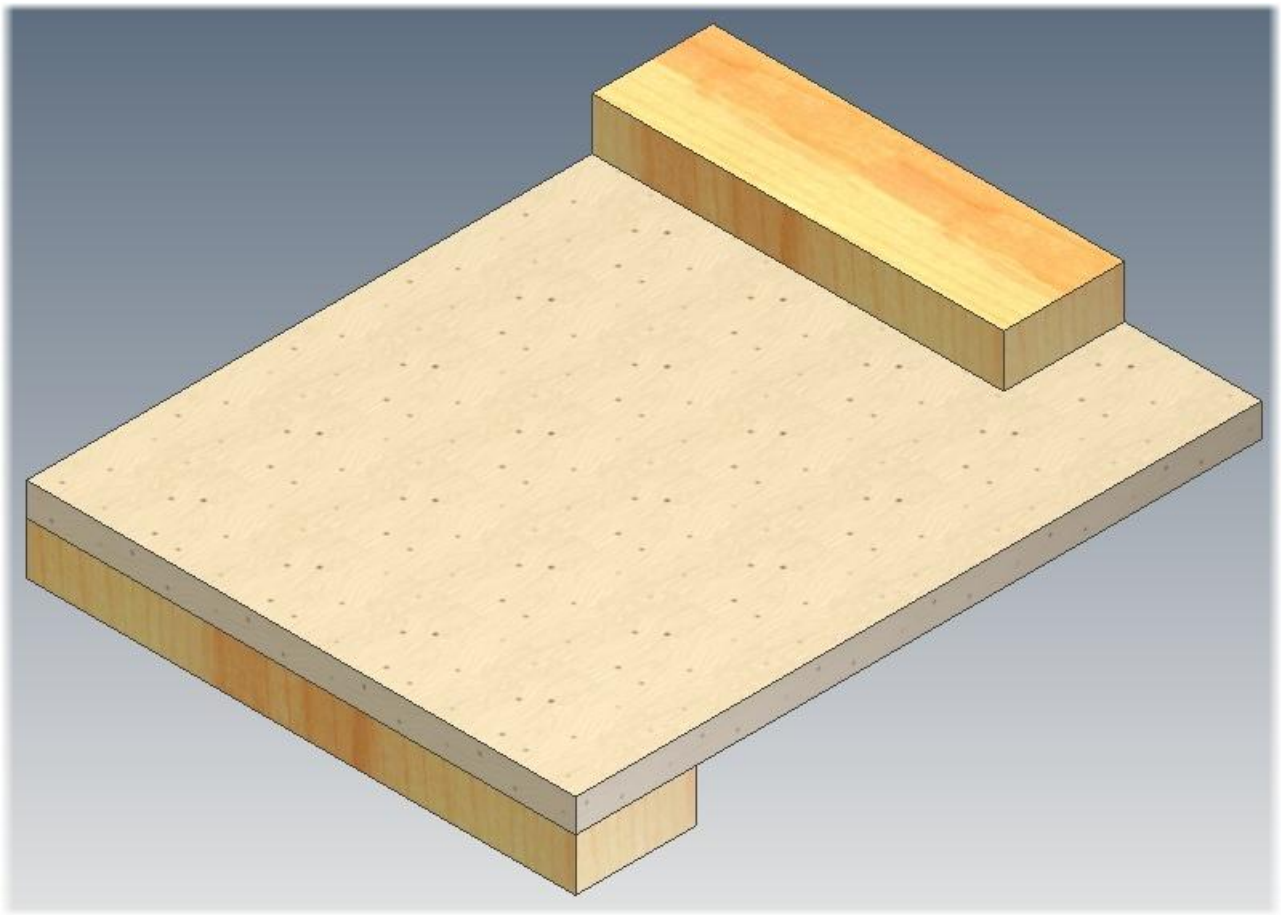
Modelling a Bench hook

Using the 'Bottom up' technique



This tutorial will show you how to create a simple assembly with Autodesk Inventor, using the 'Bottom up' technique of building an assembly model. Using the Bottom up technique we will build each individual part, and then combine the parts together into an assembly.

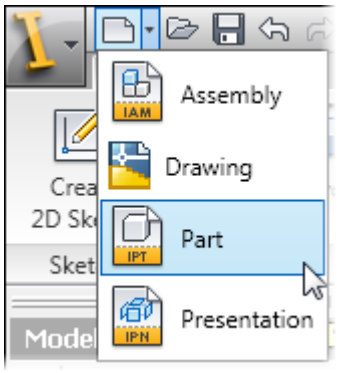
This article is aimed at novice users. However I am assuming that you are familiar with the concepts of parametric modelling, and that you've had some time to explore the Inventor user interface.

In this tutorial we will use the following workflow:

- Create a new part
- Add parameters
- Create a sketch
- Constrain the sketch
- Add a sketch based feature (an extrusion)
- Change the look of a part
- Combine the parts into an assembly
- Constrain the parts

Creating the parts

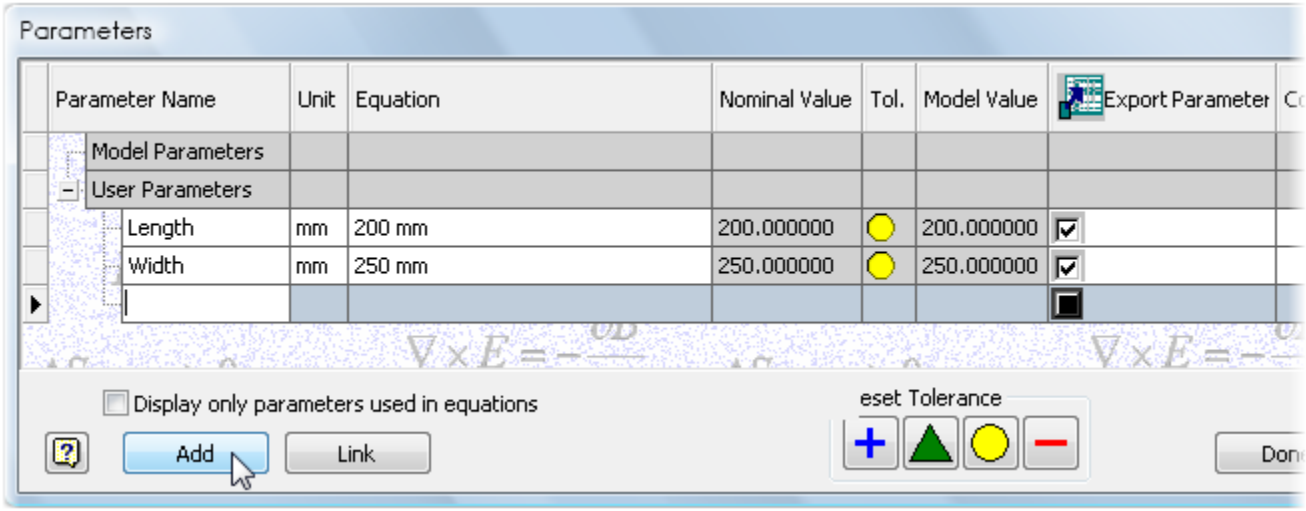
Open a new part file, and create the following parameters:



Manage Tab> Parameters Panel> Parameters tool

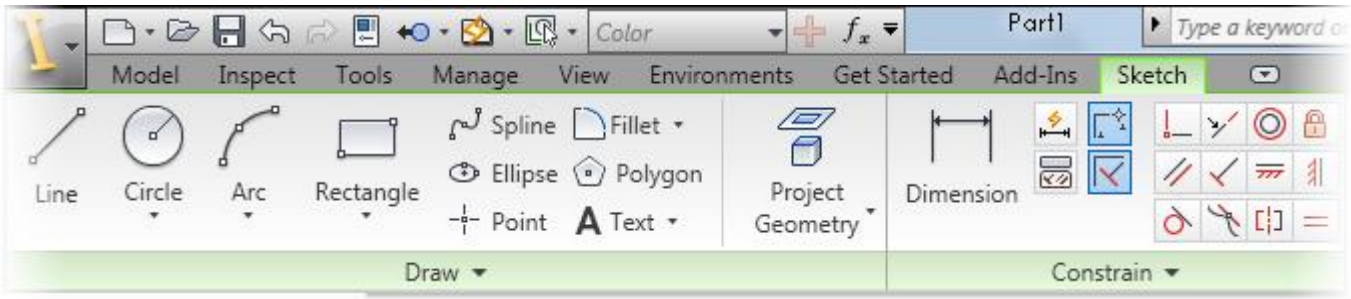
Name	Value
Length	200
Width	250
Thickness	12

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



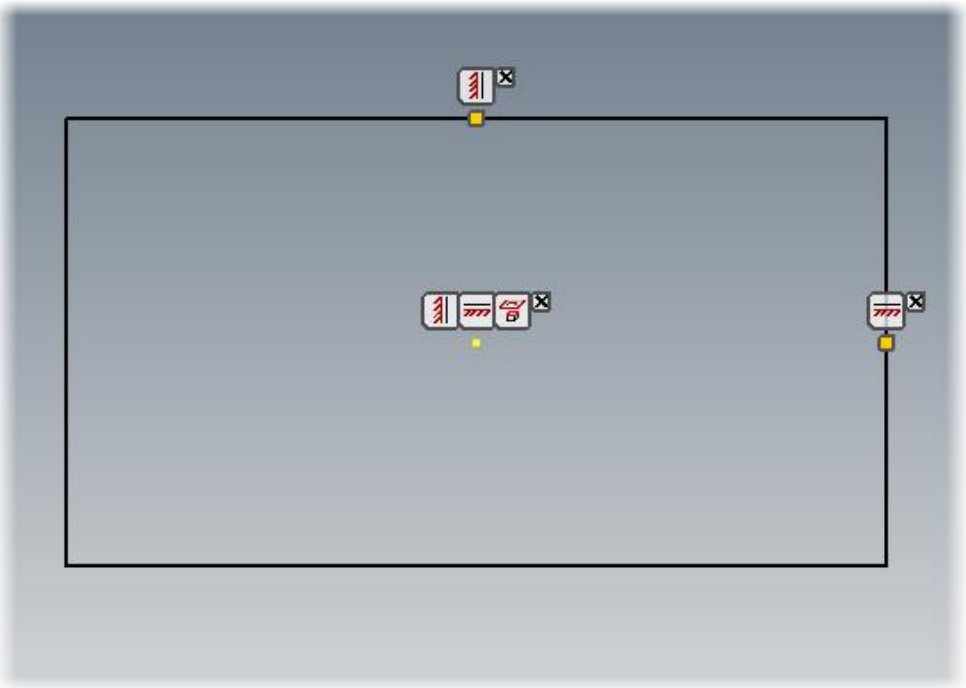
Don't forget that Your parameter names cannot contain spaces, mathematical symbols, or special characters. Parameter names must start with a Letter and they are case sensitive

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.



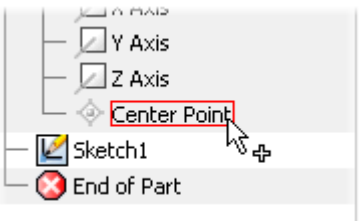
Sketch out a rectangle

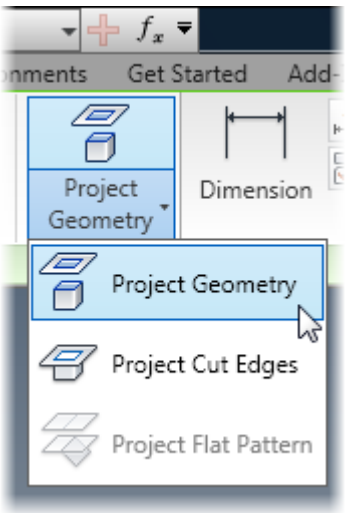
Sketch Tab>Draw Panel>Rectangle Tool



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

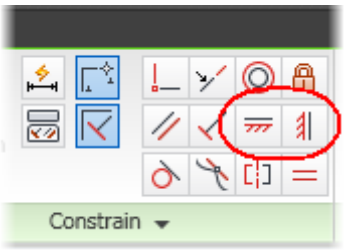
Sketch Tab > Draw Panel > Project geometry Tool





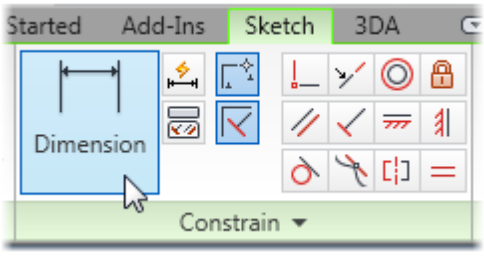
Use the Horizontal and Vertical constraint tools to centre your rectangle about the origin.

Sketch Tab>Constrain Panel>Vertical and Horizontal Constraint Tools

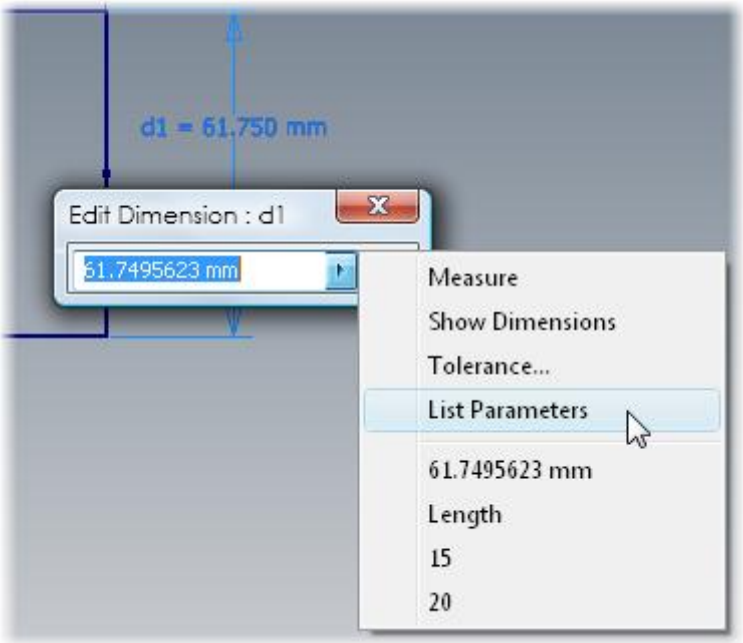


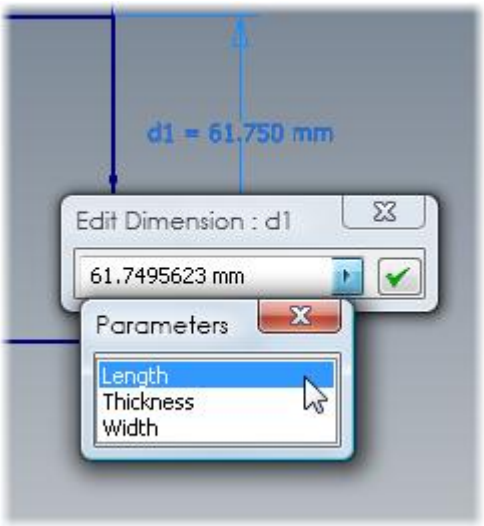
Add dimensional constraints to your sketch

Sketch Tab >Constrain Panel >Dimension Tool



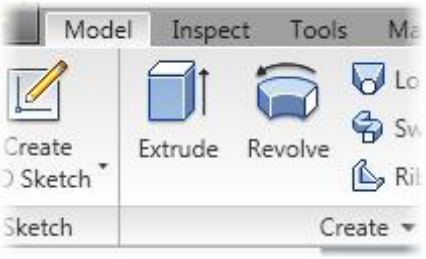
Single click on a dimension constraint to edit its value whilst using the dimension tool, or double click on a dimension constraint to edit its value at any other time. Click on the arrow at the end of the edit box and choose 'List parameters' to choose from the list of parameters that you created in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





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.

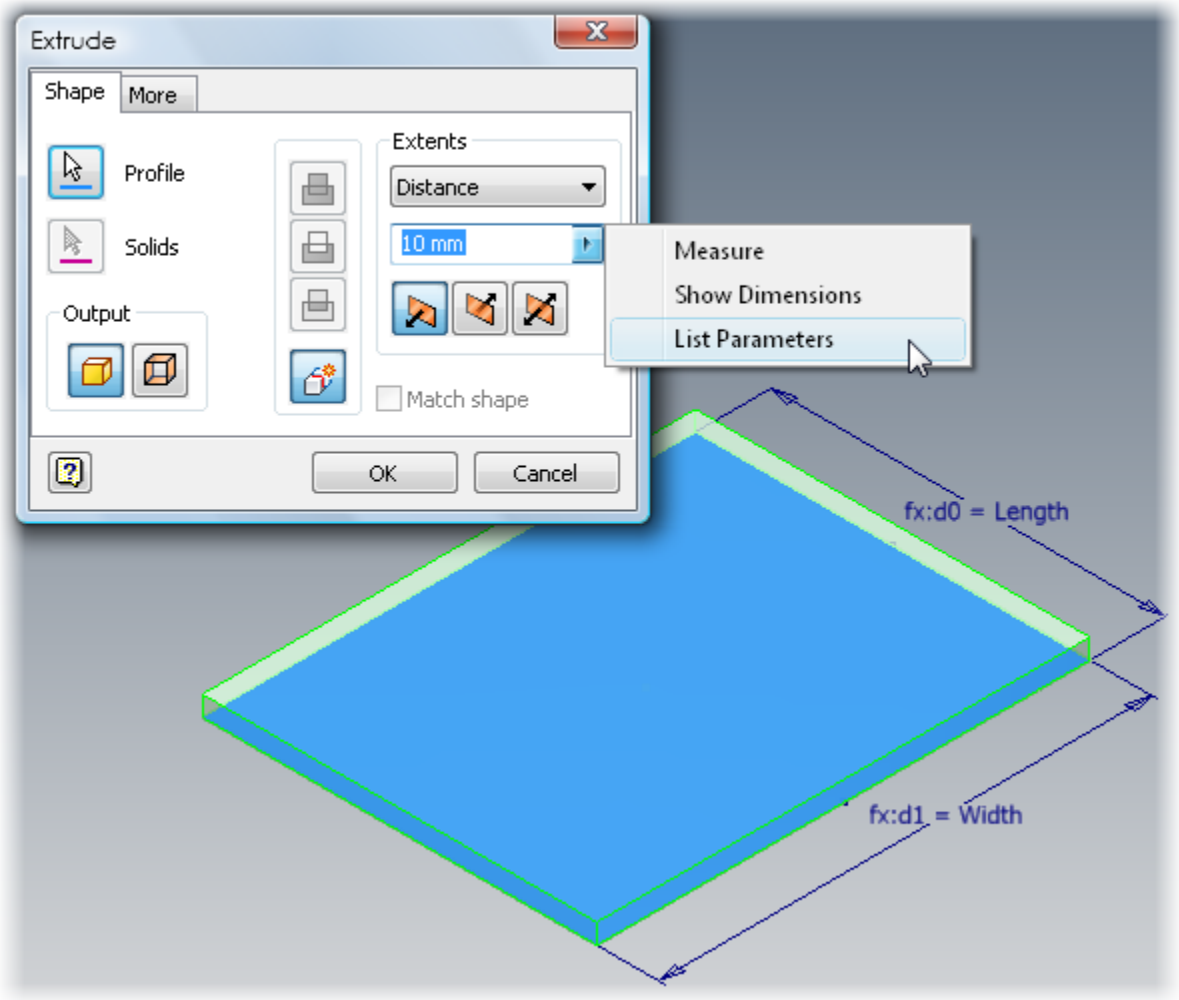
When your sketch is fully constrained, Click on the big green tick on the 'Exit' panel of the 'Sketch' Tab to complete the sketch and return to the part modelling environment.



Creating a Solid, or 'Sketch based feature'

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

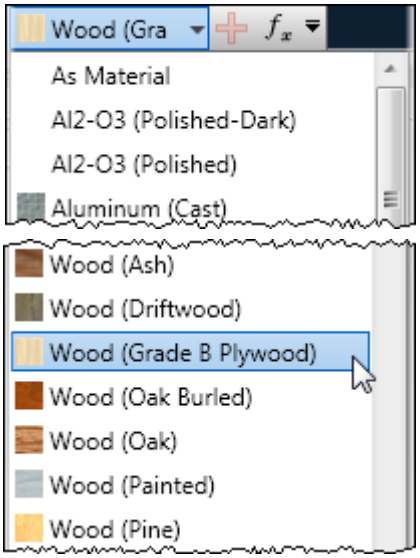
Model Tab > Create panel >Extrude Tool

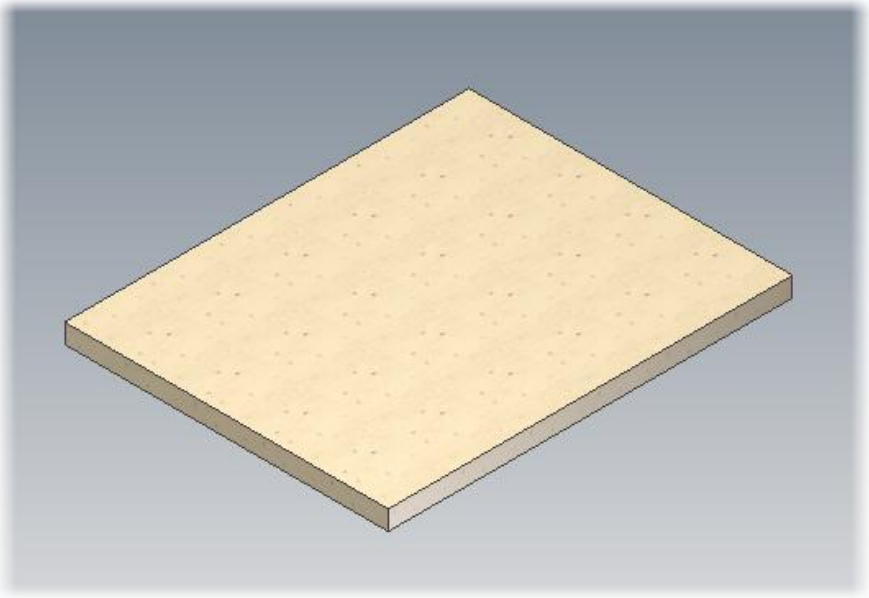


Once again use the fly out arrow at the side of the edit box to pick a parameter from your list of parameters. Or you could type 'Thickness' into the edit box directly.

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 any where in the drawing area to bring up the Dimension display options.

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



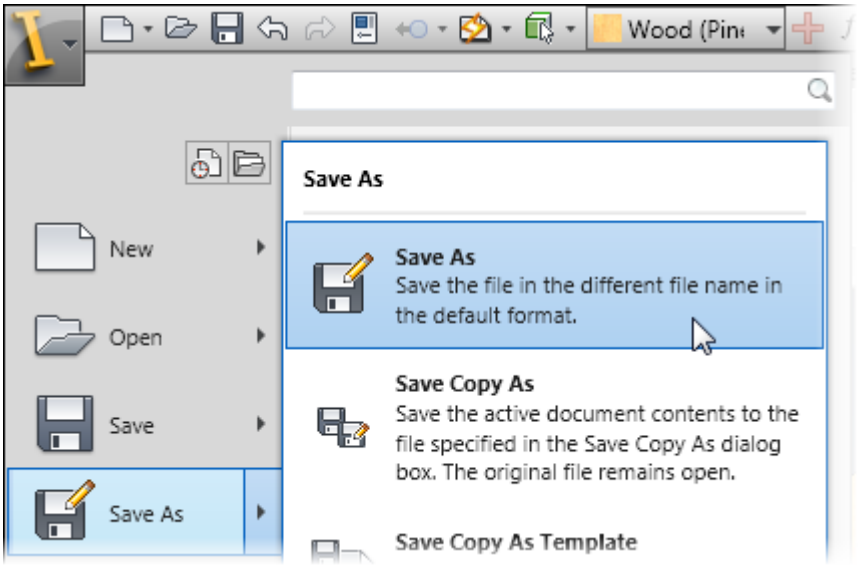


When you are happy with your part, save it as 'Bench Hook – Base Board'. You have successfully created your first part. That was easy!

Now build the remaining parts based on the information in the table below.

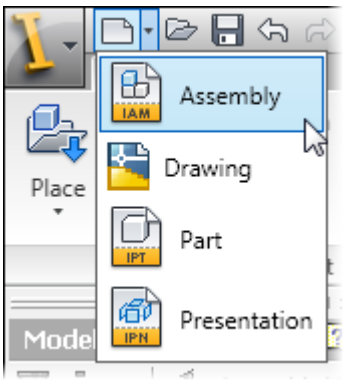
Part Name	Length	Width	Thickness	Colour
Bench Hook – Base Board	200	250	12	Wood (Grade B Ply)
Bench Hook – Stop Rail	150	44	19	Wood (Pine)
Bench Hook – Hook Rail	200	44	19	Wood (Pine)

To create these remaining parts quickly, choose 'Save as' from the Big 'I' menu and use the parameters manager to change the length width and thickness of each part.



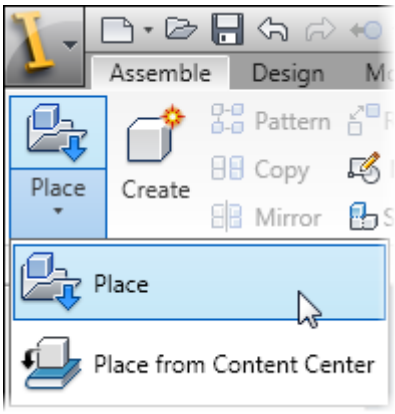
Building The Assembly

Open a new assembly file:



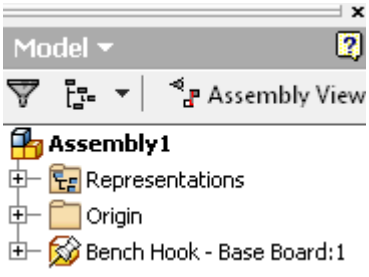
Insert the 'Bench Hook – Base Board.ipt'

Assemble Tab > Component Panel > Place Tool



Browse to the 'Bench hook – Base Board .ipt' File and click 'Open'. The first part will be placed automatically. Right click and choose 'Done', or hit the 'Escape' Key to stop placing parts.

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.

You can insert the remaining two parts now, in the same way.

Constrain your parts

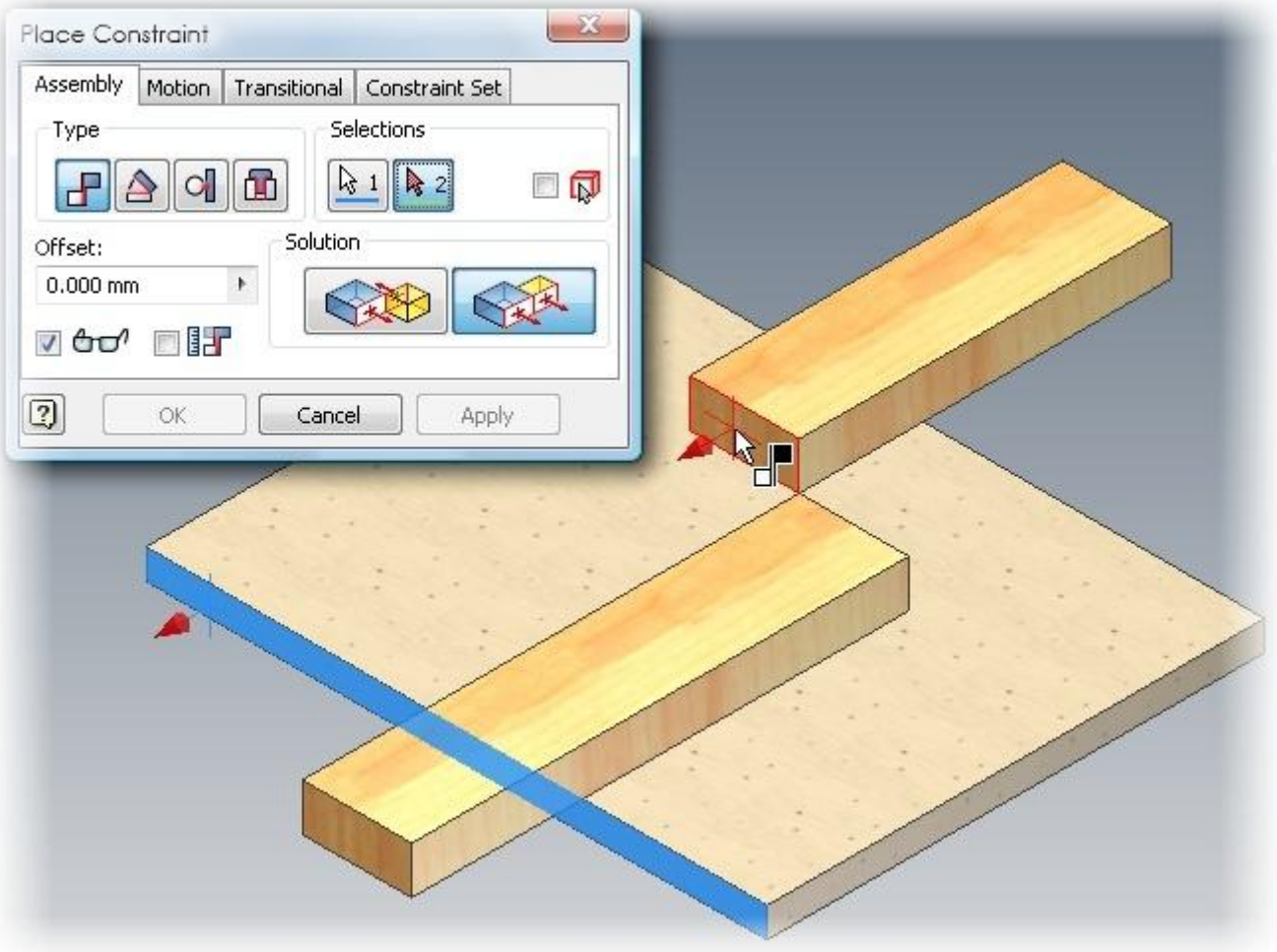
Use the View Cube to orientate the view so that you can see the side faces of the parts.

Pick the 'Constrain' tool

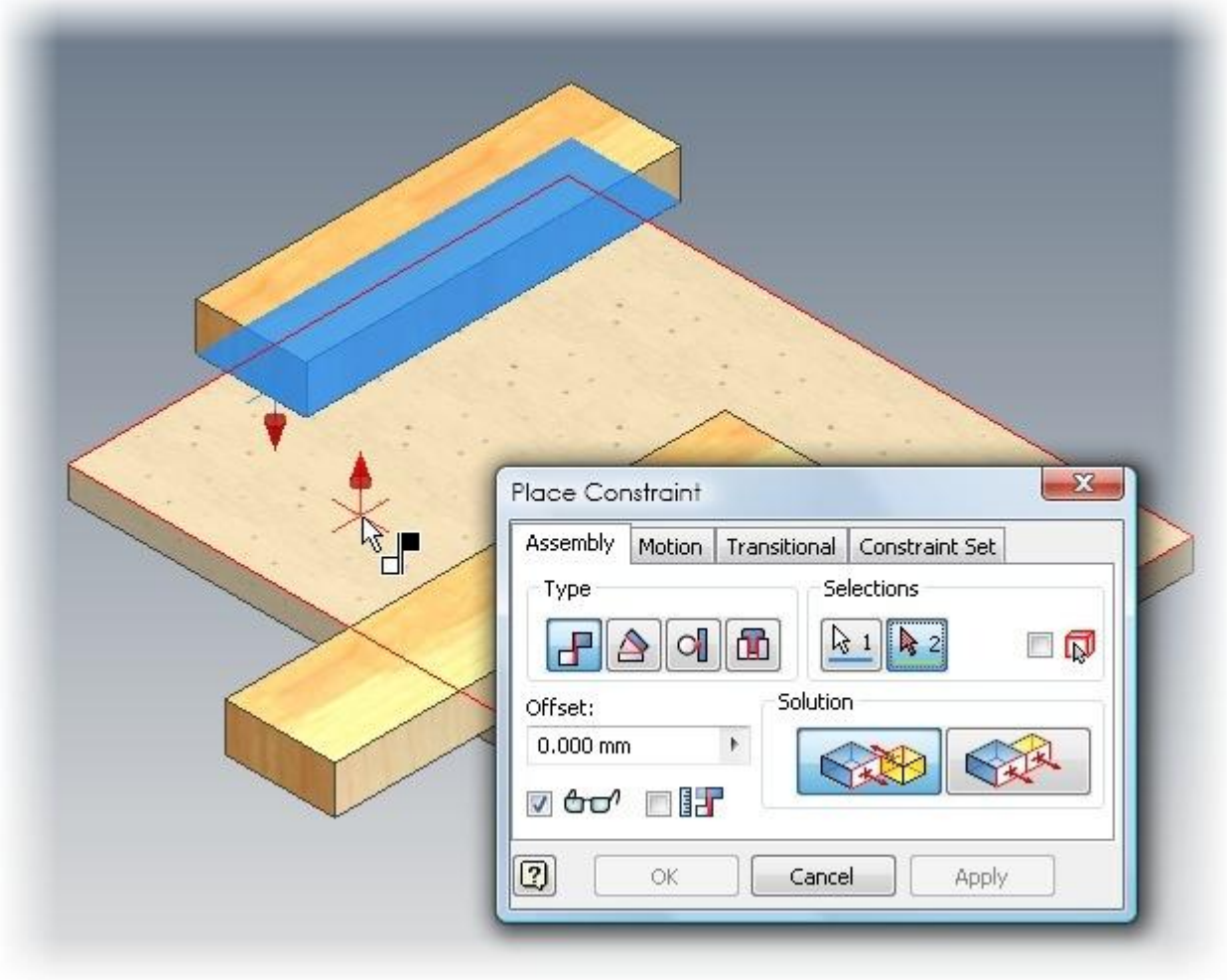
Assemble Tab > Position Panel > Constrain

Pick the 'Flush' Solution option and pick the side faces as shown to constraint the two parts to the same plane.

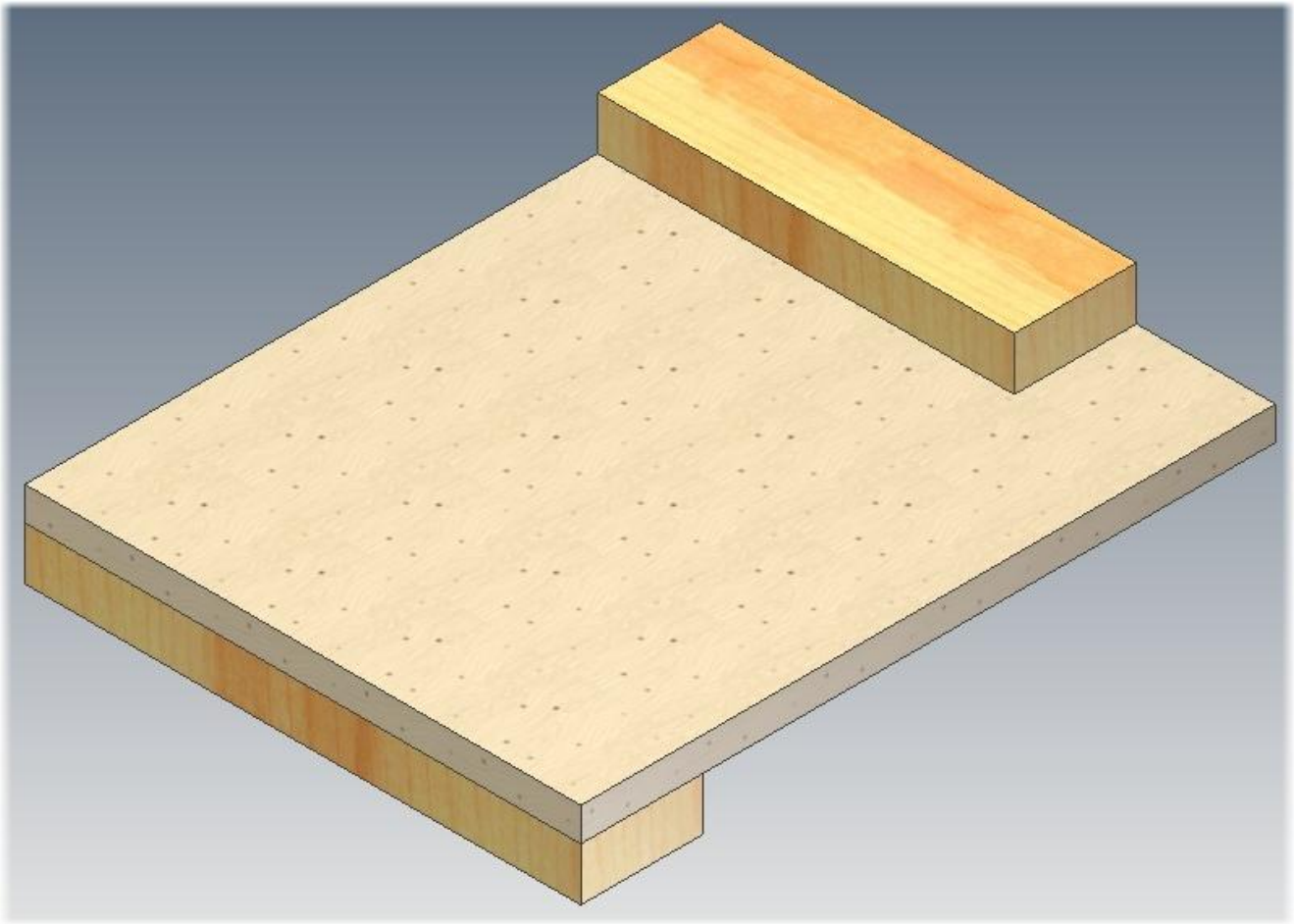
Use the 'Apply' button, rather than OK to keep the assembly constraints dialog open while you place multiple constraints



Now use the ‘Mate’ Solution option to stick the underside of the top rail to the top of the base board.



Continue placing constraints until your assembly is fully constrained.



That's your bench hook model completed! I hope that you are pleased with your results. I hope that you can now model simple parts and combine your parts together to form an assembly. This isn't the only technique for producing models in Inventor. We could also have used the 'In place' or 'Top Down' techniques, or a 'Skeletal Model', but that's next week…

Key Concepts:

1. Add Parameters
2. Geometrically constrain sketch
3. Dimension sketch
4. Add features
5. Use 'Save as' to quickly build similar parts
6. Use Assembly constraints to limit degrees of freedom.