

Modelling a Saw Horse

Using the 'Top Down' technique



This tutorial will show you how to create a simple assembly model with Autodesk Inventor, using the 'Top Down' or 'Multi body Master Part' modelling technique. We will start with a master part file which contains multiple solids which represent all the major parts needed for our assembly. We will then use Autodesk Inventor's new 'Make components' tool to 'Derive' our master solids out into our part files and create our assembly model. Using this method there is no need to add assembly constraints to lock the position of the parts, all parts will automatically be grounded using their origin coordinates. The master part file will control the size and layout of the assembly.

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

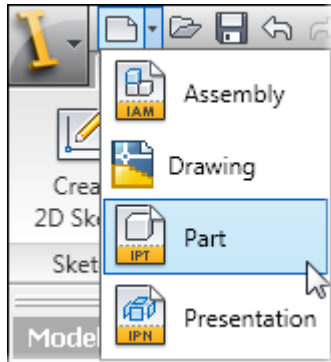
In this tutorial we will use the following workflow:

- Create a part file
- Add parameters
- Add work features
- Create a sketch
- Constrain the sketch
- Create a sketch based feature (a Solid)
- Use the 'Make components tool to build our parts and assembly.

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Creating the Master part

Open a new part file.



Save your new part as 'Saw Horse Master Part' and create the following parameters.

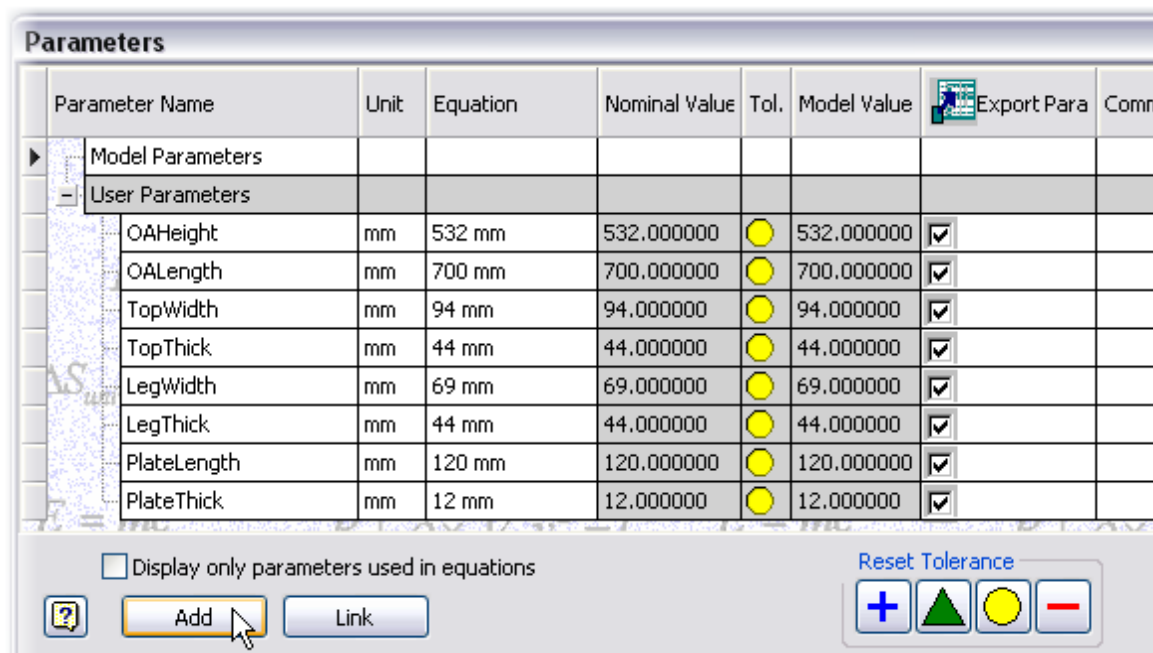
Add Parameters

Manage Tab> Parameters Panel> Parameters tool



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

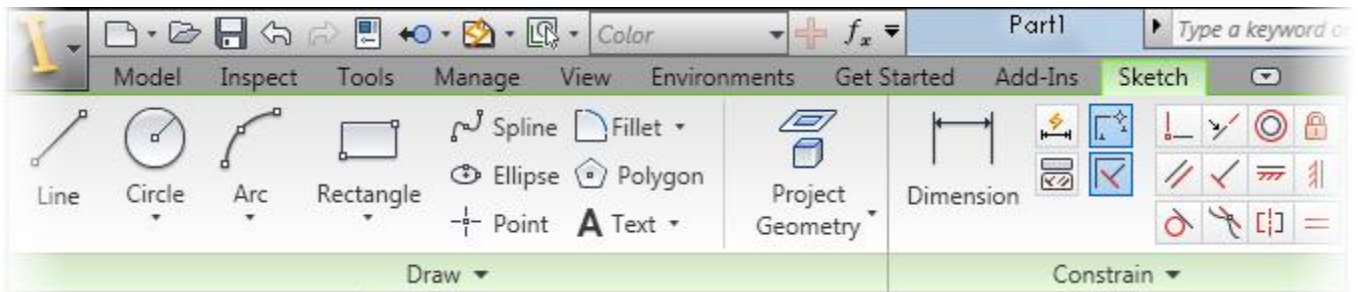
Use the 'Add' Button or ALT+A to add parameters.



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Create a your Work Planes

You may notice that if 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.



Click the 'Done' button when you have finished adding parameters and, if you need to, click on the big green tick to come out of the sketch environment and return to the part environment.

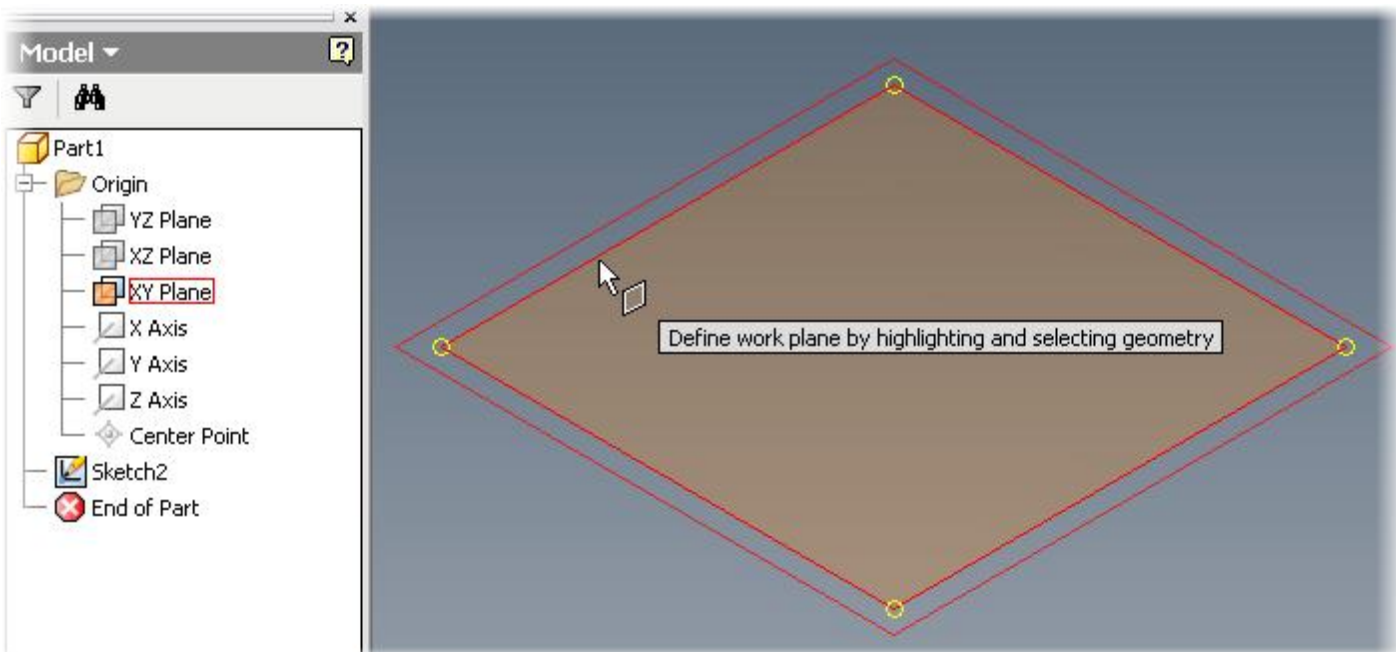
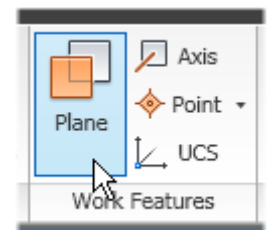


Use the 'Work plane' command to add a new work plane for the top of the Saw Horse.

Model Tab > Work Features Panel > Plane

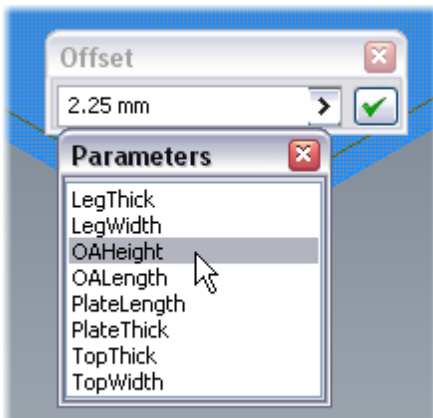
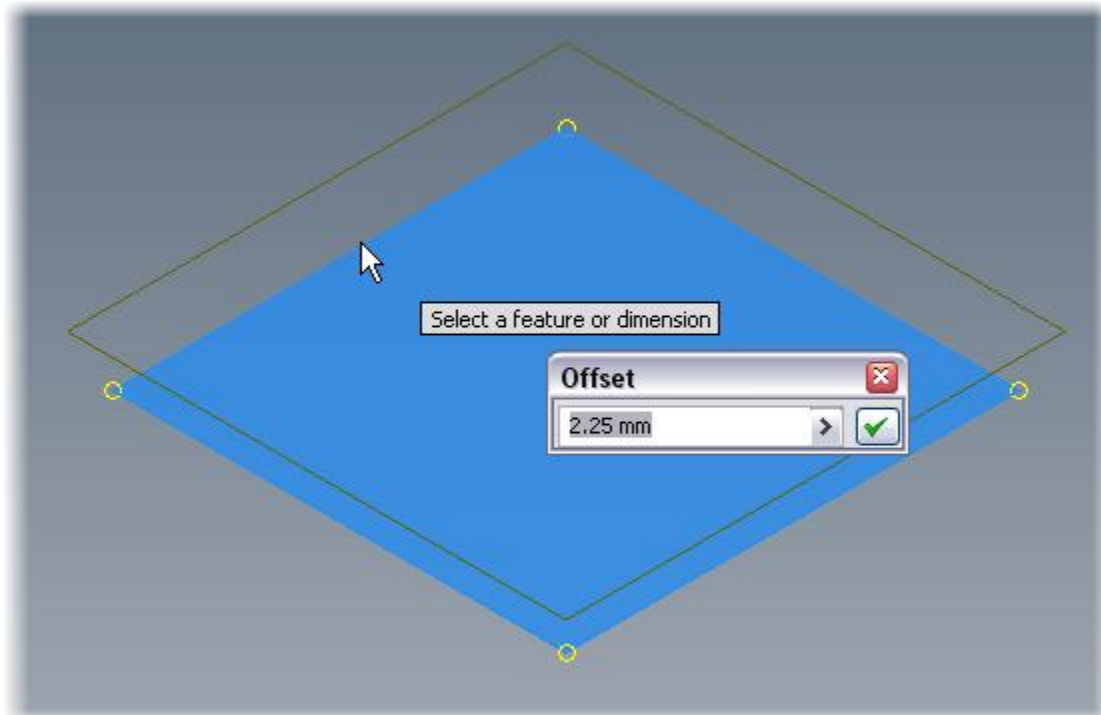
In this illustration I have made the XY Plane visible to make the process clearer, but this is not a necessary step.

Click on the XY plane to select it and keep the left hand mouse button held down. Now drag your mouse away from the XY plane to create a new offset work plane.



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Once you have started dragging you can let go of the left hand mouse button. The parameter input box will pop up prompting you for a value for 'Offset'.

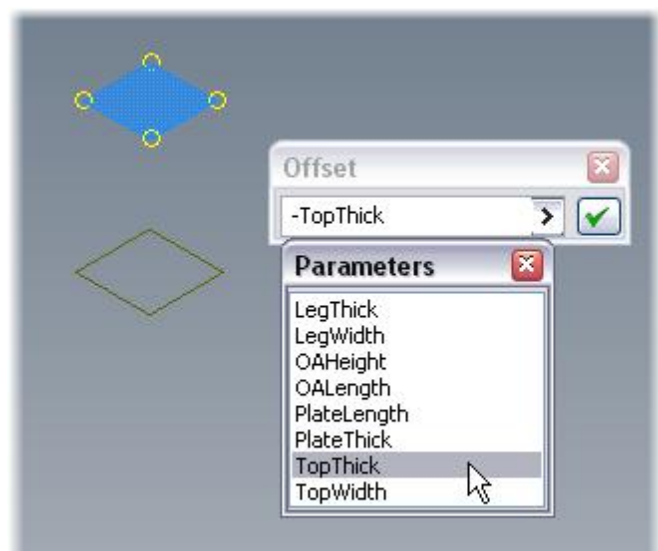


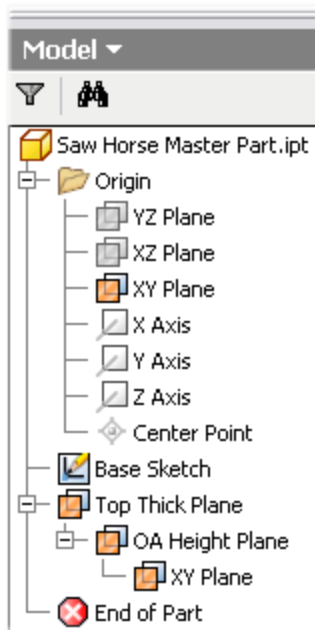
Click on the drop down arrow at the end and choose 'List parameters'. Pick the 'OAHeight' Value we created earlier. Click on the green tick at the end of the input box when you're done.

You could type this parameter value in directly, but don't forget to spell it correctly and observe Lowercase/Uppercase letters.

Click on the work plane once to select it. Click again to rename it. Rename this work Plane 'OA Height Plane'. Now create a second work plane using 'OA Height' as a starting plane, and setting the Offset value to minus 'TopThick'.

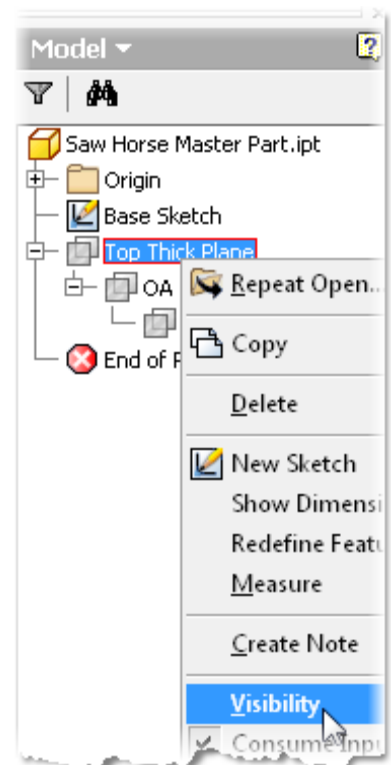
Don't forget to add a '-'(minus) before 'TopThick'. We want this plane to be below our 'OA Height' Plane. Rename this work plane 'Top Thick Plane'.



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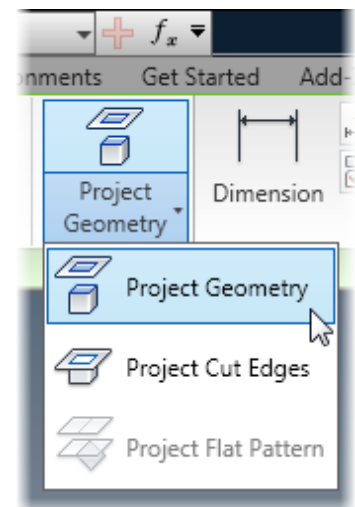
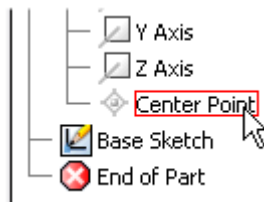
Rename the default 'Sketch1' to 'Base Sketch'. Your feature browser tree should now look something like this:

You can now turn the visibility of the Work planes off.

**Creating your Sketches**

Double click on 'Base Sketch' to activate it. If you need to, use the 'Project Geometry' tool to project the origin into the current sketch.

Sketch Tab > Draw Panel > Project geometry Tool



Set the geometry type to construction and sketch out a Rectangle.

Sketch Tab> Format Panel > Construction



Sketch Tab>Draw Panel>Rectangle Tool



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Use an Equal constraint to make the sides of the rectangle equal (i.e. make the rectangle a square)

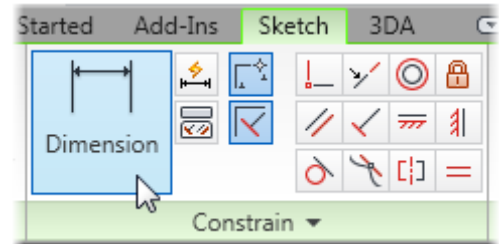
Sketch tab > Constraints Panel > Equal constraint



Now add dimension constraints to correctly size and position your sketch from the origin.

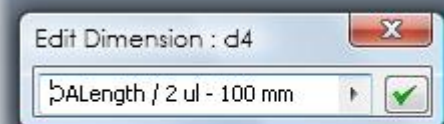
Sketch Tab > Constrain Panel > Dimension Tool

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.

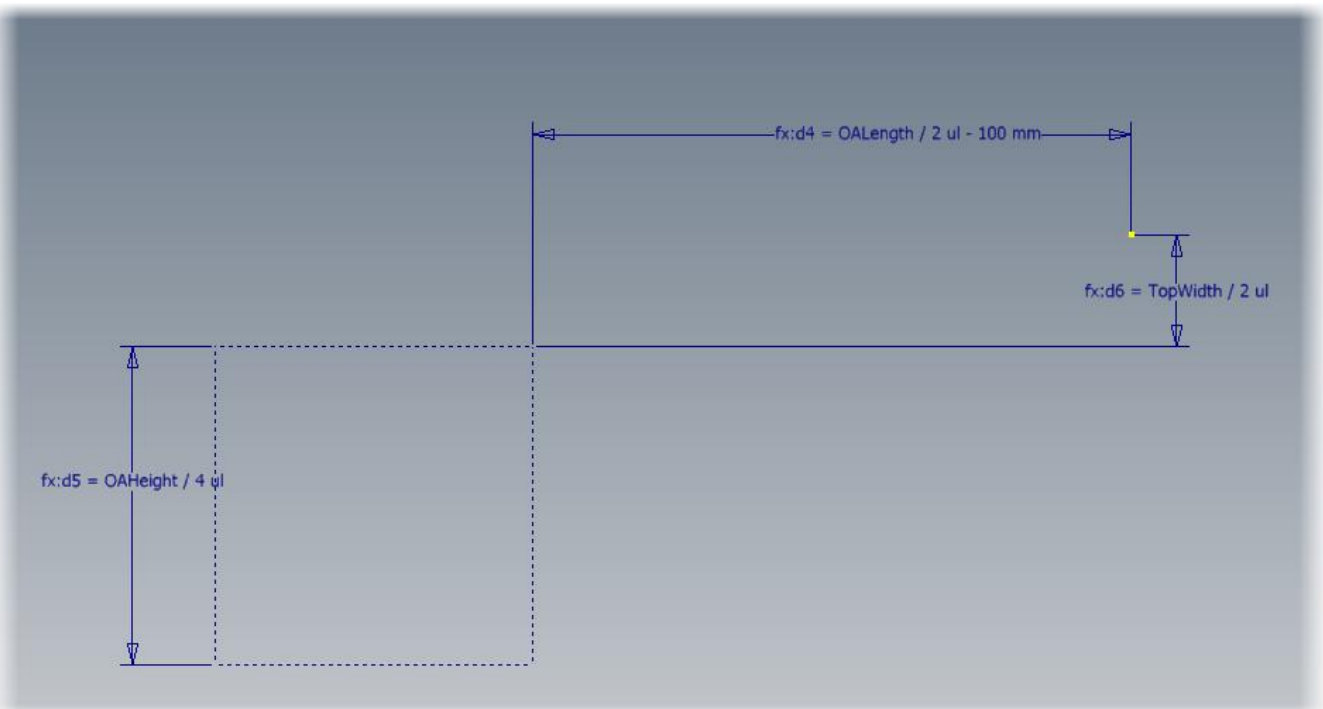


Notice that we will use a number of simple formulas. For example 'OALength/2 ul-100mm' means 'Divide the value of parameter 'OALength' by 2 ('ul' means that this is a unitless value) and take 100mm away from the answer. This gives us the result of 250mm.

Using a formula has the advantage that the design intent is built in. If we change the value of 'OALength' the rest of the values will change accordingly.



You could see the result of a formula by right clicking anywhere in the graphics window and setting the 'Dimension display' value to 'Value'. You could also double click on the dimension parameter (as if you were going to edit it) and hover over the parameter input box.



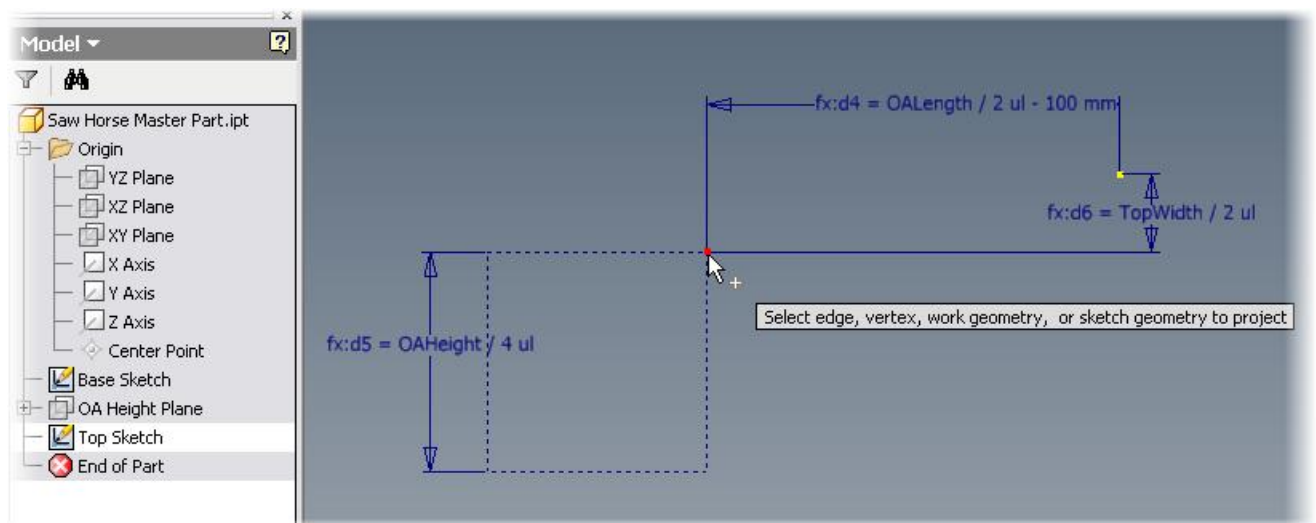
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Click on the big green tick to exit the sketch environment

Create a new sketch and select the 'OA Height Plane' as a base plane.

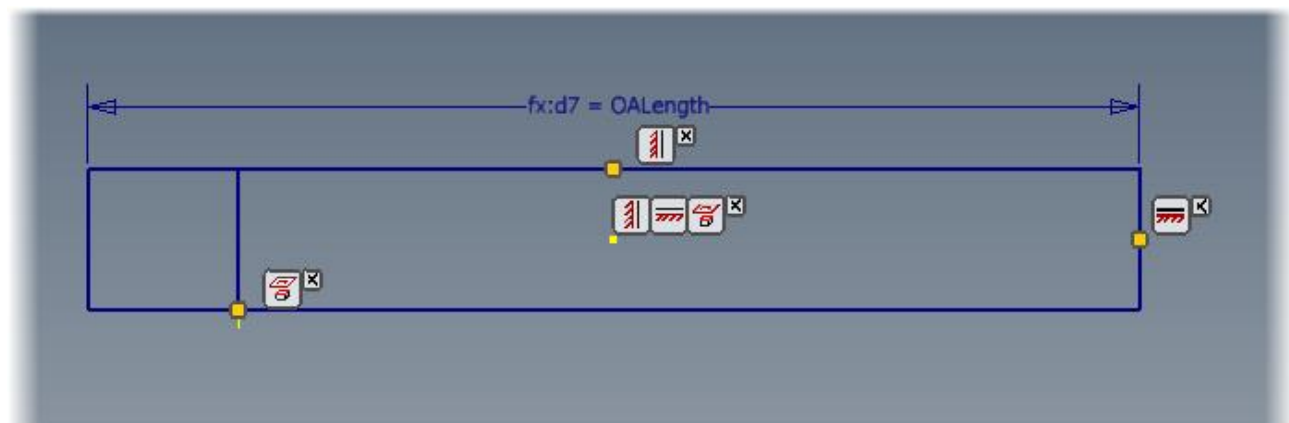
Model Tab > Sketch Panel > Create 2D Sketch

Name this sketch 'Top Sketch'. Use the 'Project Geometry' tool to project the origin point through into 'Top sketch'. Also project the top right hand corner point of the rectangle.



Finish off the top sketch by sketching out a rectangle for the top of the saw horse. Use horizontal and vertical geometry constraints to centre your rectangle about the origin.

Constrain the edge of your rectangle to the projected corner point. Sketch a perpendicular line from the projected corner point to the side of the rectangle. Use a dimension constraint to set the length of your rectangle.



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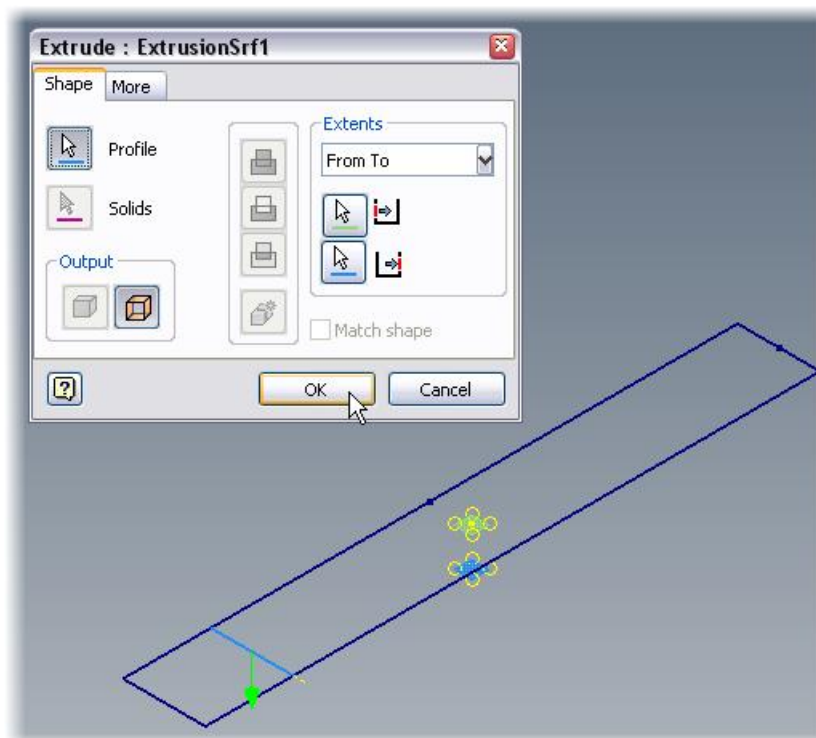
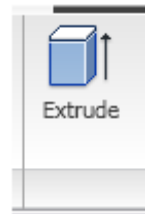
Click on the big green tick to exit the sketch environment. Use the view cube to navigate to an isometric view of your sketches.



Use the extrude tool to create a surface that projects down from the line we created in the top sketch. Set the Output to 'Surface' and set the extents to 'From To'. Pick the 'OA Height Plane' and the 'Top Thick Plane' from the feature browser. Rename this feature 'Projected Line'.

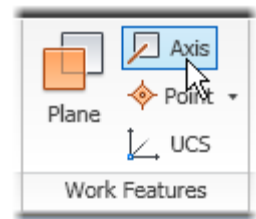
Model Tab > Create Panel > Extrude

Note that the 'Top Sketch' will be 'Consumed' by the new 'Projected Line' surface. You will need to turn the visibility of 'Top Sketch' back on for the next step. You can turn the visibility of 'Projected Line' off for the time being. We will save this for later.



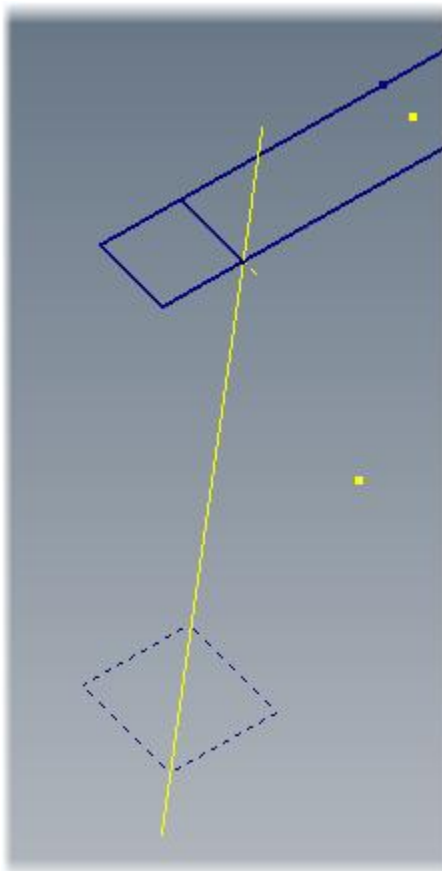
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Use the the axis tool to create a work axis for the Saw horse leg.



Model Tab > Work Features Panel > Axis

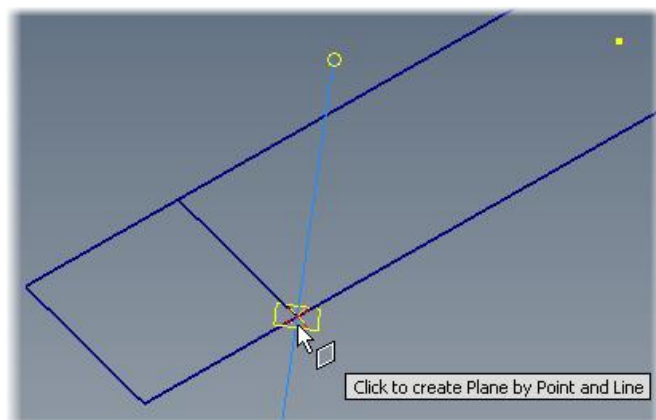
Select the bottom left hand corner of the rectangle in the 'Base sketch' as a start point. Select the projected top right corner of the rectangle from the 'Top sketch' as the second point. Right click and choose 'Done' to complete the command. Rename this feature 'Leg Arris Axis'.



Now create a new work plane for the leg section sketch. Use the 'Plane' tool. Select the 'Leg Arris Axis' as the first input and the projected top right hand point of the rectangle from the 'Top Sketch' as the second input. Rename this work plane 'Leg Sketch Plane'

This will create a work plane exactly at right angle to the leg arris axis.

You can now turn the visibility of these two features off.



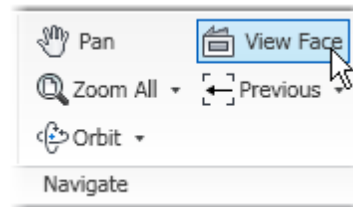
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The next job is to create a profile sketch for the leg. Before you create this sketch, you may need to turn the visibility of our 'Projected Line' surface back on.

Create a new sketch and choose our 'Leg sketch Plane' as the base plane for the sketch. Name this sketch 'Leg Section Sketch' (Don't forget that if it's visibility is turned off you can pick it from the browser instead).

You may find it helpful to use the 'View Face' tool to align your view of the sketch parallel with the screen.

View Tab > Navigate Panel > View Face
(This tool can also be found in the navigation bar)



Use the 'Project cut edges tool', to project the 'Projected Line' surface into the Leg sketch.

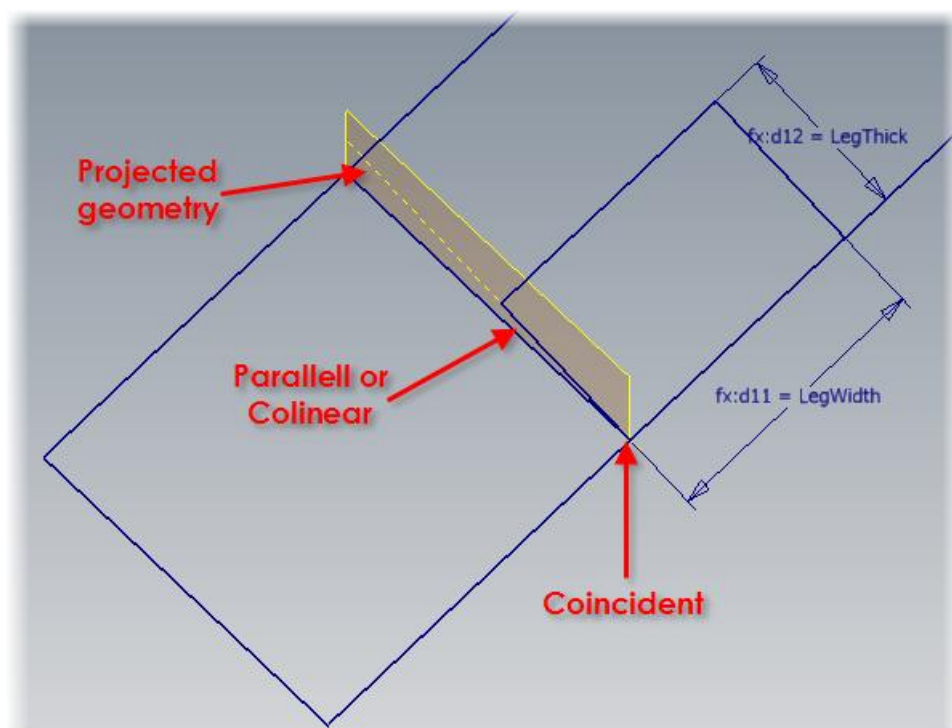
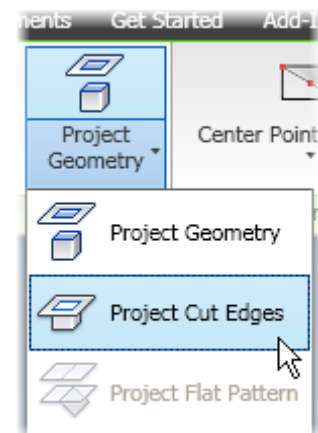
Sketch tab > Draw Panel > Project cut edges
(Under the Project Geometry tool)

The Project cut edges tool is different to the Project geometry tool. The project geometry tool always projects perpendicular to the current plane. In this case our sketch plane is not perpendicular to the geometry we are projecting. This would give us a weird result.

The project cut edges tools will create a new line where our 'Projected Line' surface cuts through our 'Leg sketch plane'.

Tip: Set the Line style to 'Construction' before you project the geometry.

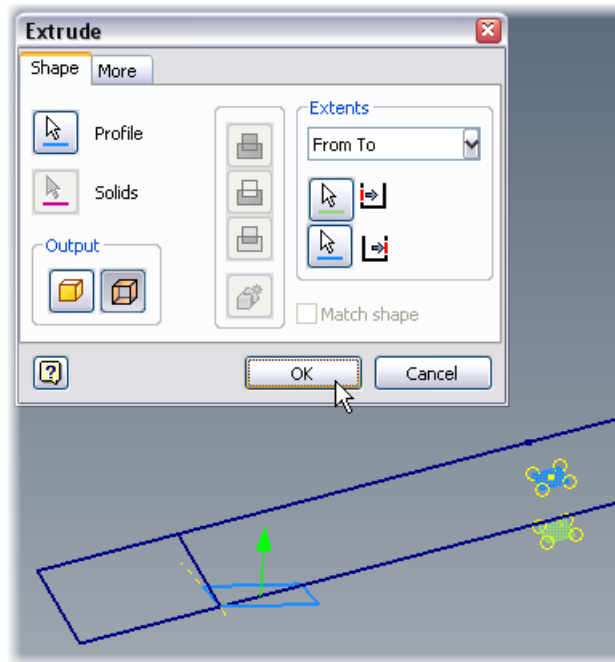
Use the 'Three point rectangle' tool to sketch a rectangle, which is coincident and parallel with the projected geometry. Use Dimension constraints to set the size of the rectangle to suit our leg.



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The next step is to use our Leg geometry to create the geometry which we will use to create the housing in the top board.

Use the Extrude tool to create an outline of the leg. Set the output type to 'Surface' and the Extents to 'From To'. Use the 'OA Height Plane' and the 'Top Thick Plane' to form the extrusion. Name this feature 'Leg Surface'.

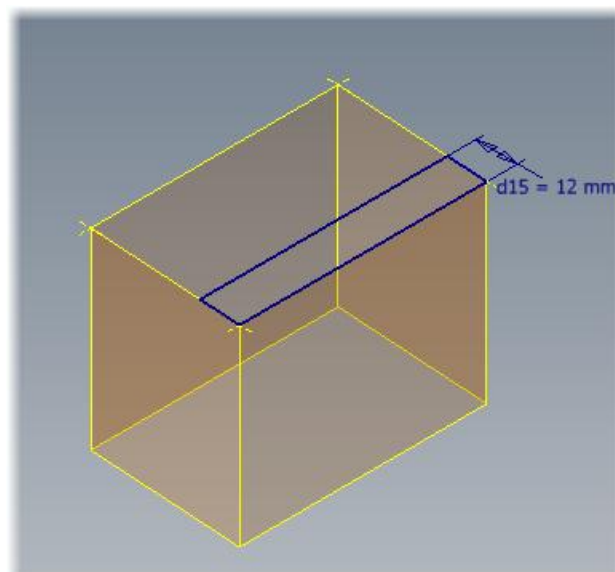


We can now use this surface to form the top and bottom of our housing.

Create a new sketch. Use the 'OA Height Plane' as a base plane. Name this sketch 'Housing Top Sketch'.

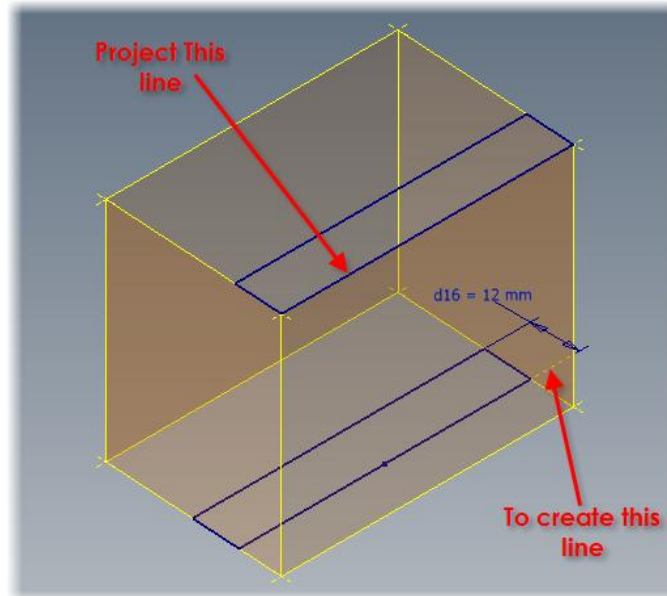
Use the Project geometry tool to project the corners of the 'Leg Surface' into the sketch. Draw the outline of the housing, making sure that the edges are not perpendicular and that they are aligned with the leg surface corners. Set the depth of the housing to 12mm.

I have found it 'Safer' to project the corners of features into a sketch, rather than the edges or loops. If something unexpectedly changes in the future it is easier to hook the corners up again than recreate the 'failed' edges.

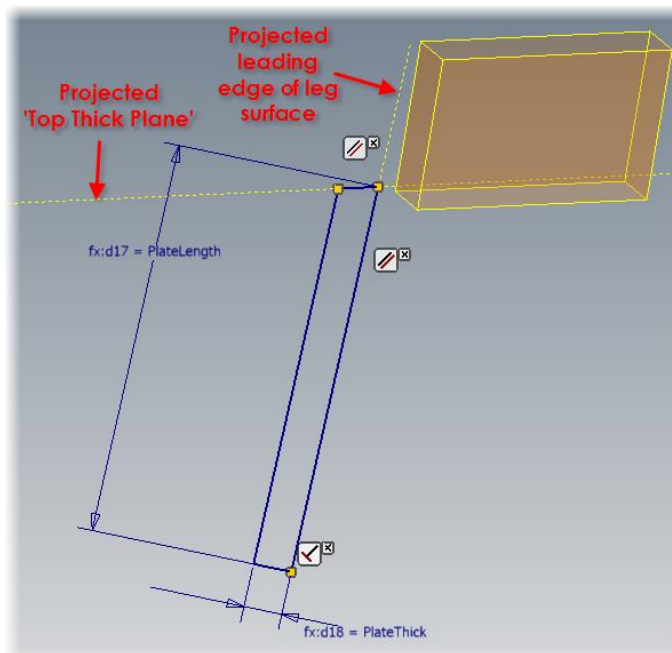


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Finally, create a further sketch using the 'Top Thick Plane' as the base plane. Once again project the corner points through from the leg surface. You will also need to project the front edge of the 'Housing Top sketch' through. Sketch the outline of the housing in and set the depth to 12mm. Call this sketch 'Housing Base Sketch'.



Our final sketch is for the end plates. You can turn the visibility of our housing sketches off for the moment. Make a new sketch, choosing the XZ Plane as a base plane. Project the 'Top Thick Plane' and the leading edge of the 'Leg Surface' into your sketch. Make these lines construction geometry. Sketch out the profile of the end plate and set its length to 'PlateLength' and its thickness to 'PlateThick'. Name this sketch 'Plate Section Sketch'.



We have now created all the geometry, work planes and surfaces we are going to need for our model. We can now start to create the 'solid' features.

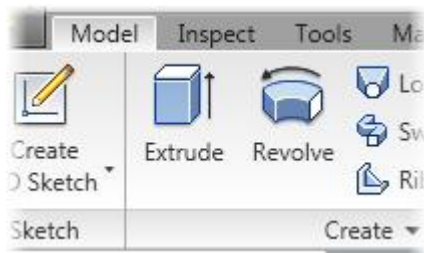
Modelling a Mitre Block**Creating a Solid, or 'Sketch based feature'**

Make 'Top Sketch' Visible and turn the visibility of all the other sketches and work features off.

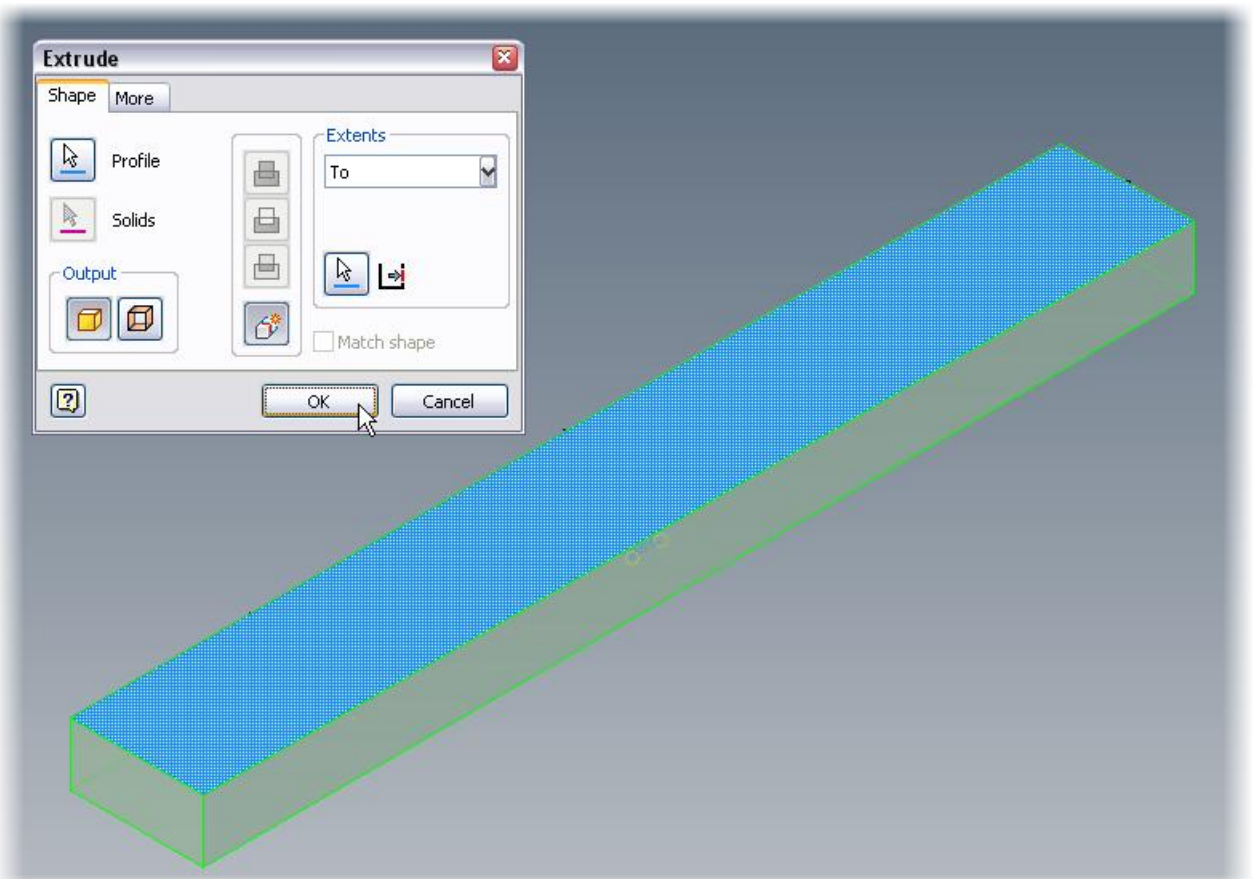
This isn't necessary, but it does make it a lot easier to see what's going on.

Use the 'Extrude' tool to give your 'Top Sketch' a thickness.

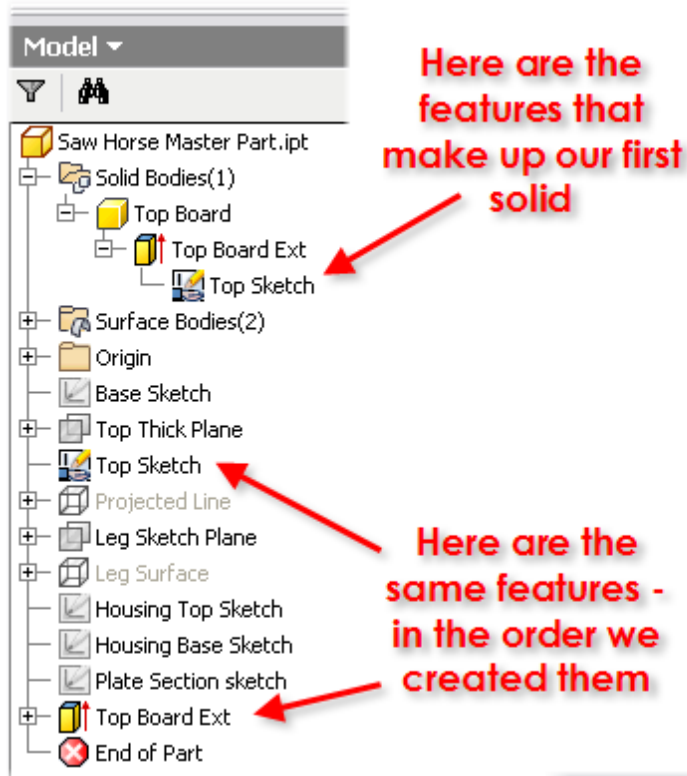
Model Tab > Create panel > Extrude Tool



Set the 'Extents' option to 'To' and pick 'Top Thick Plane' from your browser.



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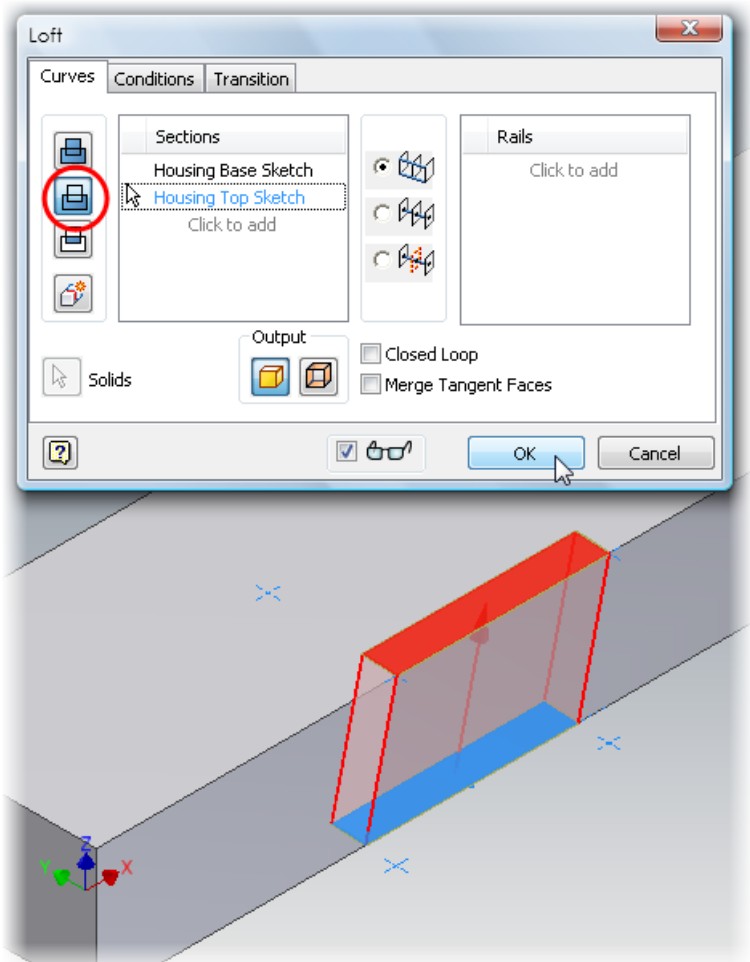
Note that a new 'Solid Bodies' Folder will appear in your feature browser. The Solid Bodies folder contains one solid. Nested below this is the same extrude feature that is shown at the base of the feature tree.

Name the Solid body 'Top Board'.

Name the Extrusion 'Top Board Ext'

Let's create the housings for the legs next. Turn the visibility of 'Top Sketch' off. Turn the visibility of 'Housing Top Sketch' and 'Housing Base sketch' on. Create a new Cut Loft feature using these two sketches.

Rename this feature 'Housing Loft'.

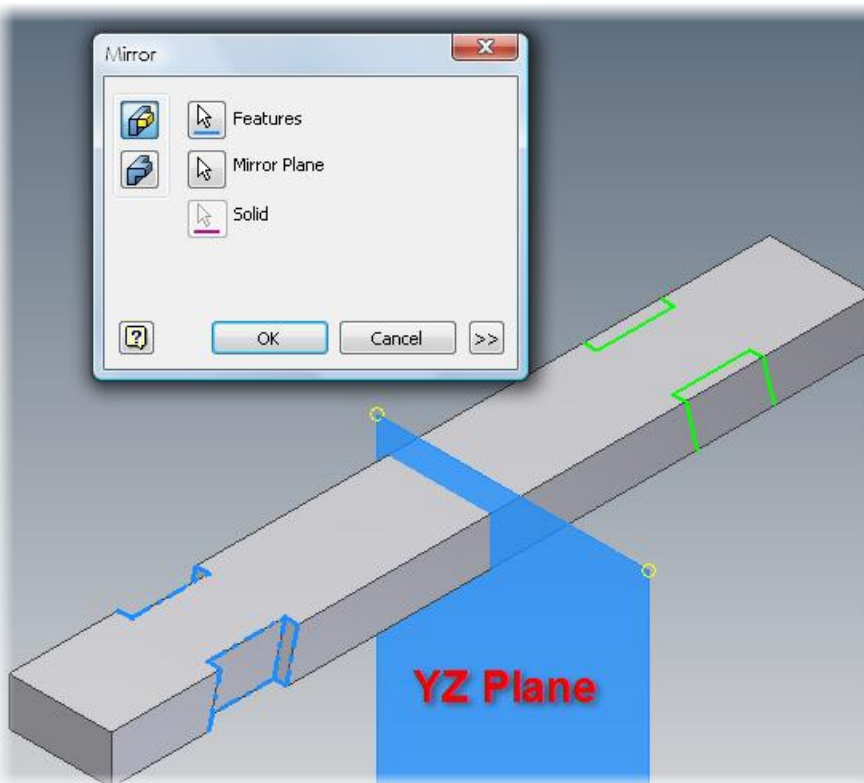
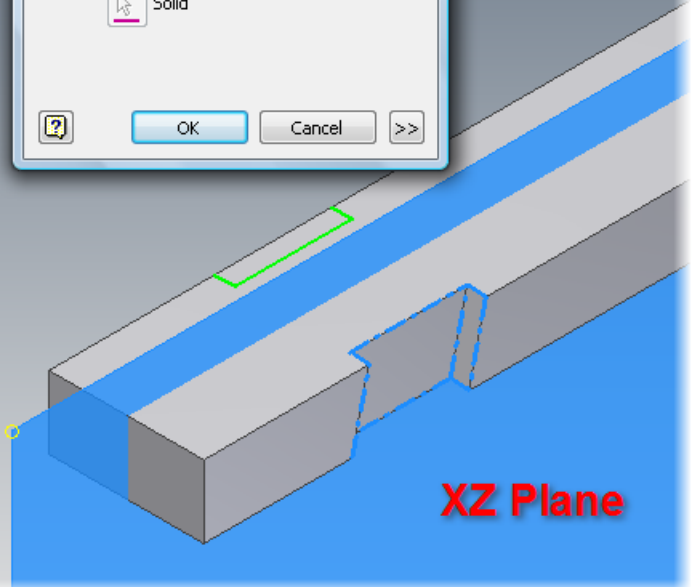
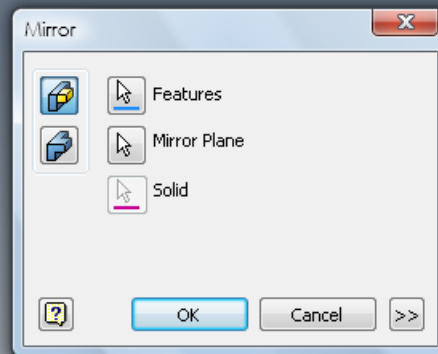
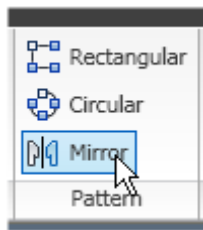


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Next we can use the Mirror command to mirror the housing twice to create all four housings.

Model Tab > Pattern Panel > Mirror

First Choose the 'Housing loft' as a feature and Choose the XZ Plane as a mirror plane. Name this Feature 'Housing Mirror'.



Next Mirror again, choosing the 'Housing Mirror' as your Feature and the XZ Plane as the mirror plane.

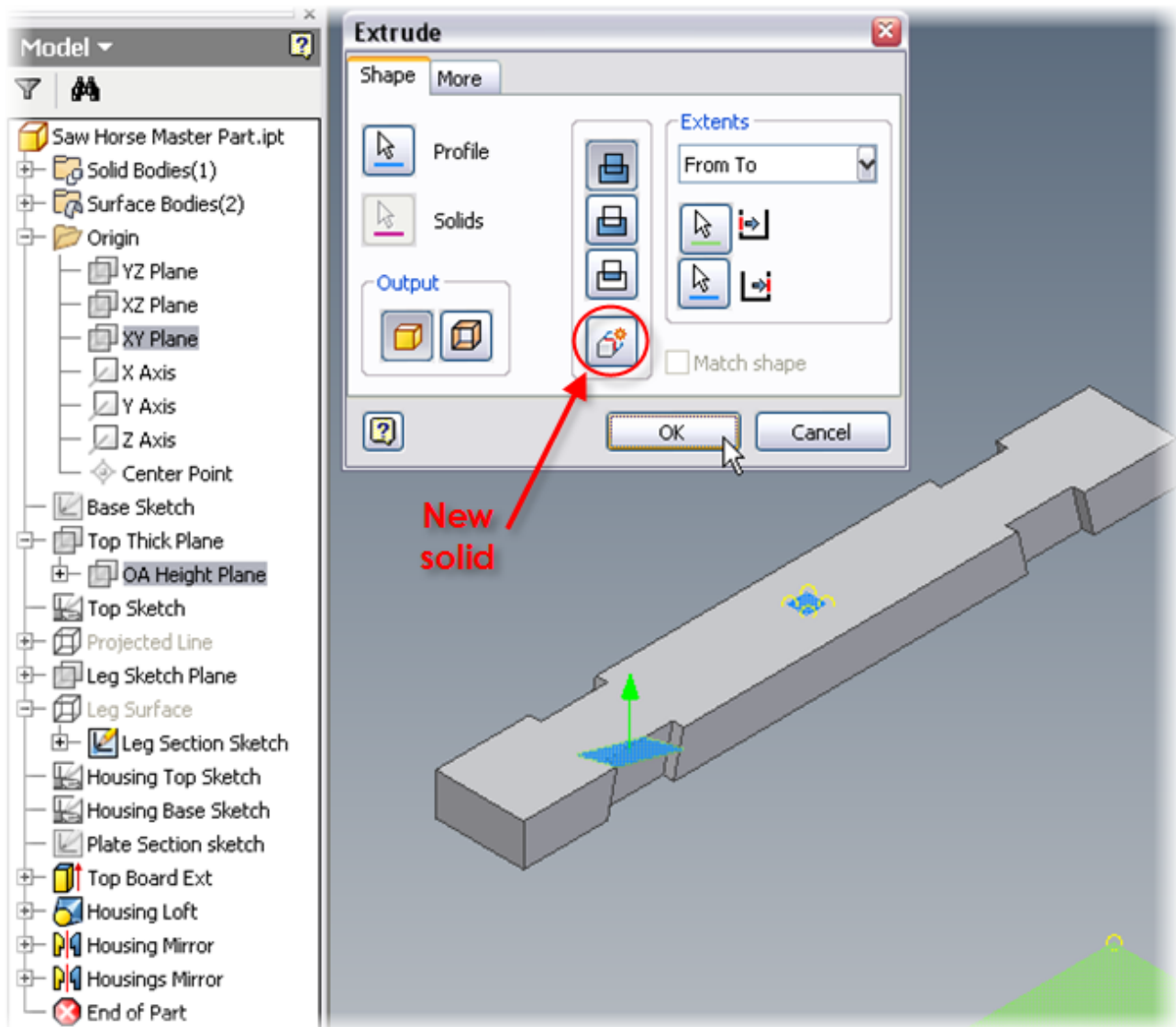
Rename this feature 'Housings Mirror'.

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Next, create the leg. Make sure the 'Leg section sketch' is visible. Use the 'Extrude' tool. Set the 'Extents' to 'From To' and pick the XY Plane and the 'OA Height Plane' as your start and finish planes.

Make sure that you have the 'New Solid' Button pressed before you hit OK.

Rename this new extrusion 'Leg Ext'. Rename the Leg Solid 'Leg'.



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We have a little bit of work here to do to create the 'Bird's Beak' joint at the top of the leg. Use the 'Combine' tool to remove the portion of the top of the leg that intersects with the top board.

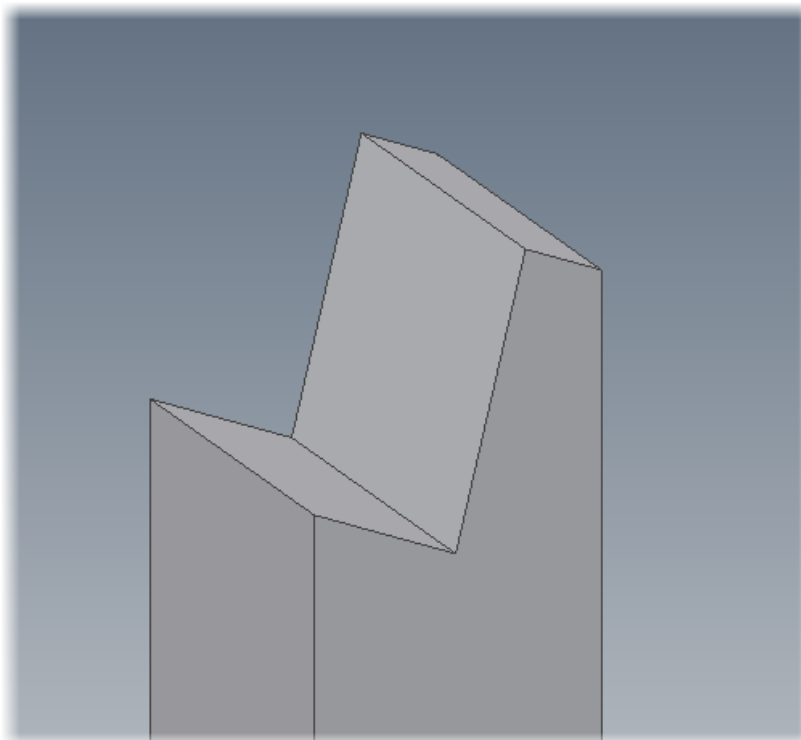
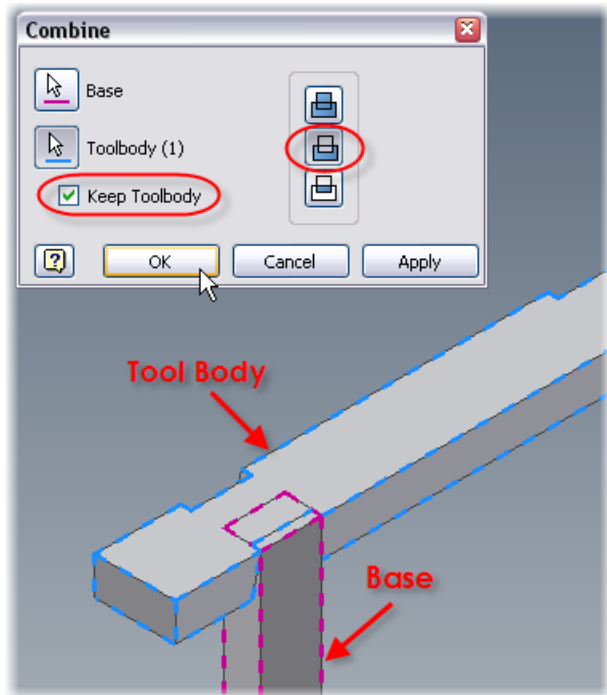
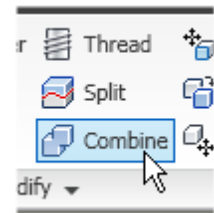
Model Tab > Modify Panel > Combine.

Use the 'Leg' as the Base solid. Use the 'Top Board' as the tool body. Set the type of combine to 'Cut'. Make sure that you have the 'Keep Tool Body' option checked.

The 'Top Board' will immediately disappear. Don't panic! Inventor has turned the visibility of this Solid off to allow you to see the result of the combine.

You can turn the visibility of 'Top Board' back on when you've finished admiring your Bird's beak joint.

Rename the 'Combine' feature 'Birds beak' and let's move on.



The finished Joint.

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The final feature to create is the End Plates. Make sure that you have the 'Leg', 'Top Board' and 'Plate Section sketch' Turned on.

Use the extrude tool to create the end plate.

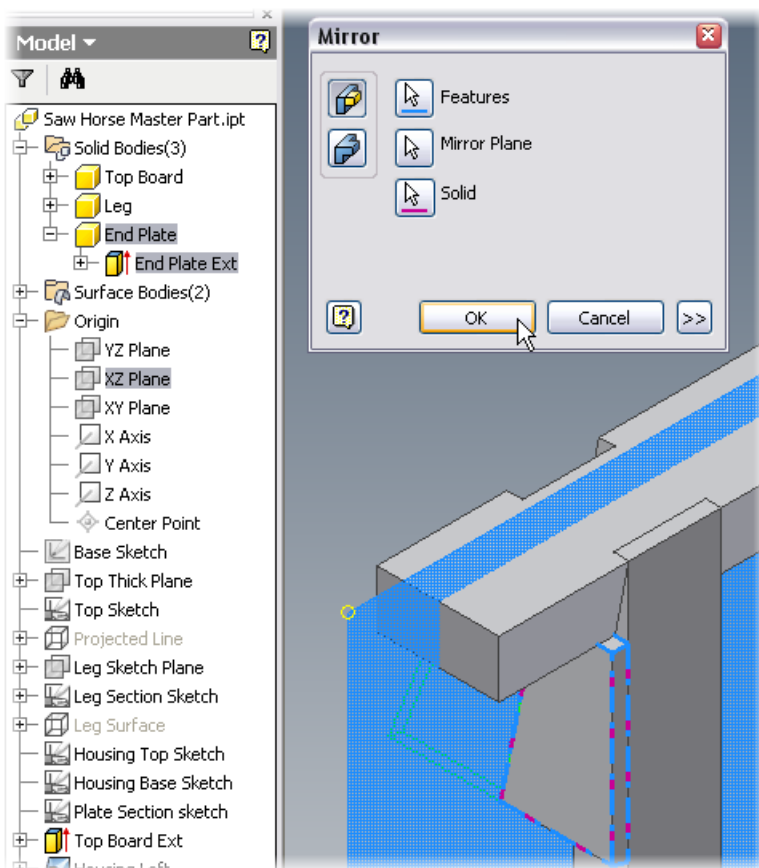
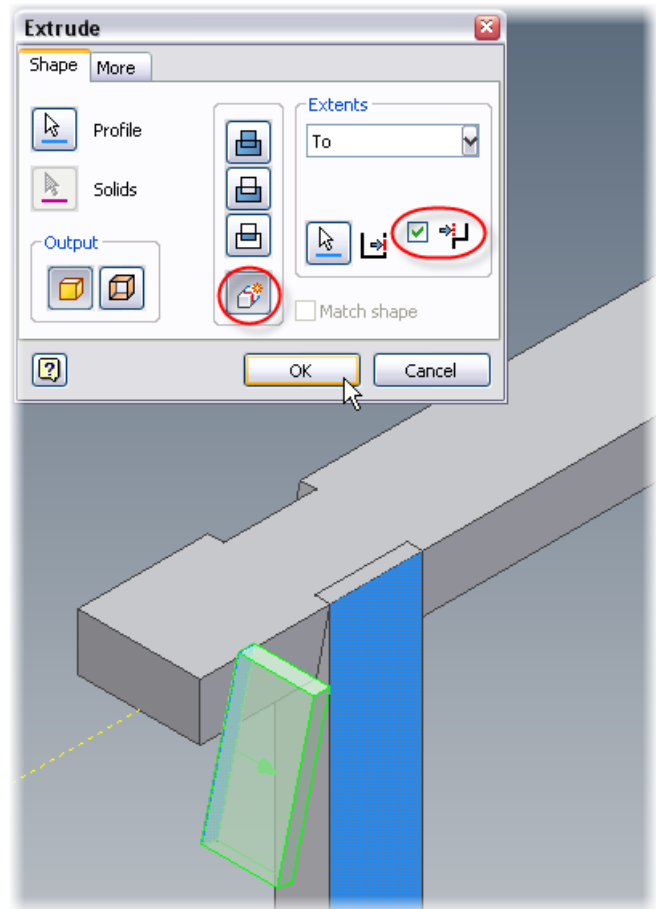
Extrude the 'Plate section sketch' using the 'To' option (under Extents).

Select the side face of the leg as the terminating surface.

Make sure that you have the 'Terminate feature on extended face' option checked.

Make sure that you've clicked on the 'New solid' button.

Rename the Extrusion 'End Plate Ext'. Rename the new solid 'End plate'.



You will notice that this will only create half of the end plate. To create the other half use the 'Mirror' tool.

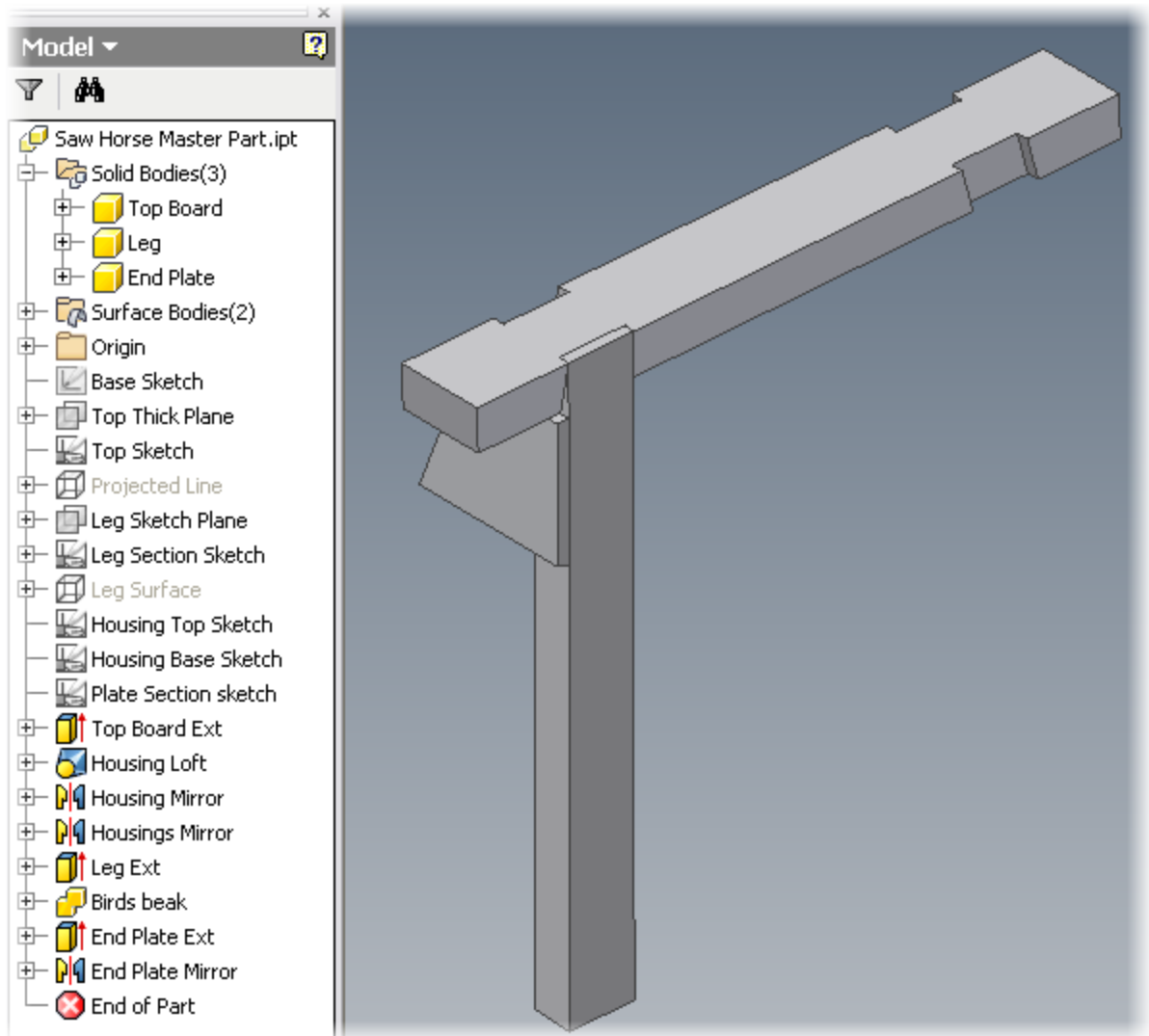
Mirror the end plate extrusion about the XZ Plane.

Notice that you have the option of which solid you want to add the feature to. By default the 'End Plate' solid will be highlighted, because this is the last solid created.

Rename this feature 'End Plate Mirror'

Modelling a Mitre Block**Creating the Assembly**

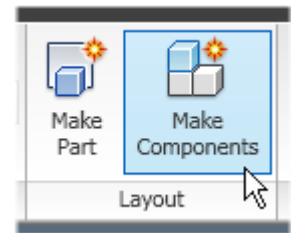
We have now created all the solids we will need to make our assembly. That was easy! We will now make use of Autodesk Inventor 2010's 'Make component's' tool to automate the process of deriving the solids out into an assembly.



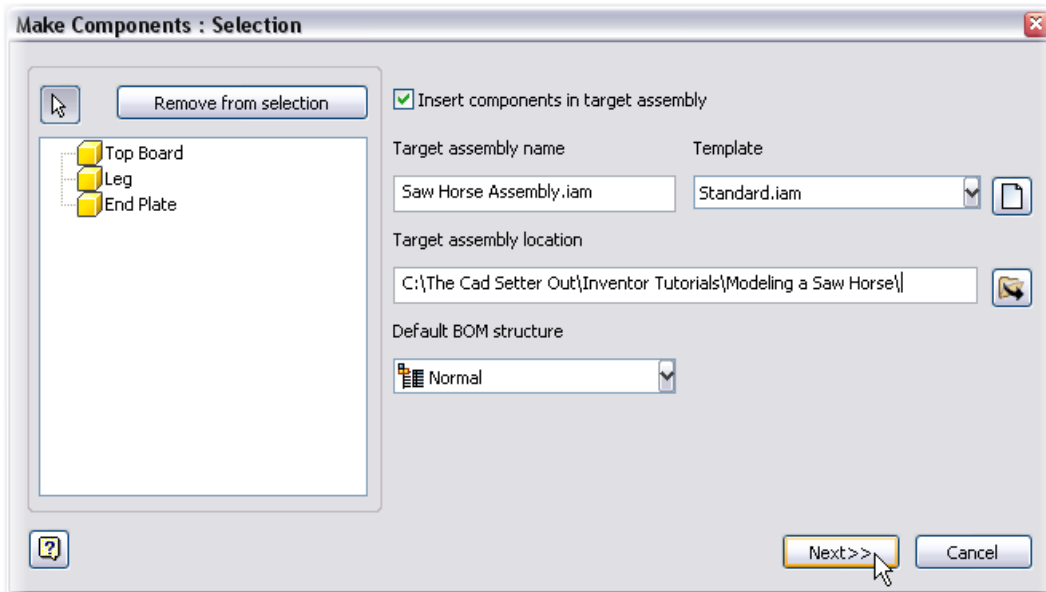
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Manage Tab > Layout Panel > Make components

Pick the solids that you want to use in your assembly from the browser tree (in our case this is all of them).



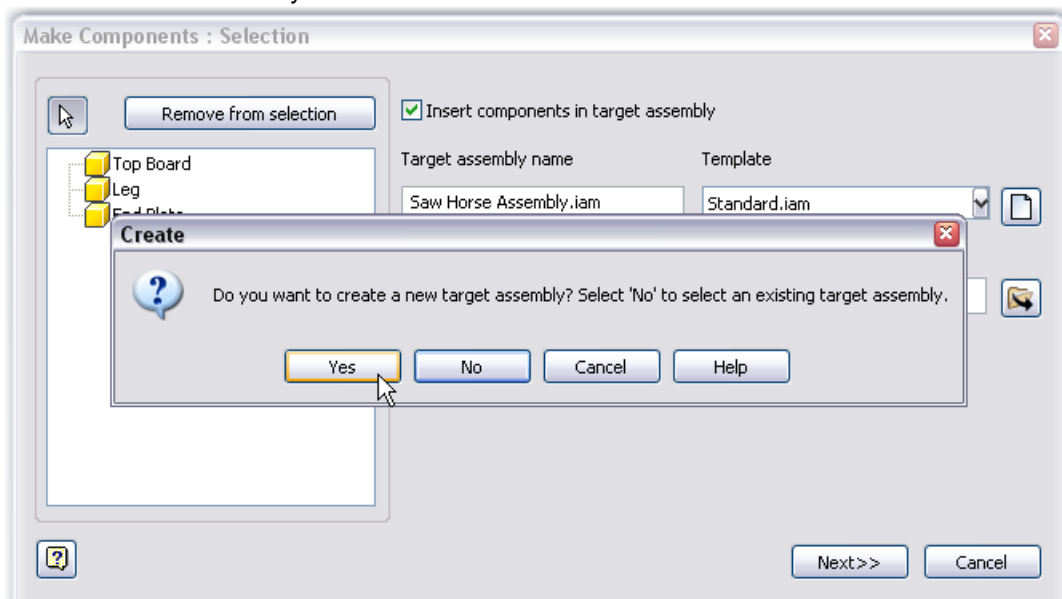
Make sure that the 'Insert components into target assembly' box is checked. You can leave 'Template' set as 'Standard.iam' and 'BOM structure' set to 'Normal'.



Click on the folder icon to choose the name of your Assembly and where you want to store it. You will immediately be confronted by this message:

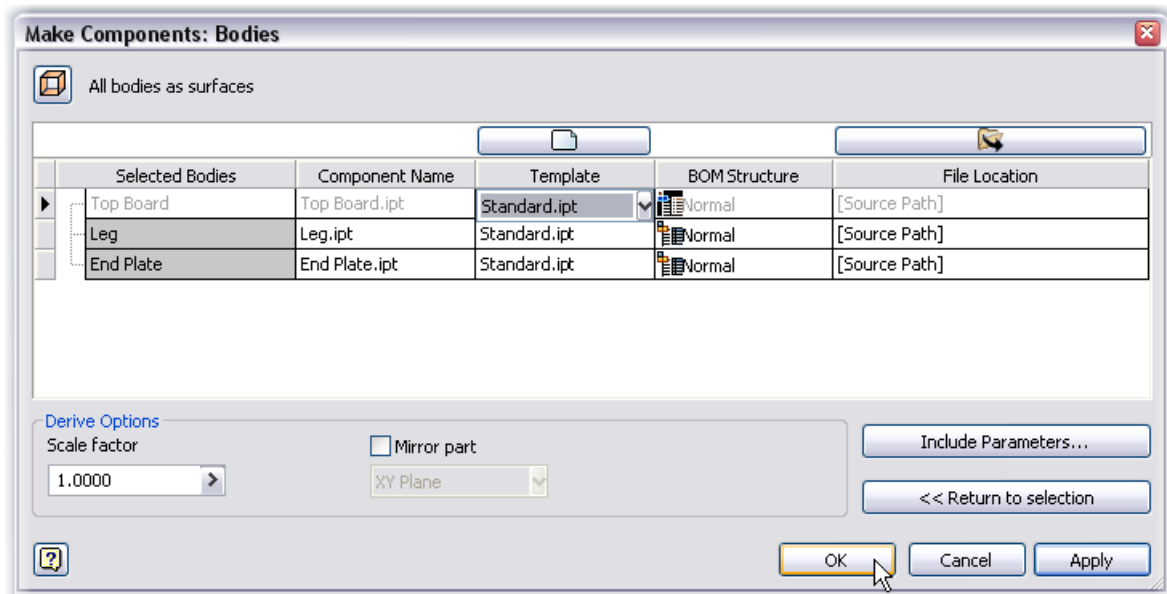
'Do you want to create a new target Assembly? Select 'No' to select an existing target assembly'

I still have to read this carefully every time I see it. In this case we are going to choose 'Yes' because we want to create a new assembly from scratch.



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The next dialog that you will see is the 'Bodies' dialog.



This dialog gives you a lot of control over how you want your assembly to be created.

To add more bodies to the dialog. Use the '<< Back to selection' button to go back to the previous dialog.

The 'Component name' column allows us to change the name of the component, just type a new name in the cell.

The 'Template' column allows us to pick a different template file for the part. When you click in the cell a drop down list appears. Note that this drop down is populated from your default templates folder. You can pick on the button at the top of the Column to browse out and pick a different template file. You can pick multiple rows and use the button at the top of the column to change the template for many solids at once.

You can save time by creating templates that are pre set to use your standard materials.

The BOM structure column allows us to pick 'Normal', 'Inseparable', 'Phantom' or 'Reference' for our parts (Using the BOM to create a parts list is next time).

The file location column allows us finite control over the individual places that the parts will be saved. Note that you can press the button at the top of the column to browse your folders.

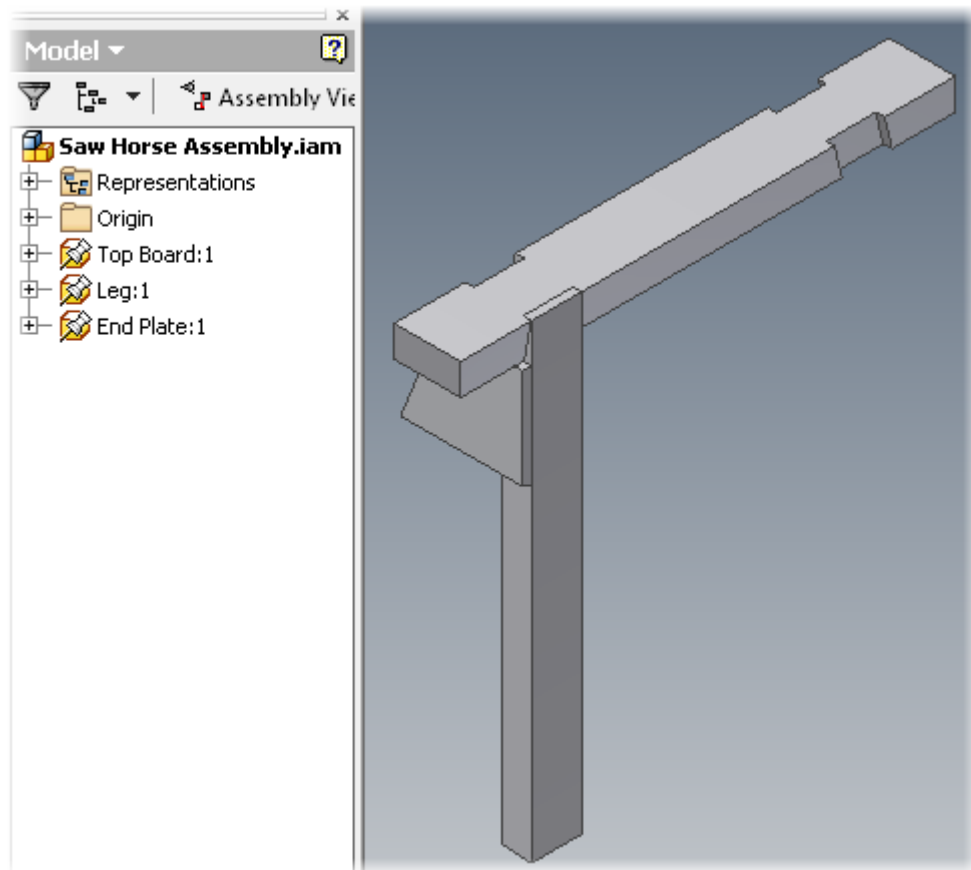
Note the 'Scale Factor' and 'Mirror part' options. The 'Include parameters' button allows you to link parameter values to your derived parts.

Click OK when you are done.

Modelling a Mitre Block

That's your assembly complete. That was easy!

All right, I noticed the lack of legs! We've got a little bit more work to do.

**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.

Note that the 'Make components' tool has created the assembly File, created all the part files, and grounded our parts at 0, 0, 0 (the origin). There is no need to apply assembly constraints. The master part file controls the dimensions and locations of the parts.

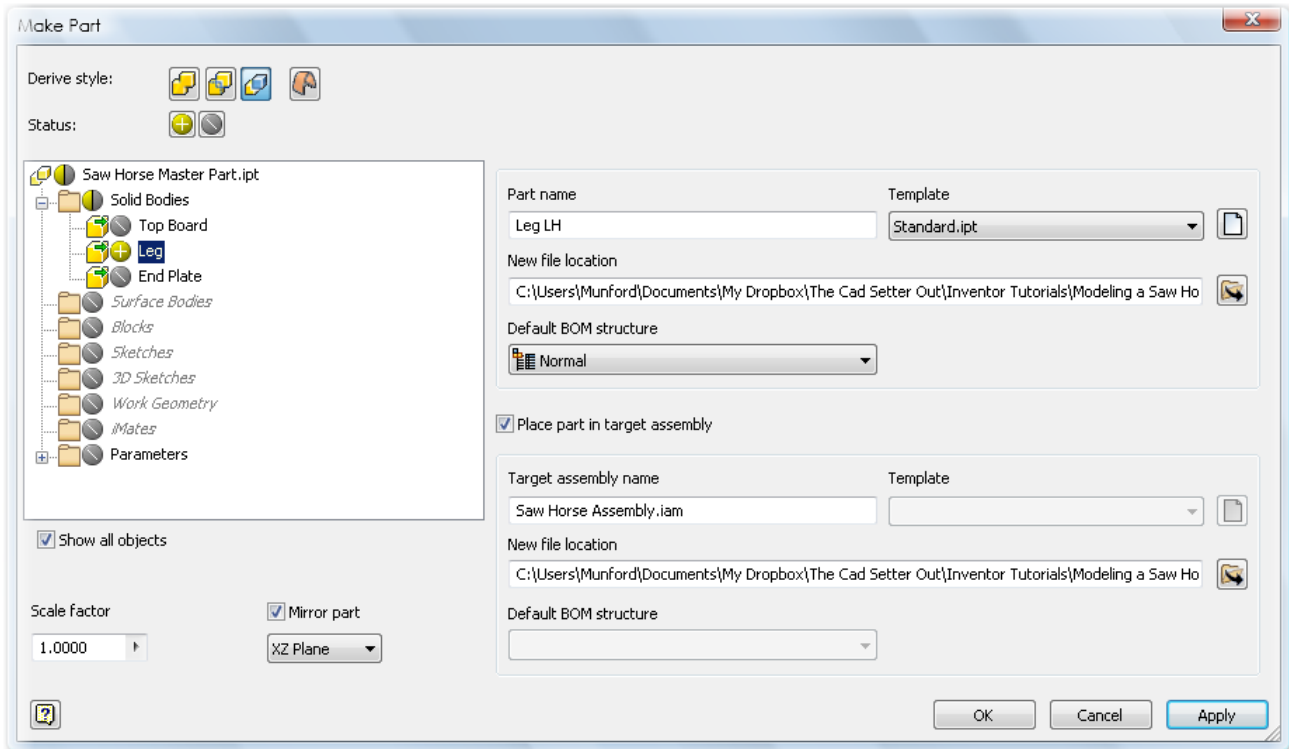
The next task is to use the 'Make part' tool to derive out a single solid – a mirrored version of the leg. Navigate back to your master part file.

Modelling a Mitre Block



Manage tab > Layout Panel > Make Part

The 'Make part' dialog is pretty daunting. But don't be put off, it is pretty much asking us for the same information as the 'Make component' dialog.



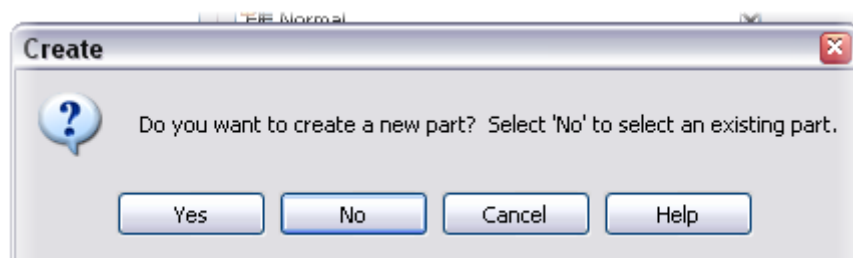
We are going to create a mirrored copy of the leg. We could create this in the assembly using the 'Mirror Component' tool, but this doesn't give us any control over which plane the part is derived around. The 'make part tool is more precise.

First you will need to browse down the 'Saw Horse Master Part' 'Tree' and select the solid you want to derive.

If you highlight the required solids before you trigger the make part command they will automatically be added to the selection.

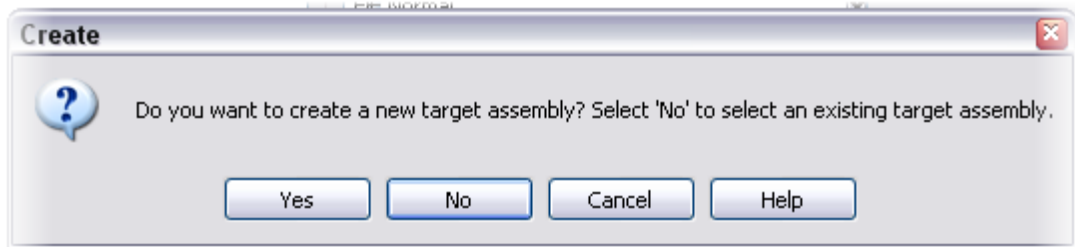
Click on the folder button at the end of the 'New File Location' input box. As with the 'Make Components' tool, you will be prompted to decide whether you want a new part or not. In this case hit 'Yes' – we want a new part. Call the part 'Leg LH'.

You can leave 'Template' and 'Default BOM structure' to their default settings.



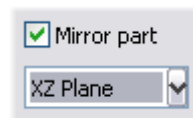
Modelling a Mitre Block

The next section of the dialog is prompting us to insert the part into an assembly. You can uncheck this if you just want to create one derived part. In this case we want to insert the part into our 'Saw Horse Assembly' – so click on the folder button at the end of the dialog.



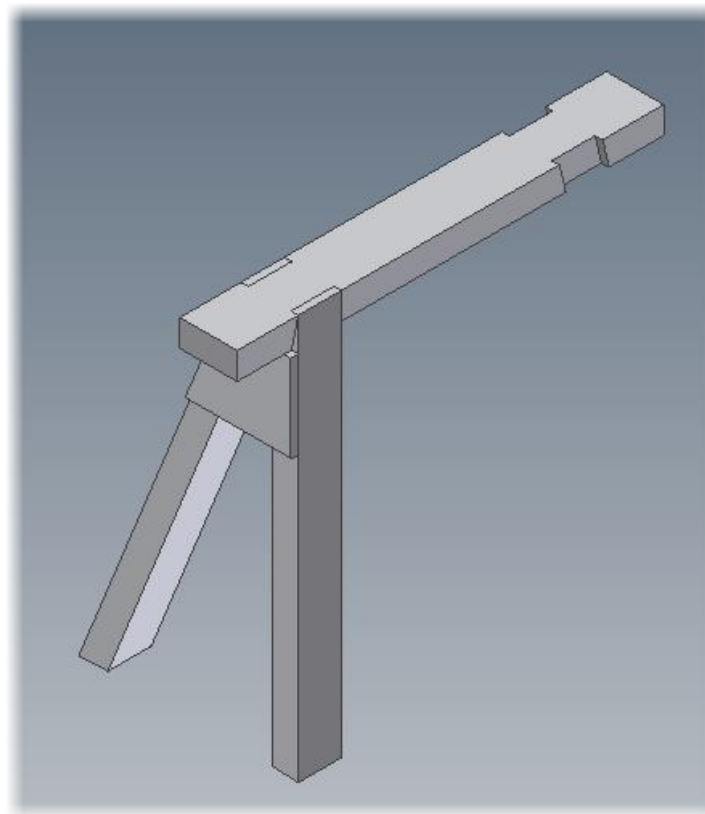
Once again you will be prompted for a choice. Do you want to create a new assembly or insert the part into an assembly you already have? In this we want to click on 'No' and browse out to our 'Saw Horse Assembly'.iam.

The final step is to check the 'Mirror Part' box and pick a plane to mirror the part around.



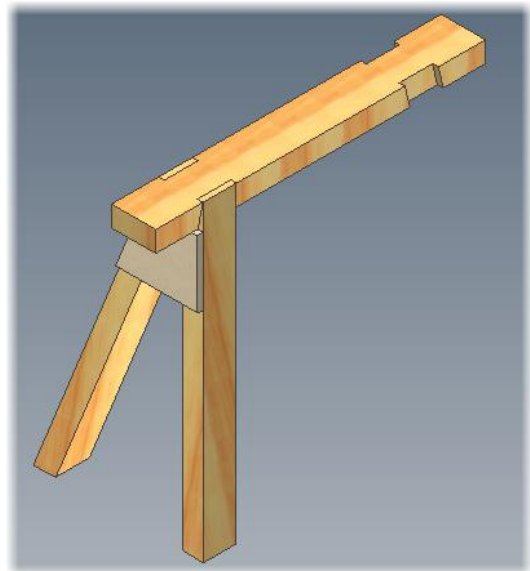
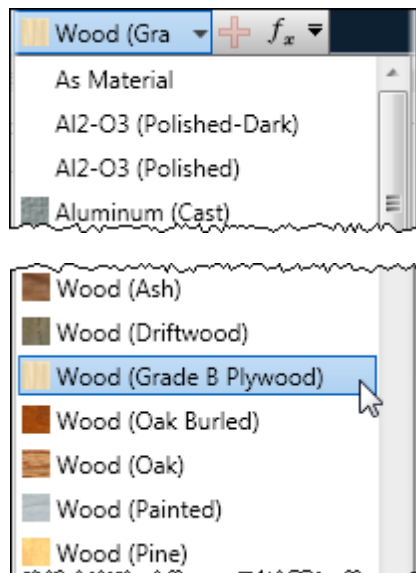
Hit the 'OK' button when you are happy with your choices.

The left hand version of our leg will be inserted into our assembly and grounded at 0, 0, 0,



Modelling a Mitre Block**Changing the look of the part**

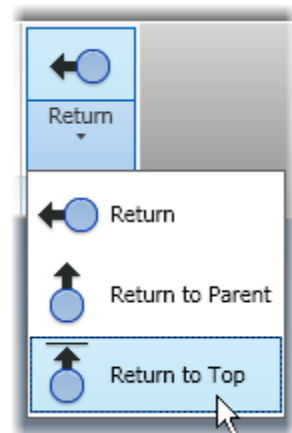
You can use the Colour override drop down to allocate a different look to your parts.



Double click on each part to edit it. When you have changed the colour of the part, double click on the next part in the assembly browser to edit it. There is no need to return to the assembly each time.

When you are happy with the colour of your parts, click on the return tool and pick 'Return to top' to return to the top level assembly.

Model Tab > Return Panel > Return tool.



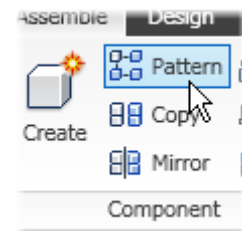
Clicking 'Return', rather than 'Return to top', will return you to the last part you edited.

There is one last process for us to complete. To create the final two legs and the other end plate we will use the 'Pattern Component' tool

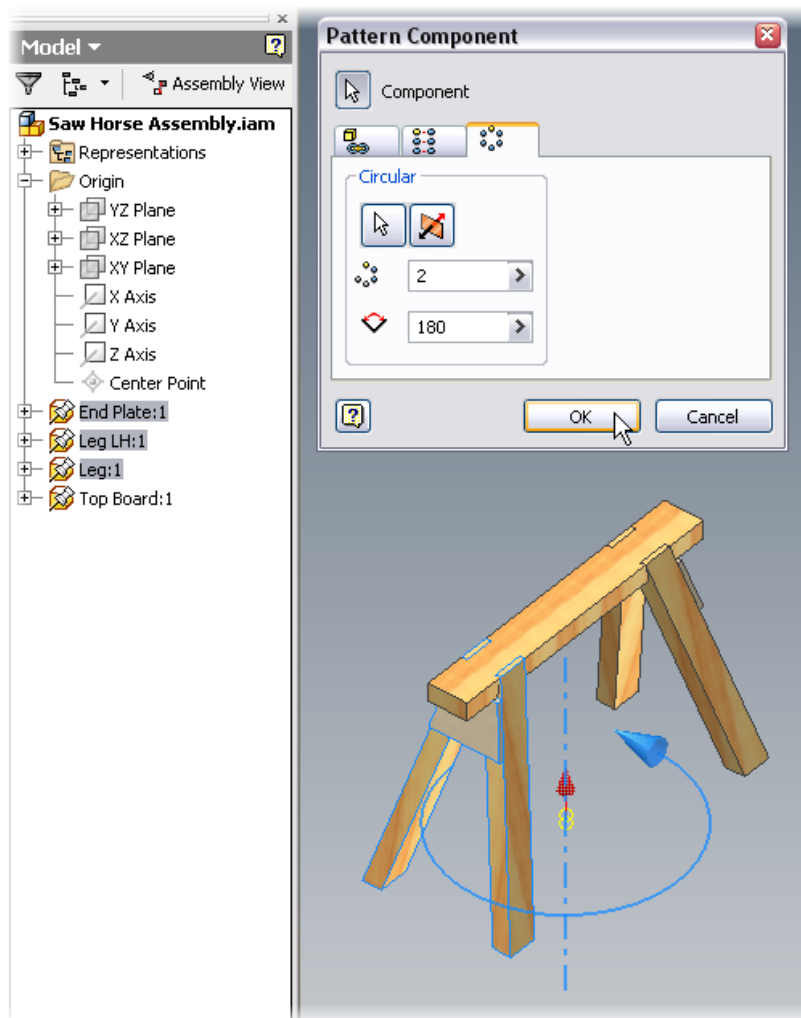
Modelling a Mitre Block**Creating the final two legs**

Assemble Tab > Component Panel > Pattern

Select the End plate, Leg and Leg LH. Pick the 'Circular pattern tab'. Pick the Z Axis to rotate around. Set the number of patterns to 2 and the angle between patterns to 180.

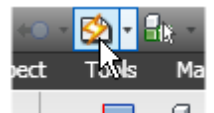


This completes our assembly. Save the assembly file as 'Saw Horse Assembly' – and you are done.



Modelling a Mitre Block**Testing your model**

Open your 'Saw Horse – Master Part' for edit. Open the parameters dialog and experiment with changing the 'OAHeight', 'OALength' or 'LegThick' parameters that we created earlier on. After each change, update and save your Master file and return to the assembly model. Click on the 'Update' tool to see the parts update (you may need to click twice).



Note that the parts that make up the Saw Horse Assembly are all linked to the Master file. As you make changes to the Master file, the assembly model updates as well. Note that there are no constraints to fail!

Master part multi solid modelling is a really cool way of creating mid to large assemblies that automatically update based on the values held within a master part. Note that the Master part file needs to be managed along side the part and assembly files. If you loose your master file the whole assembly may fall down around your ears! Multi solid modelling may not be appropriate for Library parts.

Note also that our parts don't contain any value for 'Length', 'Width' and 'Thickness' We would need to manually add these into each part if we wanted them to be included in the Bill of materials (BOM) and parts list. This can be time consuming, so don't be tempted to derive every part when you could re-use a part or use a library part instead.

Modelling a Mitre Block

That's your Saw horse model completed! I hope that you are pleased with your results. I hope that you now have an understanding of how create a master part containing multiple solids and how to use the 'Make Component' and 'Make part' tools to derive solids out into parts and build your assembly model.

This isn't the only technique for producing assembly models in Inventor. We could also have used the 'Bottom up', 'Skeletal modelling' or 'in place' techniques', but that's next week...

Key Concepts:

1. Create a master part
2. Add parameters
3. Add work features
4. Add sketches
5. Geometrically constrain sketches
6. Dimension sketches
7. Create solids
8. Derive the solids into parts and create assembly.