Type	Equation	Nominal Value	Tol.	Model Value	Key	Comment
Model Parameters							
d0	mm	225 mm	225.000000	-	223.000000		
d1	mm	225 mm	225.000000	-	220.000000		
d2	mm	220 mm	220.000000	-	220.000000		
d3	deg	0.0 deg	0.000000	-	0.000000		
d4	mm	85 mm	85.000000	-	84.500000		
d5	mm	283.500 mm	283.500000	-	283.500000		
d6	deg	0.0 deg	0.000000	-	0.000000		
d7	mm	80 mm	80.000000	-	80.000000		
d8	mm	70 mm	70.000000	-	70.000000		
d9	deg	0.0 deg	0.000000	-	0.000000		
d10	mm	125 mm	125.000000	-	125.000000		
d11	mm	28 mm	28.000000	-	28.000000		
d12	deg	0.0 deg	0.000000	-	0.000000		
d13	mm	40 mm	40.000000	-	40.000000		
d14	mm	447.250 mm	447.250000	-	447.250000		
d15	deg	0.0 deg	0.000000	-	0.000000		
d16	mm	12 mm	12.000000	-	12.000000		
d19	mm	4 mm	4.000000	-	4.000000		
d20	mm	4 mm	4.000000	-	4.000000		

So now we have reset all of the dimensions to the lower tolerance value I selected in the Parameters dialogue, lets see how that effects the 3D part:























You can see from the screen shot above the model is now smaller, if I measure the width of the part I can see that it is 223mm rather than the 225mm I applied in the sketch. This is because I have select to re-evaluate this model at the lower tolerance so 2mm has been taken off the width.

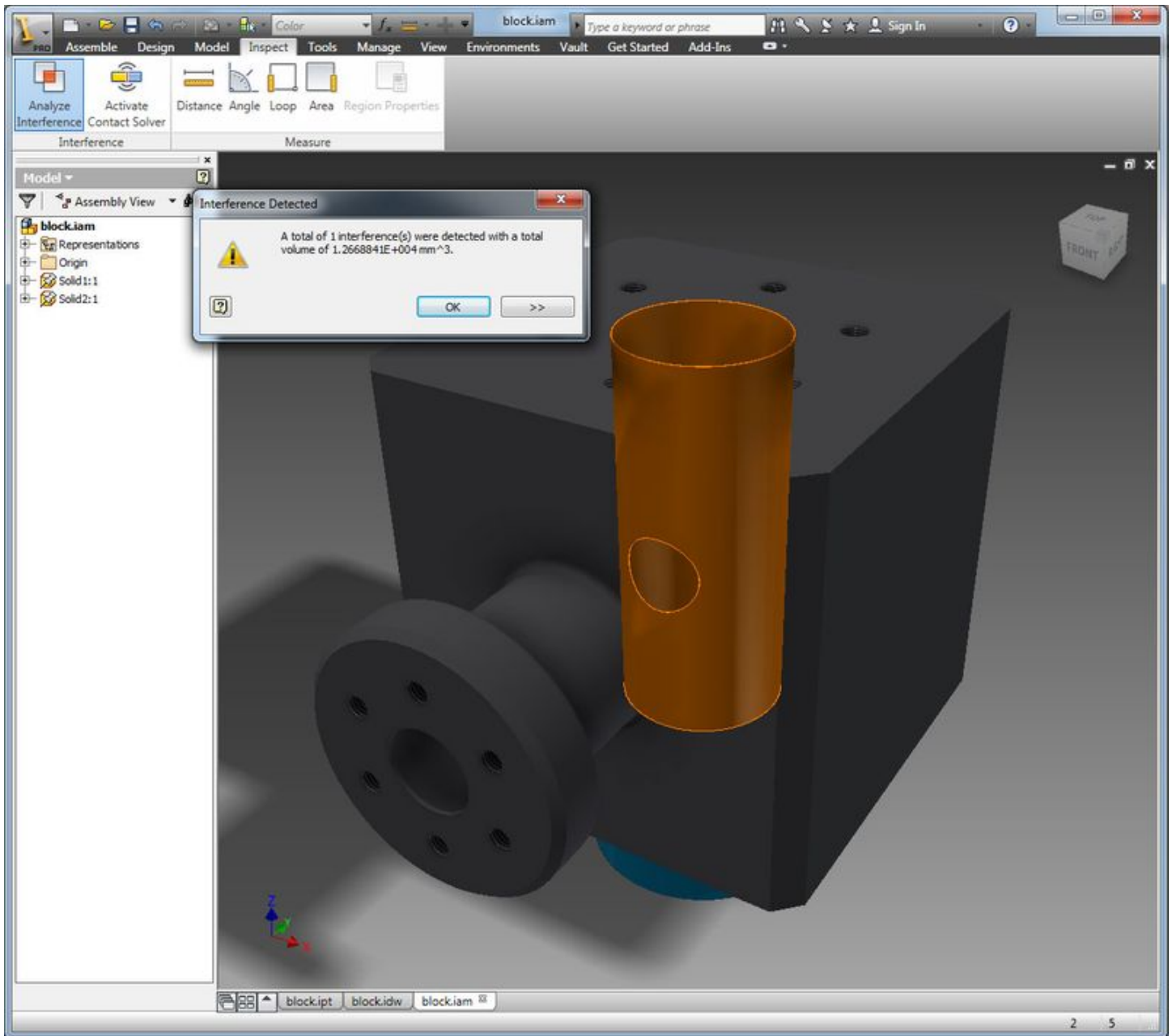
When we see the drawing detail of this, Autodesk Inventor shows in the annotation that the dimension has been re-evaluated by adding an underline to any dimension that has an upper, lower or median tolerance applied in the 3D part:



We don't have to apply tolerances to all dimensions at once, we can selectively choose which dimensions we wish to re-evaluate by selecting the tolerance from the drop list in the parameters dialogue. This allows us to mix upper, lower, median and nominal in different sketches:

Parameter Name	Unit/Type	Equation	Nominal Value	Tol.	Model Value	Key		Comment
Model Parameters								
d0	mm	225 mm	225.000000		225.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d1	mm	225 mm	225.000000		225.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d2	mm	220 mm	220.000000		220.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d3	deg	0.0 deg	0.000000		0.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d4	mm	85 mm	85.000000		85.200000	<input type="checkbox"/>	<input type="checkbox"/>	
d5	mm	203.500 mm	203.500000		203.500000	<input type="checkbox"/>	<input type="checkbox"/>	
d6	deg	0.0 deg	0.000000		0.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d7	mm	80 mm	80.000000		80.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d8	mm	70 mm	70.000000		70.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d9	deg	0.0 deg	0.000000		0.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d10	mm	125 mm	125.000000		125.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d11	mm	28 mm	28.000000		28.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d12	deg	0.0 deg	0.000000		0.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d13	mm	40 mm	40.000000		40.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d14	mm	447.250 mm	447.250000		447.250000	<input type="checkbox"/>	<input type="checkbox"/>	
d15	deg	0.0 deg	0.000000		0.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d16	mm	12 mm	12.000000		12.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d19	mm	4 mm	4.000000		4.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d20	mm	4 mm	4.000000		4.000000	<input type="checkbox"/>	<input type="checkbox"/>	

Having 3D parts in an assembly that have been re-valuated at different tolerances gives us the ability to check for clashes and interference so we can see exactly how components will behave with each other when manufactured at different tolerances:



Thanks to [Mike Watkins](#) for the prompt to put these two posts together.

Posted by [Steve Bedder](#) on 08/07/2011 in [Inventor, Tips & Tricks](#) | [Permalink](#)

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Comments



[Rodney Sparks](#) said...

It seems like it'd be a good idea to enter either "external" or "internal" dimension type to the "comments" column upon dimension creation, if possible (I haven't looked into it). That way, values can be set to minimum for internal dims and maximum for external dims to reflect MMC, for example. Setting all dimensions to maximum wouldn't be MMC, since both the internal and external dims will increase.

[Reply 11/07/2011 at 02:33 PM](#)

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