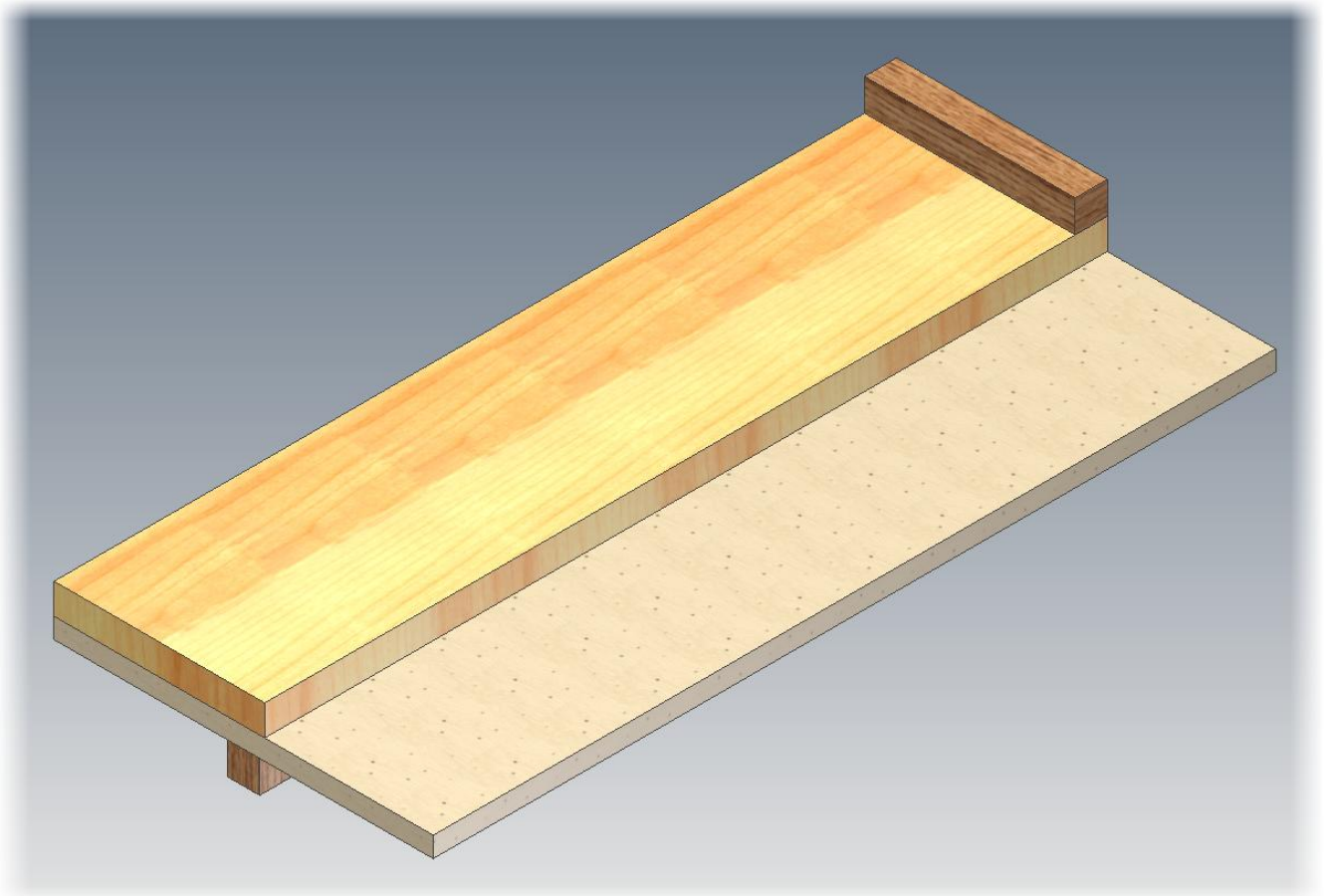


Modelling a Shooting Board



This tutorial will show you how to create a simple assembly model with Autodesk Inventor, using the 'In place' modelling technique. Using the in place technique, we will start with an assembly file and then build each part inside the assembly. This tutorial also demonstrates how to use 'Adaptive' parts to control the size of an assembly model from one base part.

This article is aimed at novice users. However I am assuming that you are familiar with the concepts of parametric modelling, and that you've had some time to explore the Inventor user interface.

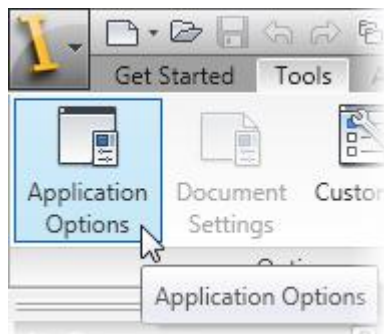
In this tutorial we will use the following workflow:

- Create an Assembly
- Create a new Part 'In place'
- Add parameters
- Create a sketch
- Constrain the sketch
- Add a sketch based feature (an extrusion)
- Change the look of a part
- Create the next part
- Link the parts together to create an adaptive Assembly

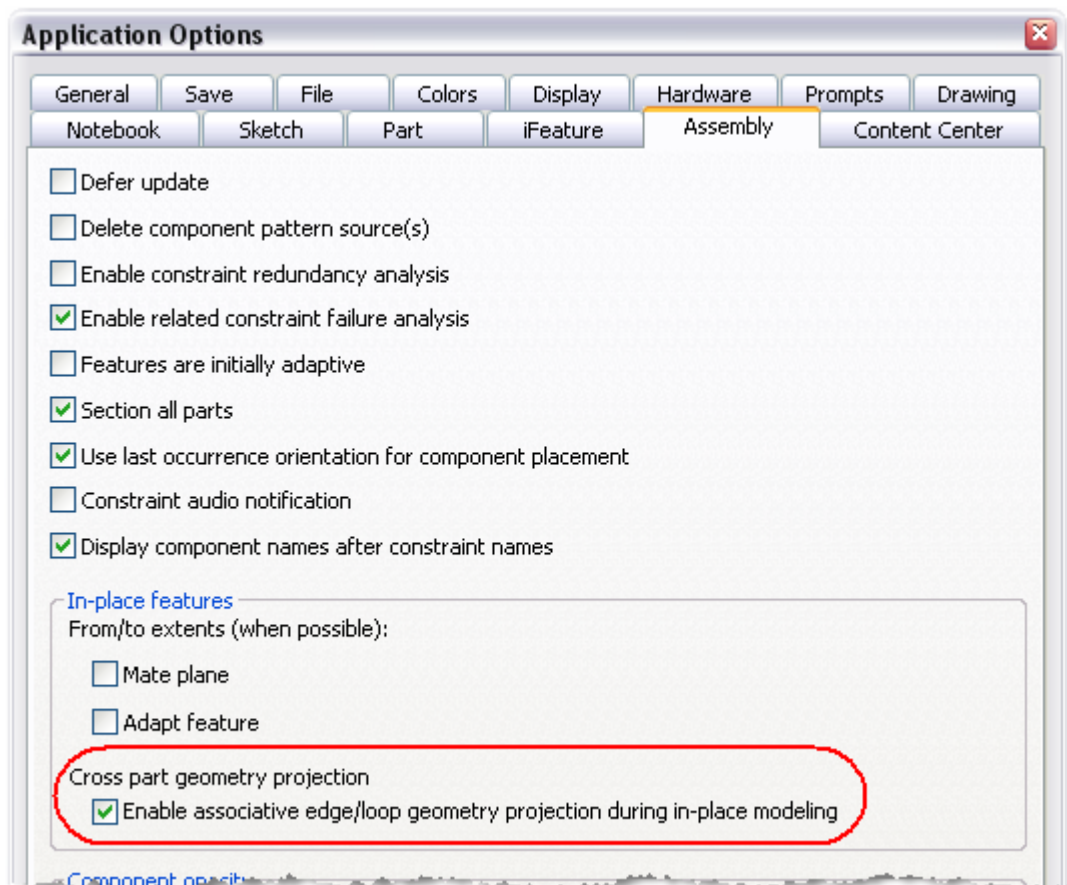
Getting set up

For the purposes of this tutorial, please open your Application Options...

Tools Tab > Options Panel > Application options

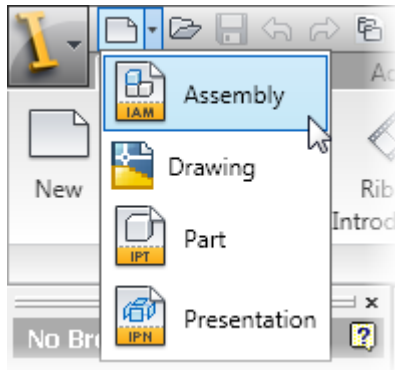


... And make sure that 'Enable Associative Edge/Loop Geometry Projection during In-Place Modelling' is checked

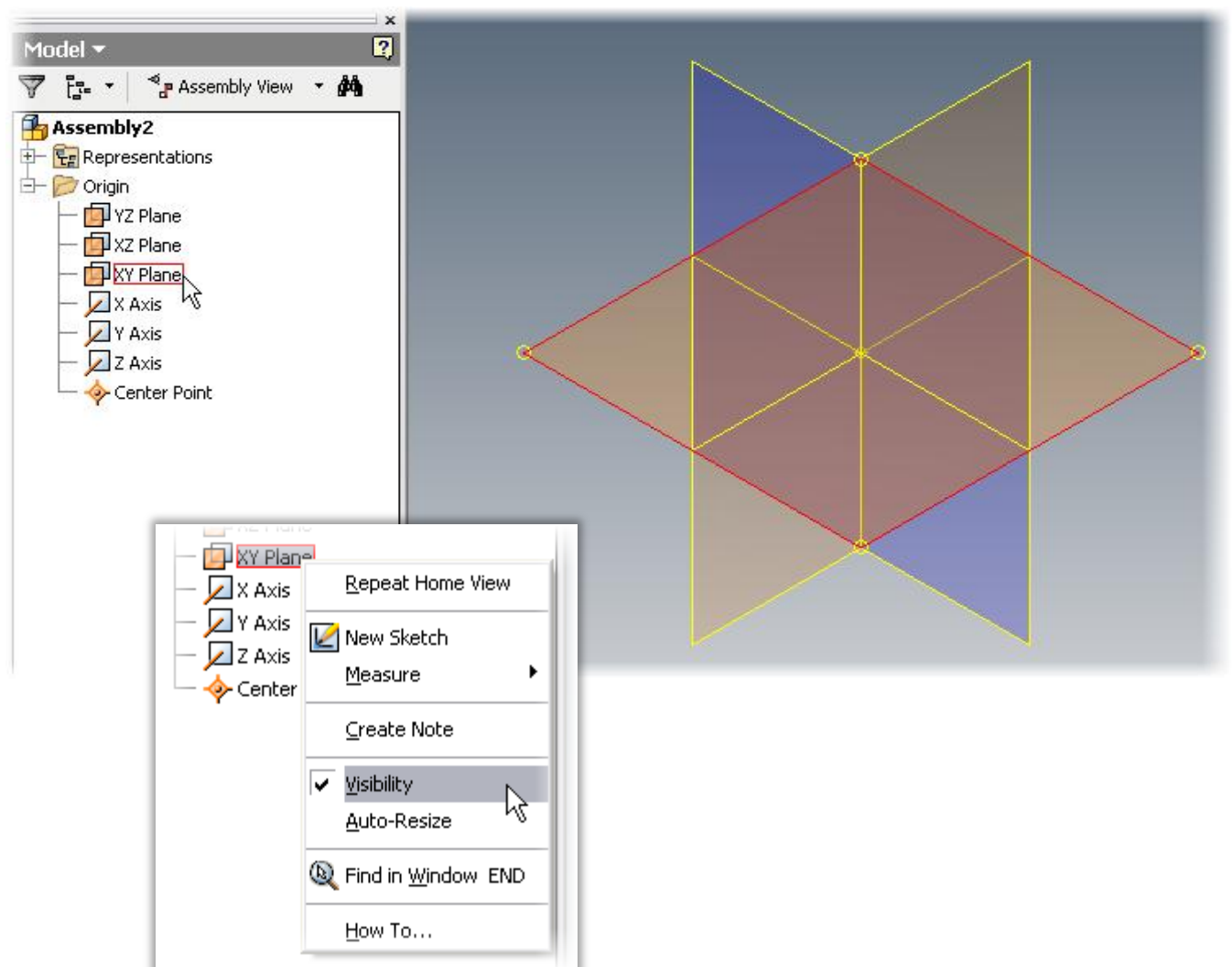


Creating the Assembly

Open a new Assembly file.



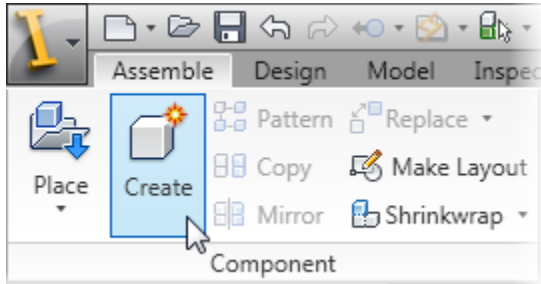
Expand the Origin folder and note the different planes available. Hover over each one to see it's outline, or right click and chose 'Visibility' to see the origin planes on screen.



Creating the first part

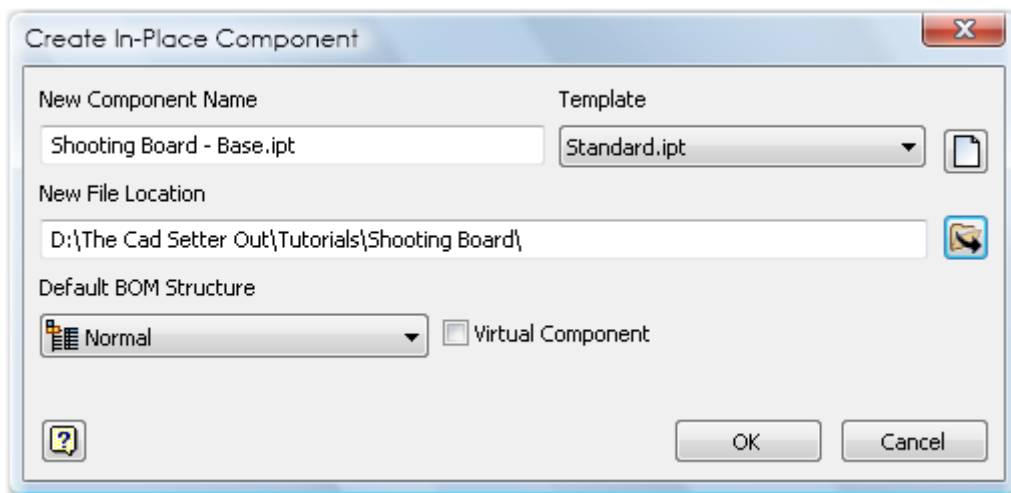
Use the 'Create Component' tool to create the first part.

Assemble Tab > Component Panel > Create Tool



You will be prompted for the following:

- Name of your part
Shooting Board- Base
- Template File
For this example, use the standard template
- Add a file path
The default file path is set in your project file
- 'BOM' structure
This comes into play when you are creating cutting tickets – you can leave it set to Normal

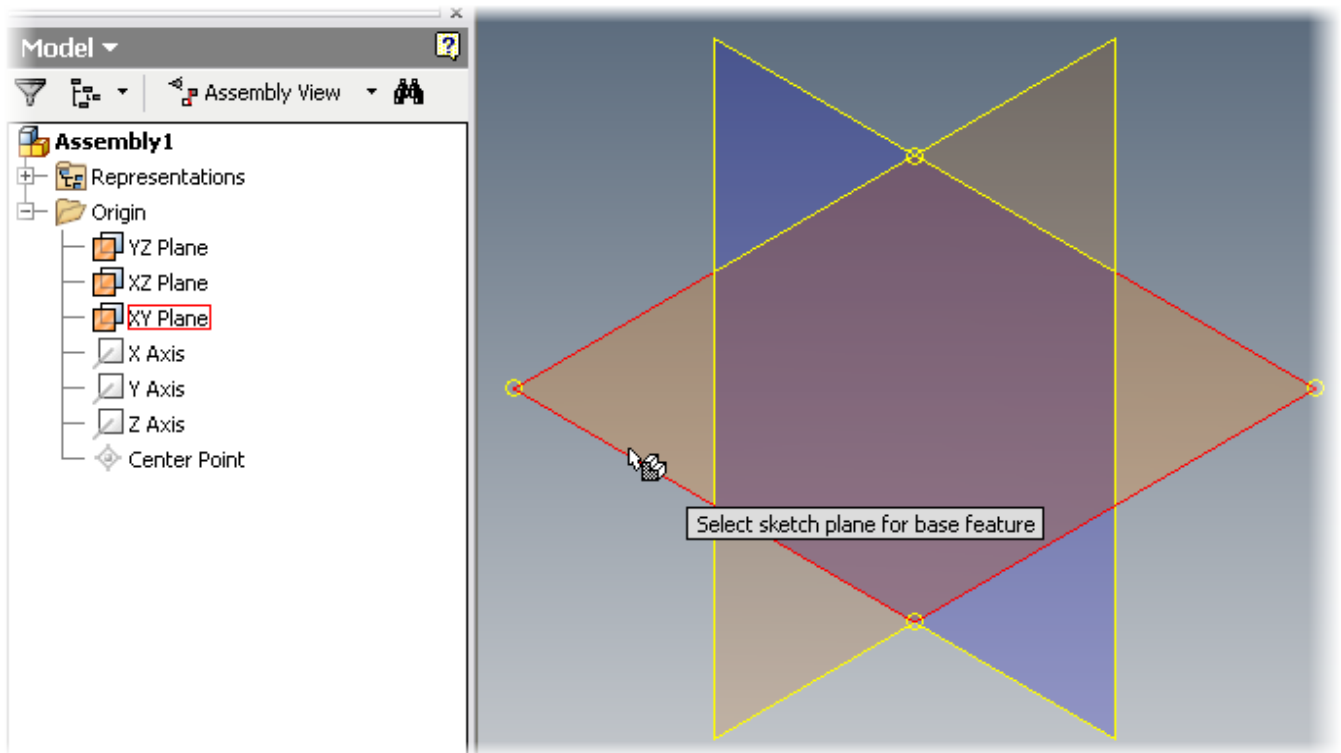


Make sure that 'Virtual component' is unchecked.

Modelling a Shooting Board

Pick the XY plane as your first sketch plane.

You could pick any plane or surface here – later on in the tutorial we will pick one of our existing parts as a base surface.



Note that I only have the origin planes visible in this picture to help you see what's happening. You do not need to make the origin planes visible in order to select them – just pick them from the browser.

Modelling a Shooting Board

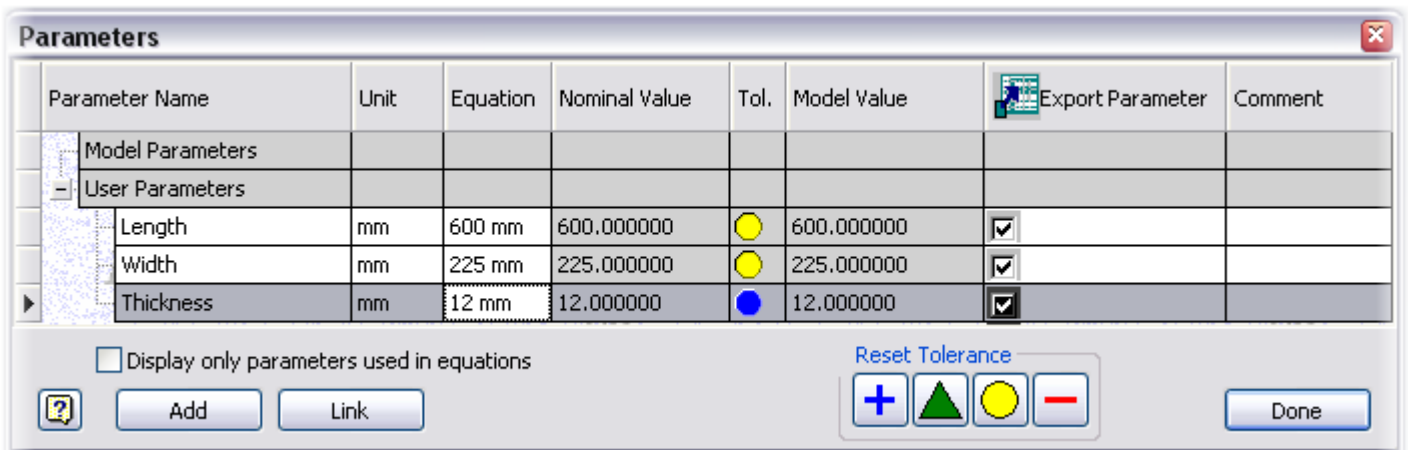
Add Parameters

Manage Tab> Parameters Panel> Parameters tool

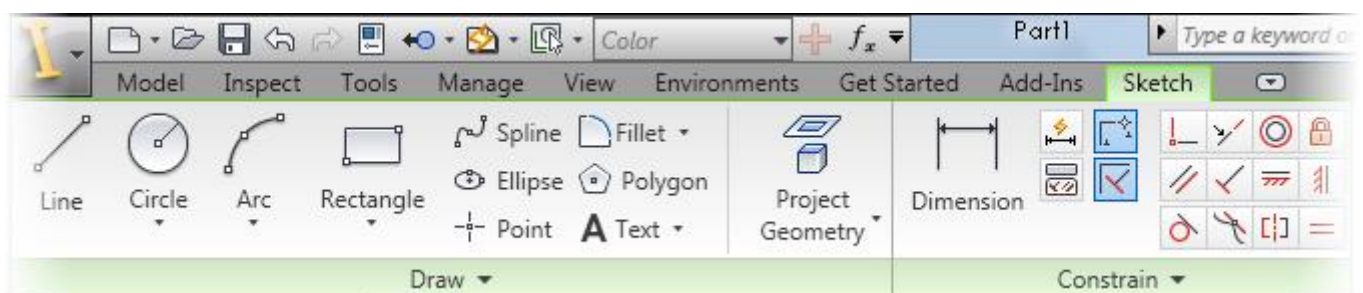


Name	Value
Length	600
Width	225
Thickness	12

Use the 'Add' Button or ALT+A to add parameters.



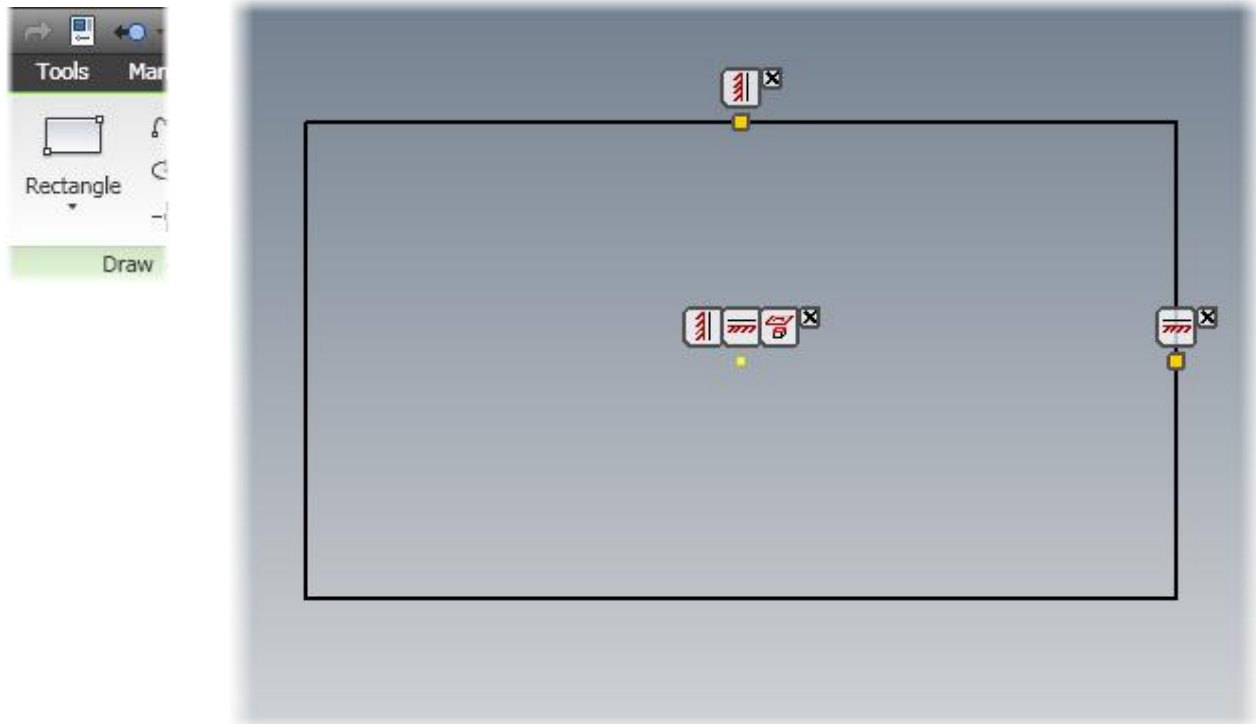
- Click the 'Done' button when you have finished adding parameters and, if you need to, double click on the default 'Sketch1' to return to the sketch environment. You may notice that once you are in the sketch environment, the sketch tab in the ribbon is tinted green. Should you need to switch to a different tab this will help to guide you back to the currently active tab.



Modelling a Shooting Board

- Sketch out a rectangle

Sketch Tab>Draw Panel>Rectangle Tool



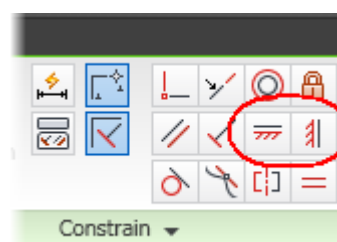
- If you need to, use the 'Project Geometry' tool to project the origin into the current sketch.

Sketch Tab > Draw Panel > Project geometry Tool



- Use the Horizontal and Vertical constraint tools to centre your rectangle about the origin.

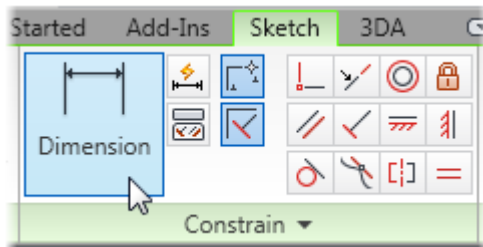
Sketch Tab>Constrain Panel>Vertical and Horizontal Constraint Tools



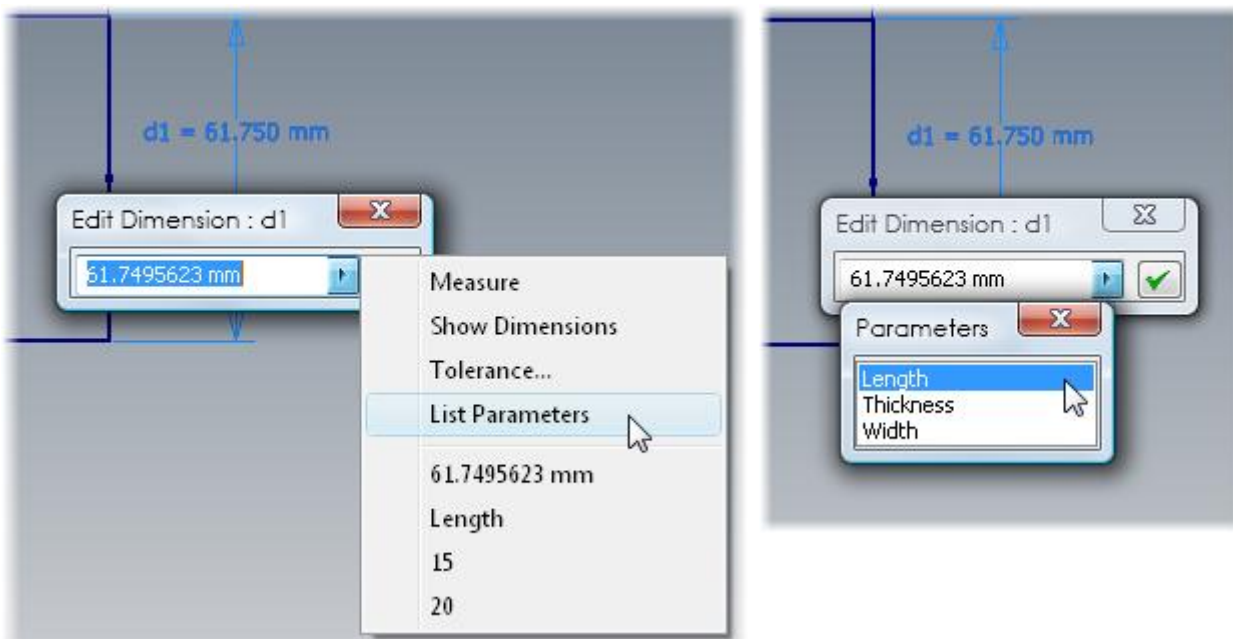
Modelling a Shooting Board

- Add dimensional constraints to your sketch

Sketch Tab >Constrain Panel >Dimension Tool



Single click on a dimension constraint to edit its value whilst using the dimension tool, or double click on a dimension constraint to edit its value at any other time. Click on the arrow at the end of the edit box and choose 'List parameters' to choose from the list of parameters that you created in step one.



You can also just type the name of your parameter into the edit box directly, but make sure that you spell the parameter name correctly and observe Lowercase/Uppercase Letters. Click on the green tick at the end of the edit box when you're done.

When your sketch is fully constrained, Click on the big green tick on the 'Exit' panel of the 'Sketch' Tab to complete the sketch and return to the part modelling environment.



Modelling a Shooting Board

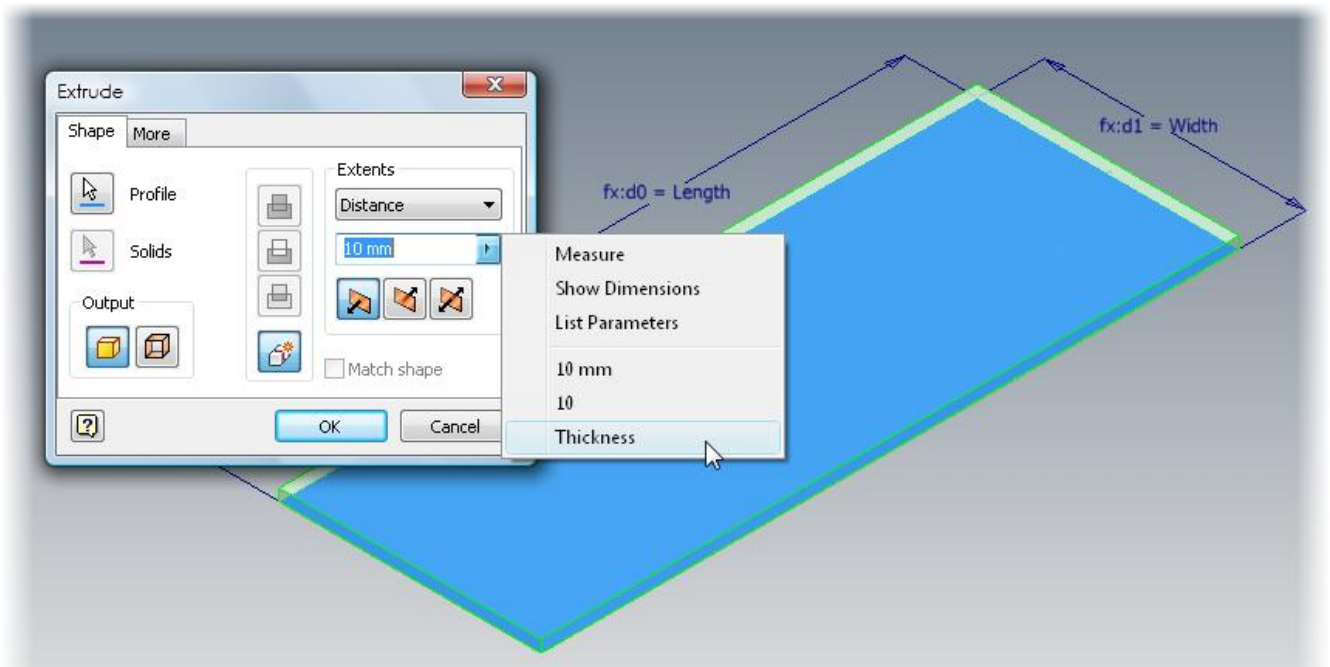
Creating a Solid, or 'Sketch based feature'

Use the 'Extrude' tool to give your sketch a thickness.

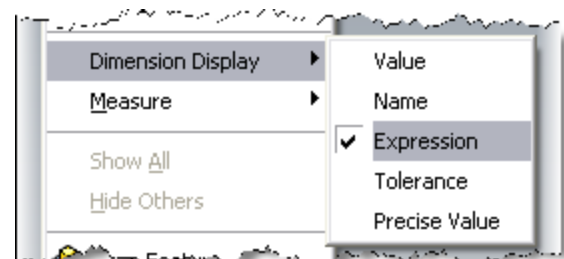
Model Tab > Create panel > Extrude Tool



Once again use the fly out arrow at the side of the edit box to pick a parameter from your list of parameters. Or you could type 'Thickness' into the edit box directly.

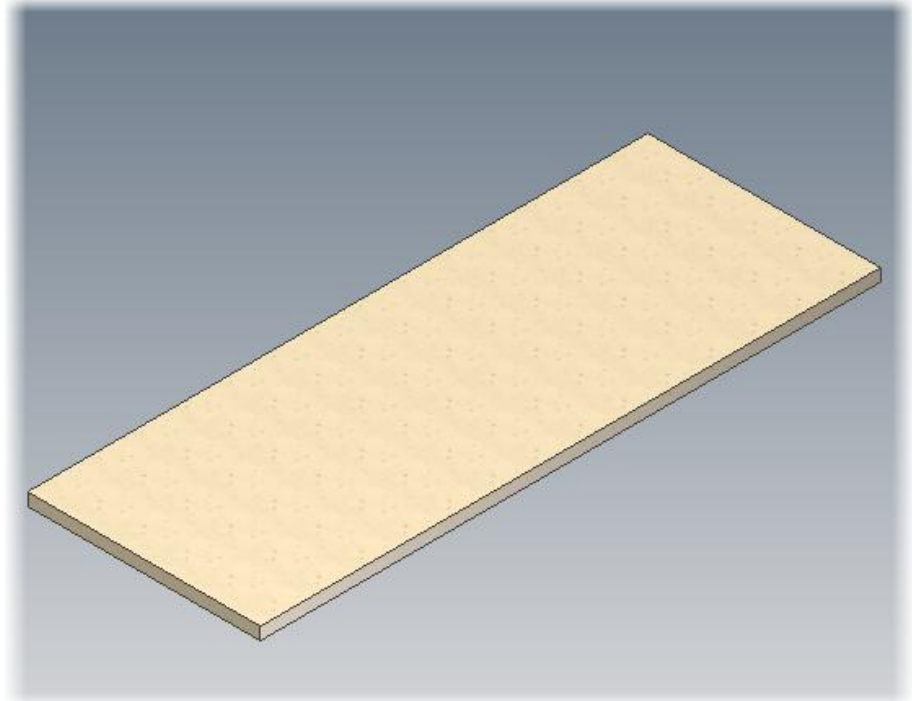
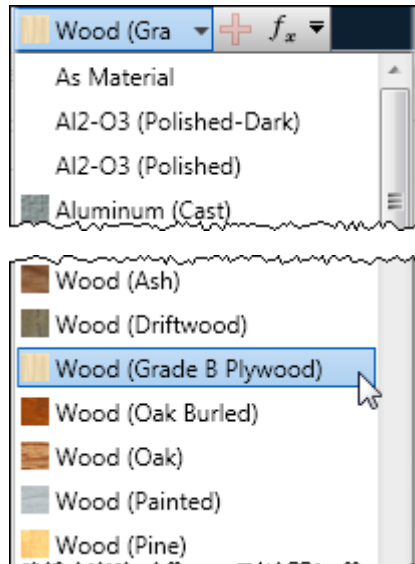


If Your Dimension parameters don't show the parameter Expression, as shown in the illustration above , make sure that you don't have anything selected and right click anywhere in the drawing area to bring up the Dimension display options.

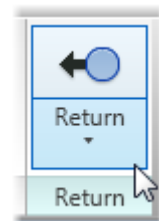


Modelling a Shooting Board

You can use the Colour override drop down to allocate a different look to your part.



When you are happy with your part hit 'Return' to return to the Assembly environment. Save your Assembly as 'Shooting Board Assembly .iam'. You have successfully created your assembly – and your first part. That was easy!



Assembly constraints

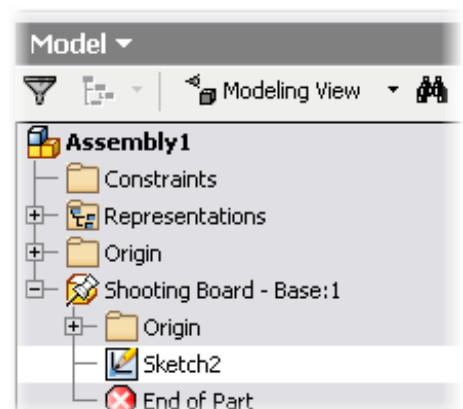
You may notice that the first part that you insert in any assembly will automatically be 'Grounded' – This is indicated by the Push Pin Icon on the part node in the browser.



Every part that you insert into an assembly has Six degrees of freedom. Forwards and Backwards – Left and Right – Up and Down, and your part can rotate about its X, Y, and Z Axis.

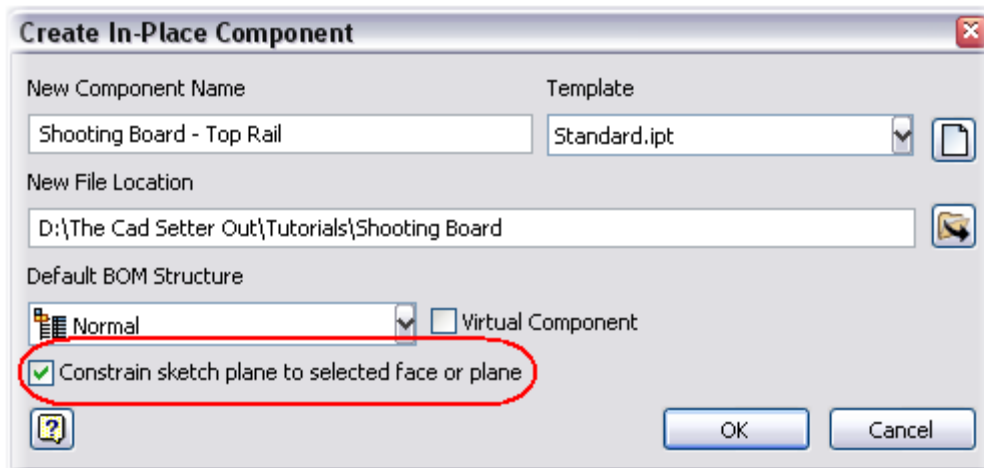
You can use Assembly constraints to lock down the position of your parts. Grounding the first part makes it easier to ensure that you have limited the freedom of your parts.

Using the 'In place' modelling technique each part you create will automatically be constrained to the surface or plane that you picked as a base feature. Usually you would need to further constrain the part to fully lock down its position. Using adaptivity we will link the parts together instead.

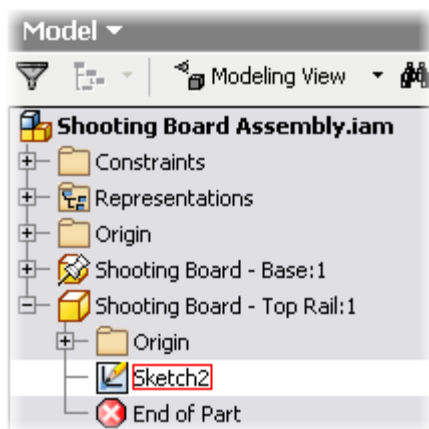


Modelling a Shooting Board**Adding the next part**

Once again use the 'Create' tool to create a new part, called 'Shooting Block – Top Rail'. Notice that you have a new option. Make sure that 'Constrain sketch plane to selected face or plane' is checked.



This time use the top face of your first part (the Base board) as your sketch plane. Your part will be automatically created with a new sketch open.



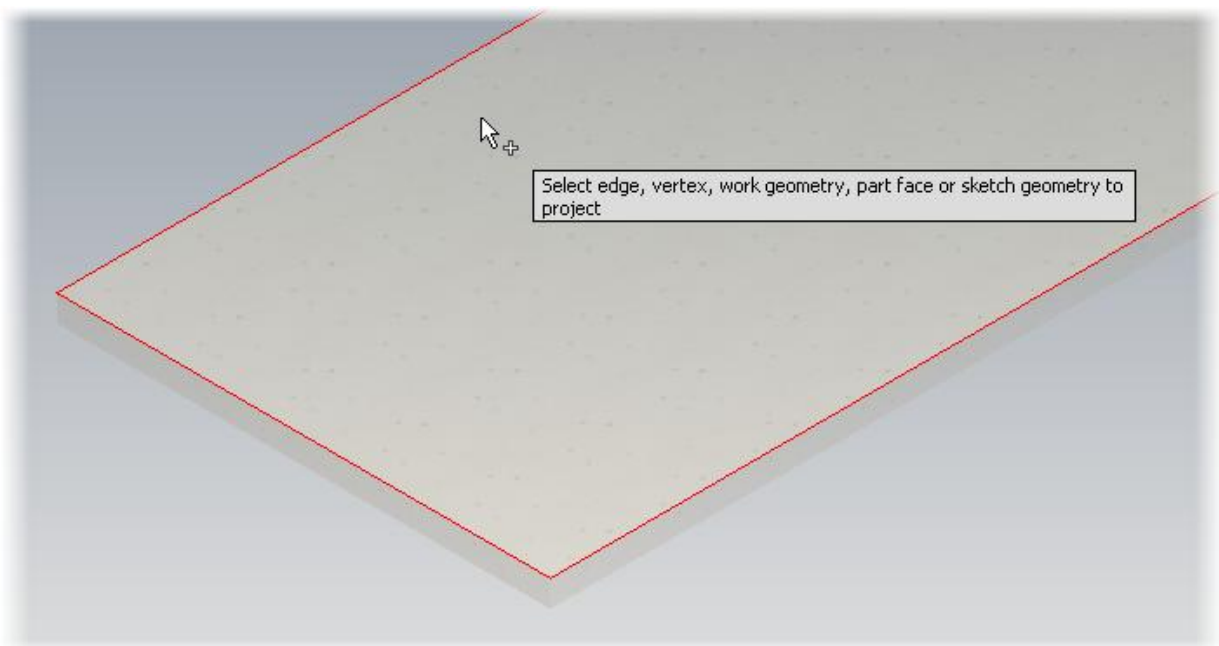
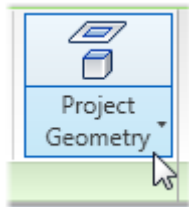
Add new parameters called 'Length', 'Width' and 'Thickness'. Note that the parameter 'Length' does not need a value at this stage. Later on we will link the Length of the part to the Base board.

Name	Value
Length	-
Width	125
Thickness	18

Modelling a Shooting Board

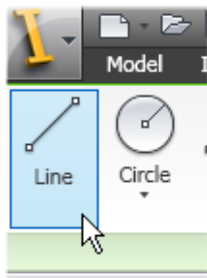
Use the 'Project Geometry tool to project the edges of your base board into your Top rail sketch.

Sketch Tab > Draw Panel > Project Geometry

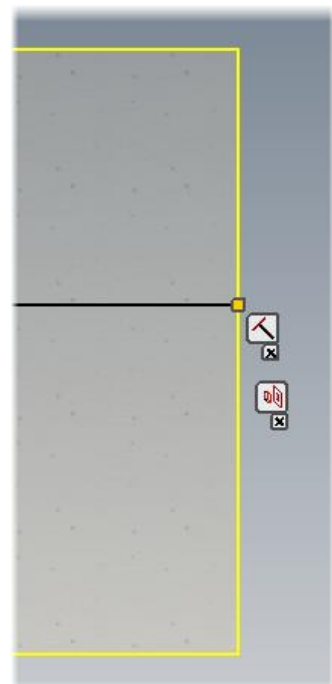


Use the line tool to complete the rectangle.

Sketch Tab > Draw Panel > Line Tool



Right click anywhere in the drawing area and choose 'Show all constraints' to make sure that your line is constrained to the projected edges.



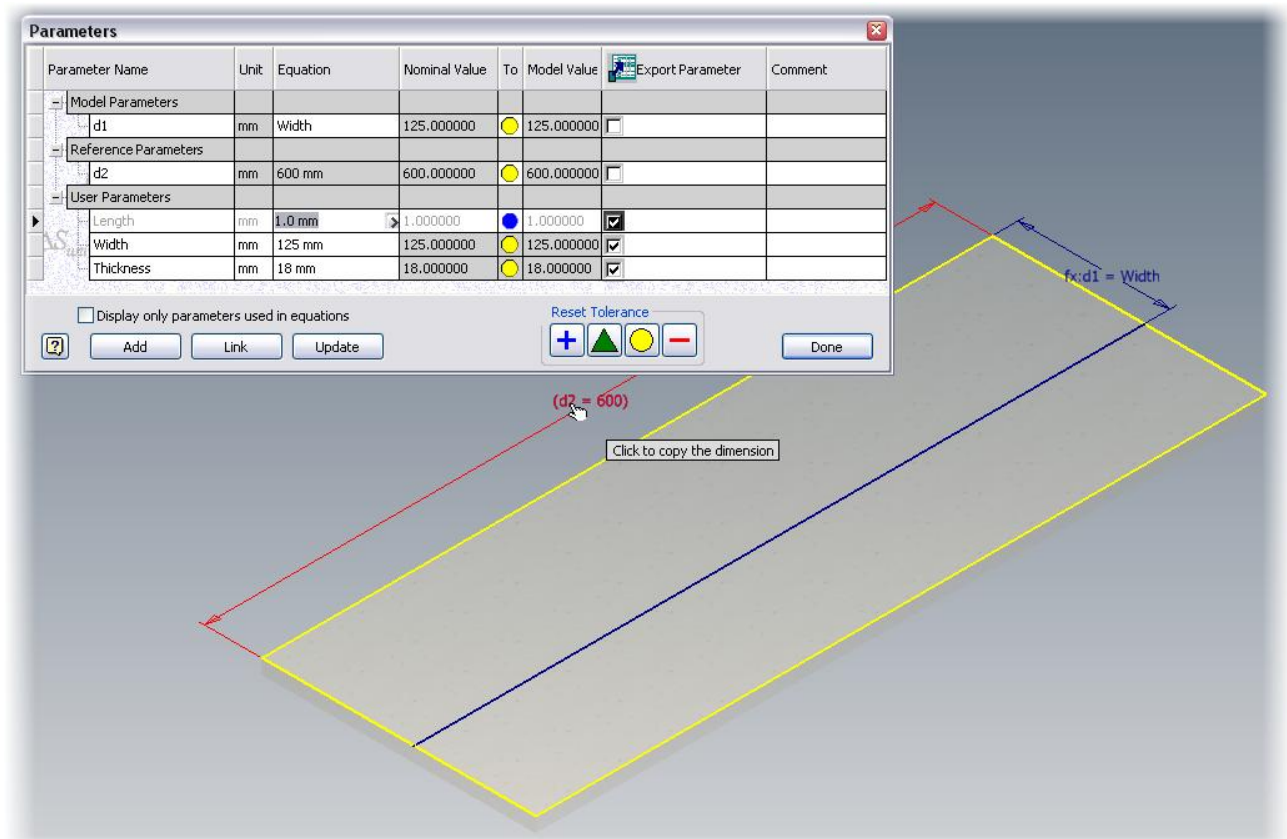
Modelling a Shooting Board

Add dimension Constraints

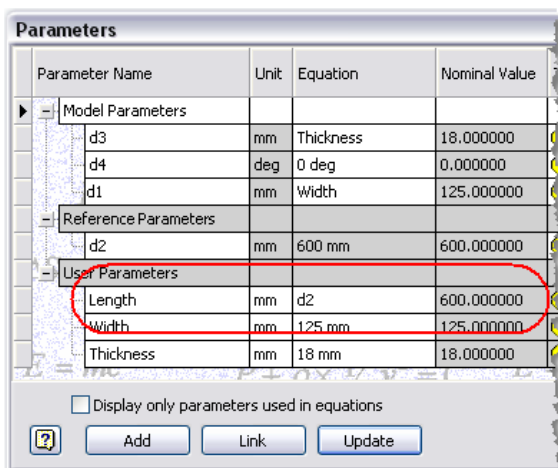
Add the width dimension constraint as we did before. Add a dimension constraint along the length. You will notice that Inventor automatically makes this a 'Reference' constraint. The value of the constraint is shown in brackets and instead of including a value that you can change the constraint is simply telling you what its value is.

—(d2 = 600)—

Open the parameters dialog and note that a new reference parameter has been added. It is called 'd2' and its value is 600 – just as we saw on screen. We want to get some feedback on the length of the board, so we are going to 'Map' the Length constraint we created to this value. With the parameters dialog open, either type 'd2' into the 'Equation' column of the 'Length' parameter, or simply click inside this cell to highlight it and then click on the dimension constraint in the drawing window to copy (Link) its value.

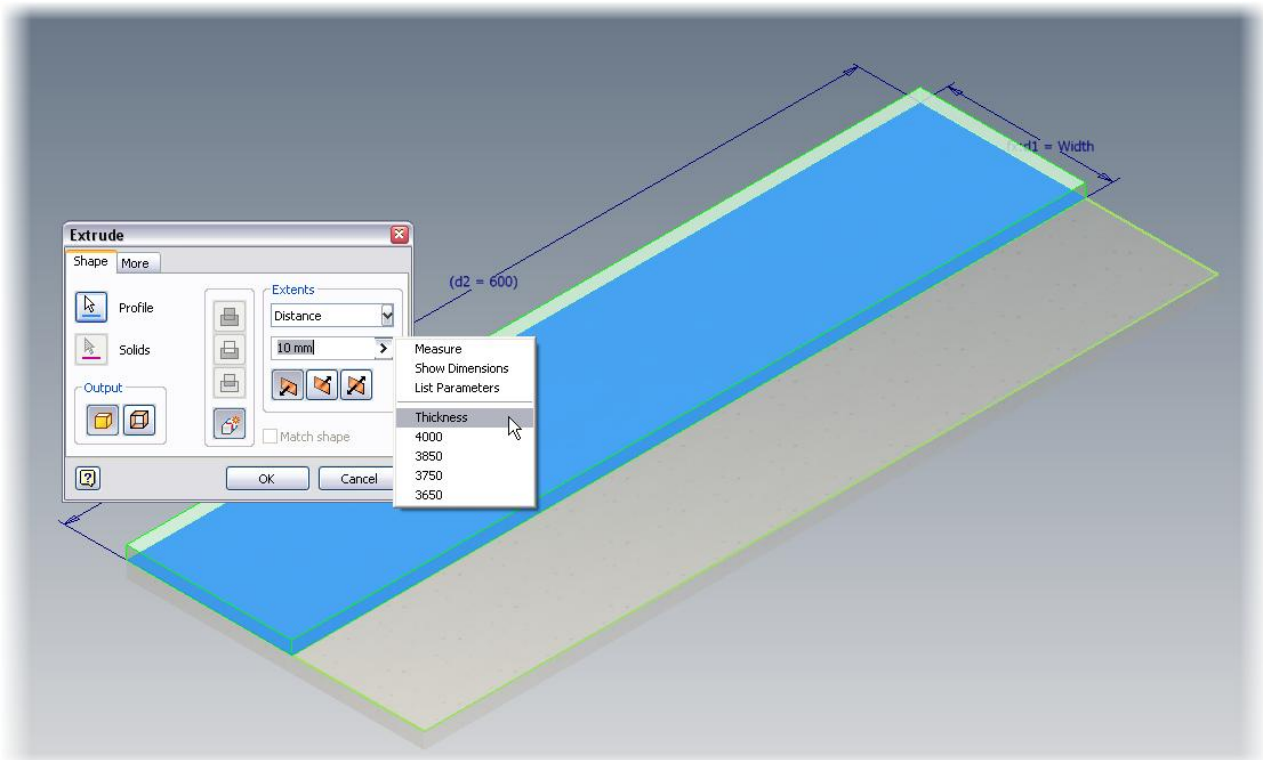


The Length parameter now has a value of 600.

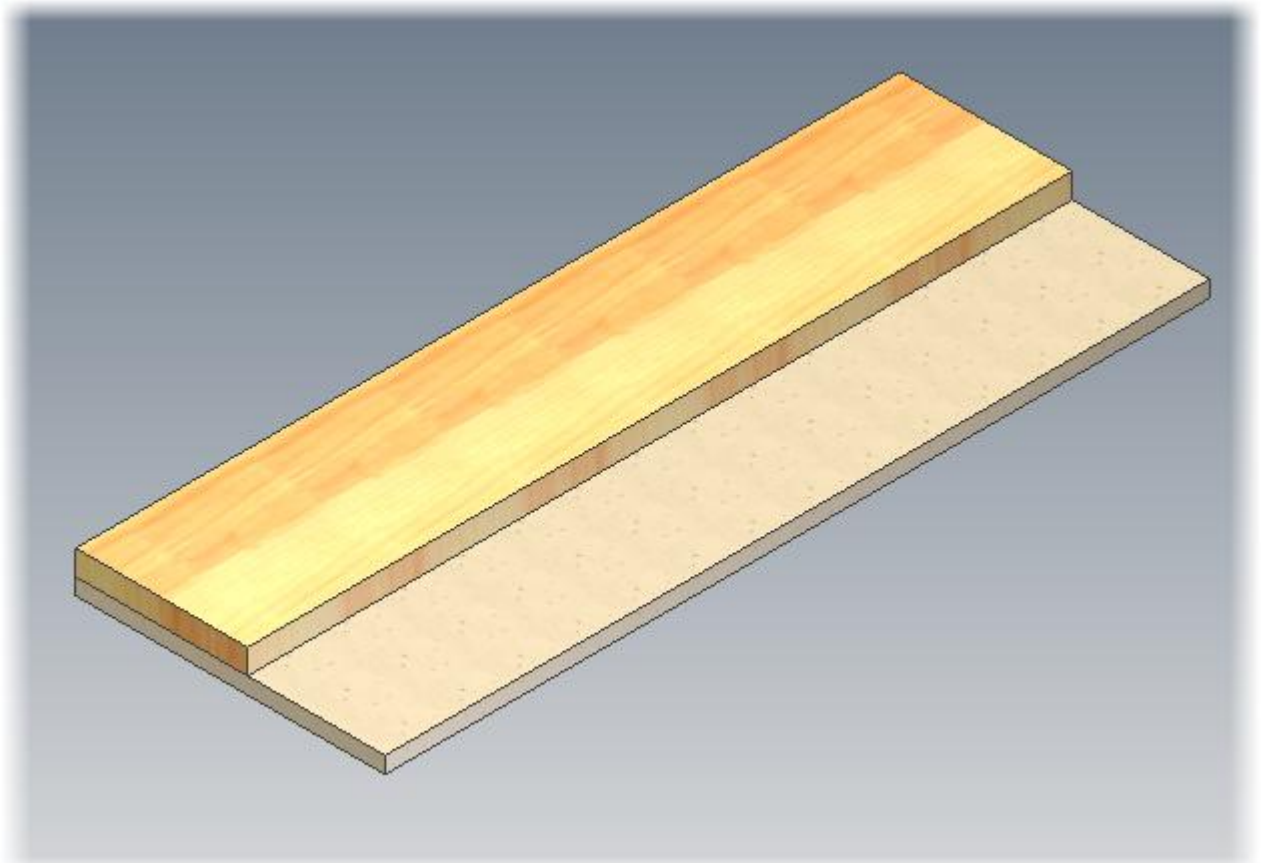


Modelling a Shooting Board

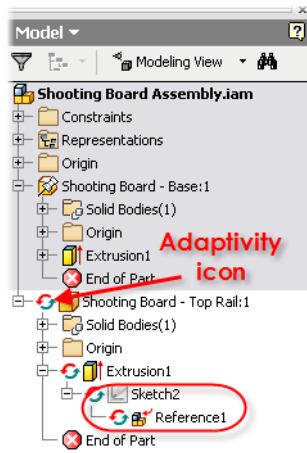
Finally, use the extrude tool to create our feature, using the 'Thickness' parameter we created earlier.



And change the colour of the part if you wish...



Modelling a Shooting Board



You may notice that your Assembly browser contains a new icon, which looks like a blue and red pair of rotating arrows. This is the 'Adaptivity' icon, indicating that this part is linked to another.

Underneath the sketch node you will see an icon marked 'Reference'. This indicates that your sketch contains a reference, or Link, to another part – in this case the base board.

Note that the reference, sketch, Feature and part all have the Adaptive icon next to them.

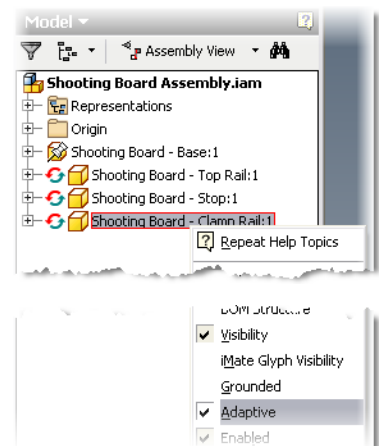
Testing your adaptive part

Double click on the 'Shooting Board – Base' Part node in the browser to enter edit mode for that part. Open the parameters dialog and experiment with changing the 'Length', 'Width' and 'Thickness' parameters that we created earlier on. After each change, return to the assembly environment to see the parts update.

Note that the 'Shooting board – Top rail' is linked to the 'Shooting Board – Base Board' and that both parts operate as one. That's Magic!

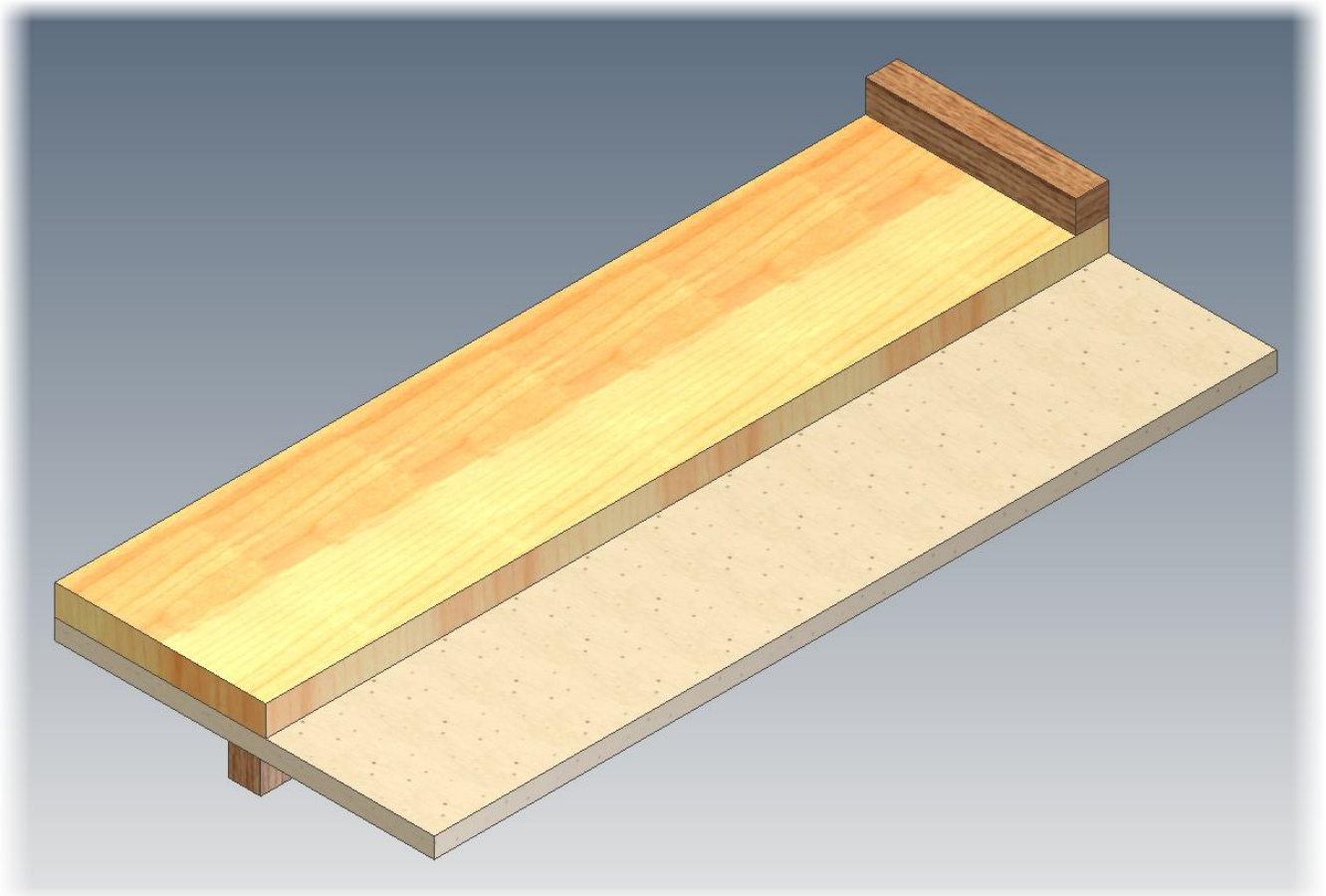
Adaptivity is a really cool way of creating small assemblies that automatically update based on the size of a master part. But adaptivity does have its limits and there are more robust techniques to use if you have a larger assembly to build.

- *Bear in mind that only one occurrence of a part defines its adaptive features. If you use multiple placements of the same part in an assembly, only one occurrence can be adaptive, but all occurrences reflect changes to it, including placements in other assemblies.*
- *An adaptive part can only be adaptive in one assembly at a time, so adaptivity is not good for library parts.*
- *Adaptive parts can take Inventor a while to calculate. Once you have a part how you want it, right click and uncheck 'Adaptive' to save calculation time. Remember to turn adaptivity back on if you need to Inventor to re-calculate your dimensions.*
- *Avoid linking adaptive parts to more than one other part, this can lead to 'Circular' references which Inventor can't calculate.*



Finishing off...

Create the stop using the top of the Top rail as a base face and the Clamp rail using the underside of the Base board as a base face.



That's your shooting board model completed! I hope that you are pleased with your results. I hope that you now have an understanding of how to create new parts whilst inside an assembly and how to link parts using 'Adaptivity' to create a simple assembly model, in which the assembly is controlled by a master part.

This isn't the only technique for producing assembly models in Inventor. We could also have used the 'Top Down' technique, or a 'Skeletal Model', but that's next week...

Key Concepts:

1. Create a new part 'In Place', using a work plane or surface of another part.
2. Add parameters
3. Geometrically constrain sketches
4. Dimension sketches
5. Use 'Reference' Dimension constraints to give feedback of sizes
6. Add features
7. Project geometry across parts
8. Use adaptive parts in an assembly