

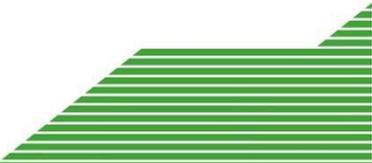
DENFORD[®]

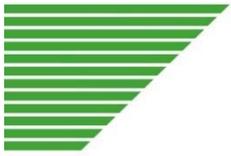
CAD/CAM Projects

Living Hinge Container Project



CAD SUPPORT BOOKLET





Denford: Living Hinge Container Project – VLS Series Laser Cutter CAD Support Booklet

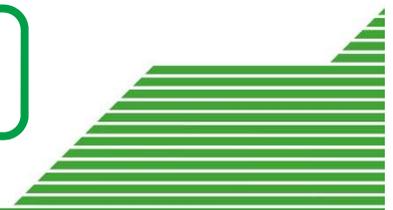
To draw this product in AutoDesk Fusion 360, we will be designing in the following order. This is to ensure that all bodies will relate to each other and will be fit for manufacture on the Universal Laser Cutter.

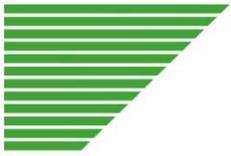
The process will be as follows:

1. Creating the main body
2. Adding the sides to the body
3. Unfolding your design
4. Designing your living hinge
5. Adding detail to your design
6. Rendering your design to produce a photo export
7. Exporting your design for manufacture

Creating the Main Body	
<p>Log into Fusion Launch Autodesk Fusion 360. Use your Autodesk ID to log into Fusion.</p> <p>If you do not have an Autodesk ID, you can create an account free of charge.</p>	
<p>In front of you is the main work area. During this tutorial you will learn about its various functions.</p>	
<p>Open the Data Panel. Click to display the Data Panel. The data panel slides open on the left.</p>	

Note: This project uses AutoDesk Fusion 360. However, the project can be adapted for use with other 3D Design Software Packages.



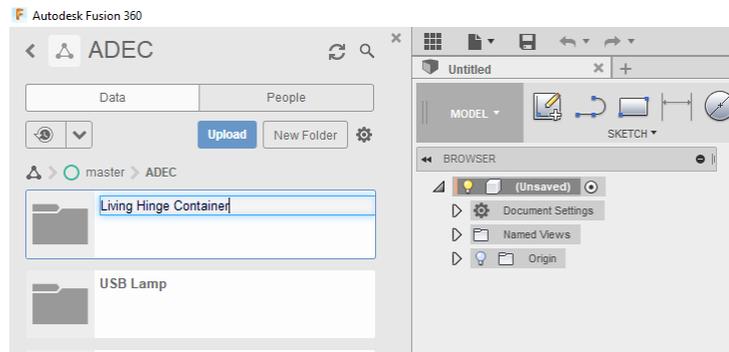


Creating the Main Body

Create a new project.
Click on **"New Project"**.

Enter **Living Hinge Container** in the Name field.

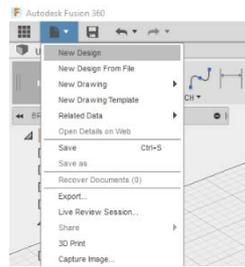
Double click on the new project you have created to start working in this folder.



Create a new design.

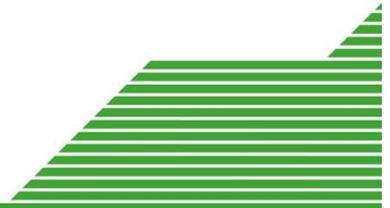
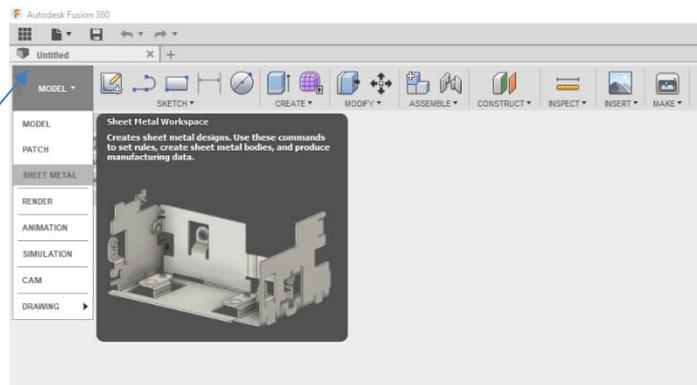
Click and select **New Design**.

A new empty design is created.



For this exercise, we will be working in the Sheet Metal environment.

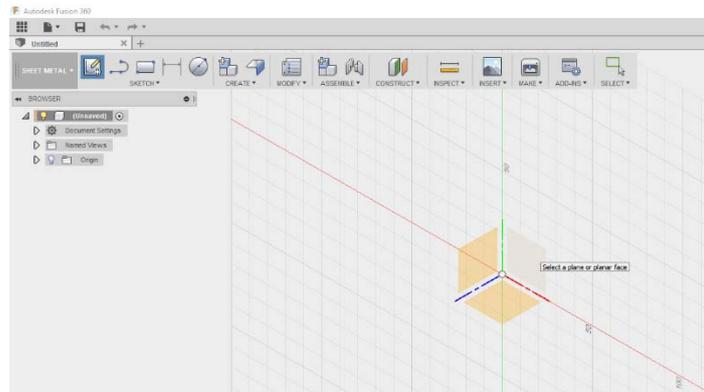
To access this **Click Model - Sheet Metal**



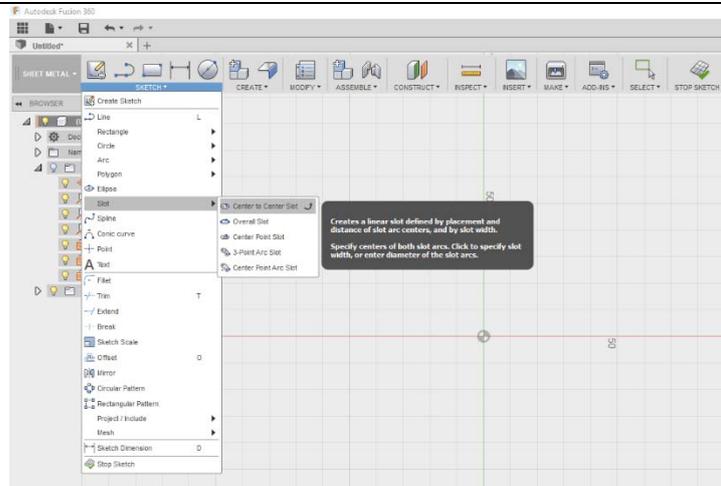


Creating the Main Body

Click **Create Sketch** and select the workplane you will be working in. This should be the XY (right) workplane.



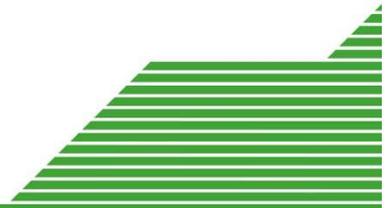
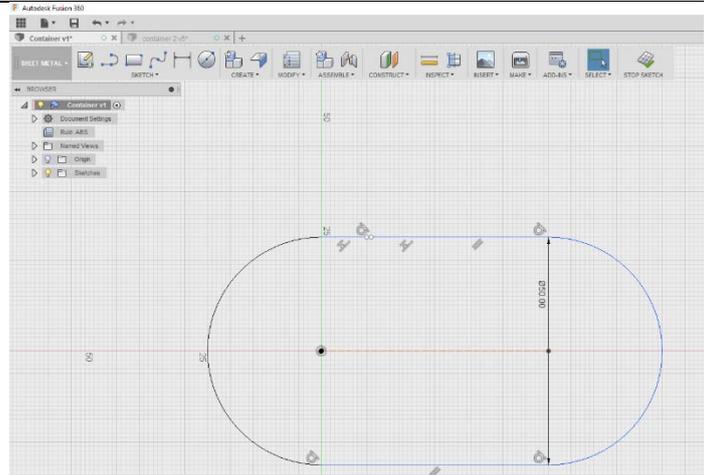
Under the sketch menu, click **Slot – Center to Center Slot**

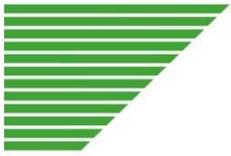


Starting at the origin point  **left click** and move your mouse to the right. Type in: **50** press **Enter** twice, then move your mouse up and type **50**

Finish the slot by pressing **Enter** on your keyboard.

You will now have a fully constrained slot.





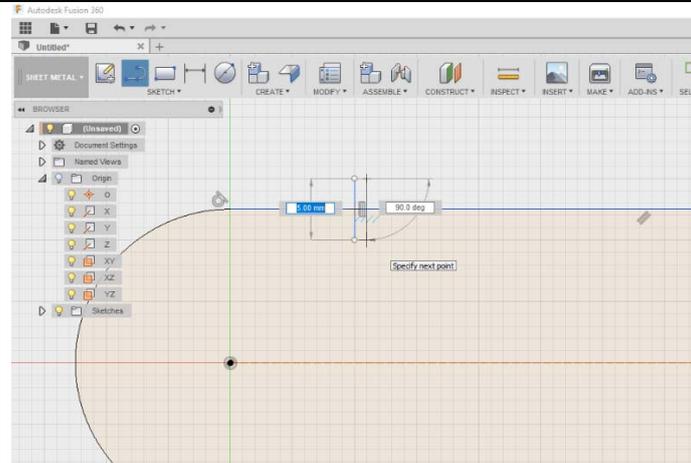
Creating the Main Body

For this exercise, we do not want a full slot, as we want to be able to open the container.

Click **Sketch - Line**

Draw two vertical lines intersecting the top horizontal line.

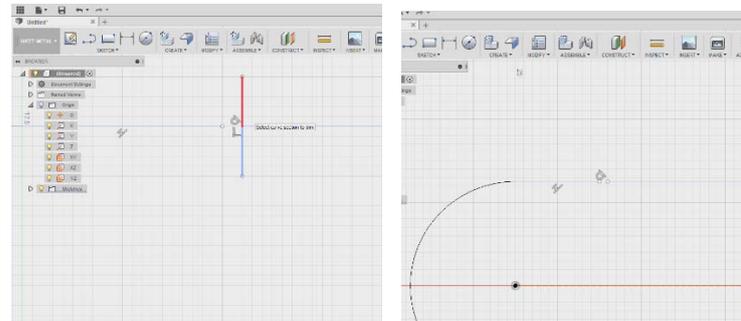
The lines opposite are **10mm** in from the end of the line with **1mm** between them.



Next click **Sketch - Trim**

Now delete the section between the two lines, and delete the two vertical lines so you end up with a sketch like the one on the right.

Click **Stop Sketch**



Click **Flange**, and select the line.

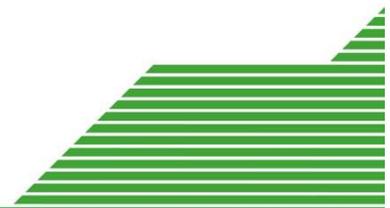
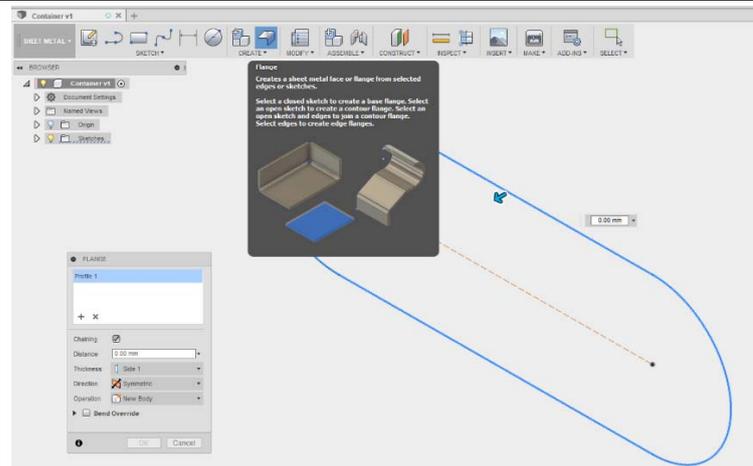
A Dialog box will pop up with an arrow on the line.

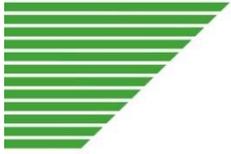
Type in the following:

Distance: **100**

Direction: **Symmetric**

Click **OK** to accept.





Creating the Main Body

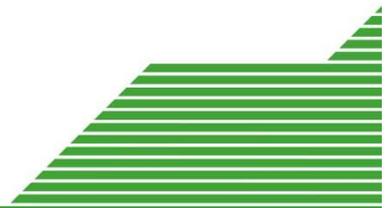
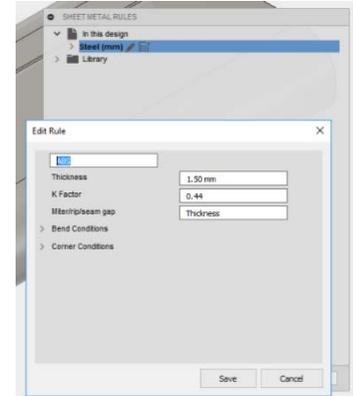
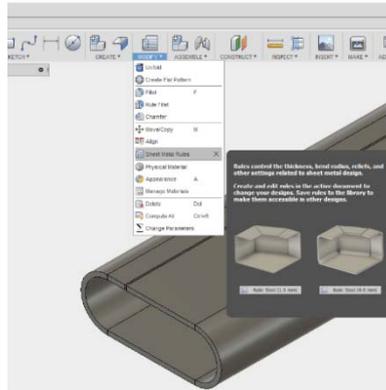
You will now have a 3D shape, However, it is too thick, so we need to change to the same thickness as our sheet of material (1.5mm).

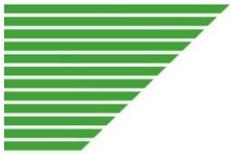
Click **Modify - Sheet Metal Rules**

In the pop-up box move your mouse over the steel(mm) and **click the pen** Now change the material name to **ABS** and Thickness to **1.5mm**

Click **Save** and **Close**

You will now see the wall thickness has changed.



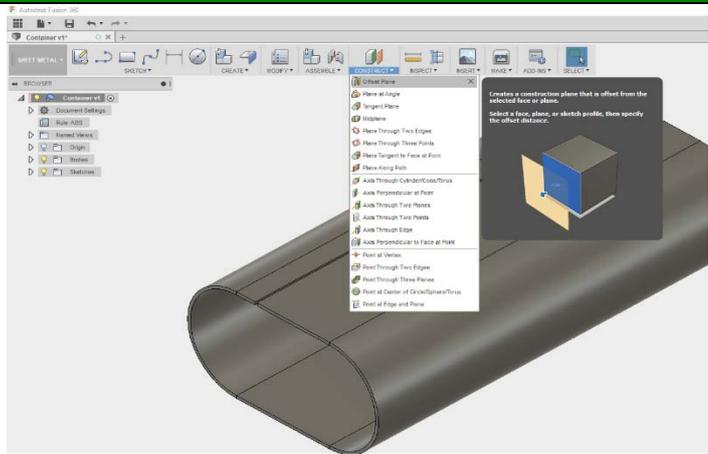


Adding the sides to the Body

We are now going to add the sides to the part. This is going to be done by using the side profile and projecting a sketch to extrude from.

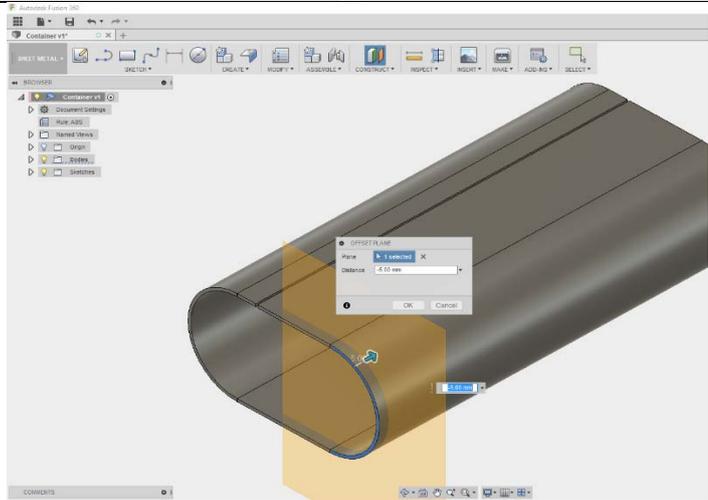
Firstly we will create an offset from the face.

Click **Construct- Offset Plane**



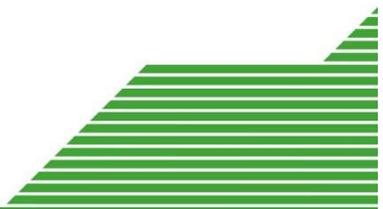
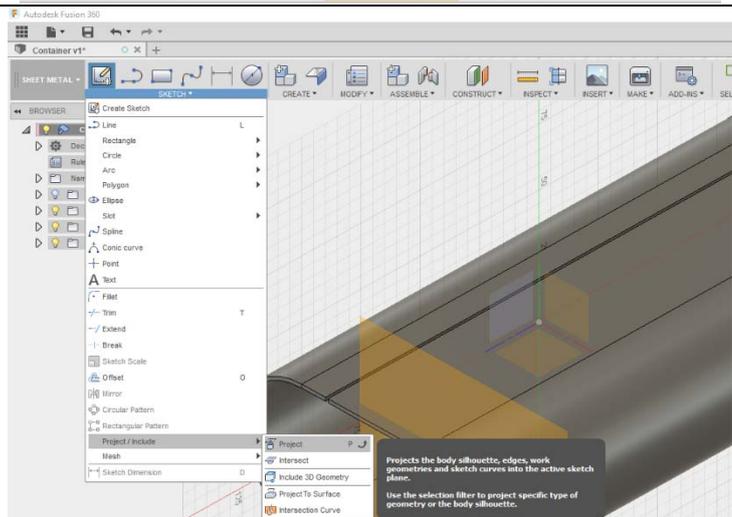
Select the front face and type **-5**

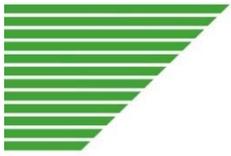
Click **OK** to accept.



Click **Sketch - Project/include - Project**

Select the workplane you have just created.



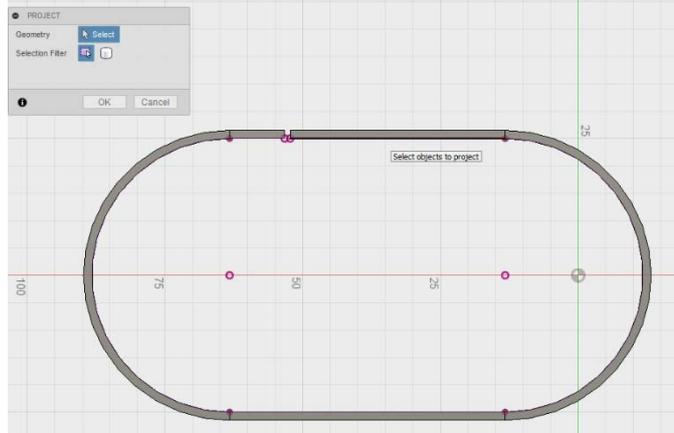


Adding the sides to the Body

Select the internal lines/curves one by one and click **OK**

Now draw a **line** between the open slot at the top to complete an enclosed shape.

Click **Stop Sketch**



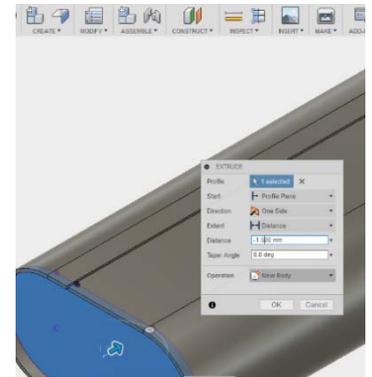
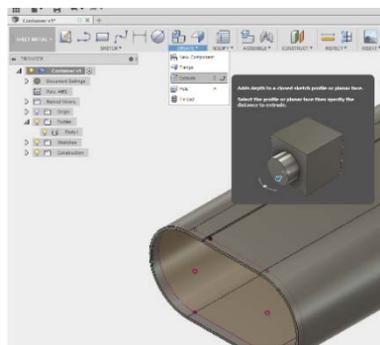
Create - Extrude

In the dialog box type

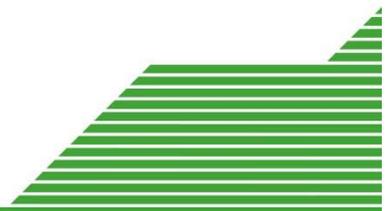
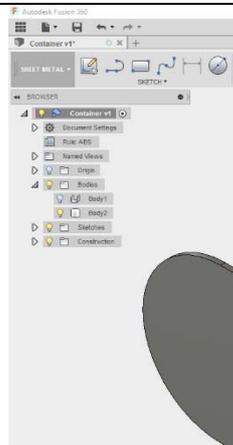
Distance: **1.5mm**

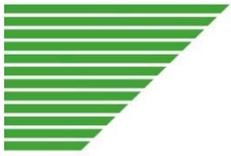
Operation: **New Body**

Click **OK** to accept.



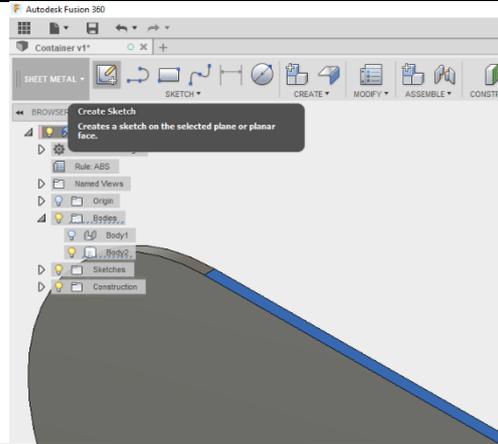
Toggle off the bulb on Body 1, so you are only working on the side you have just created.





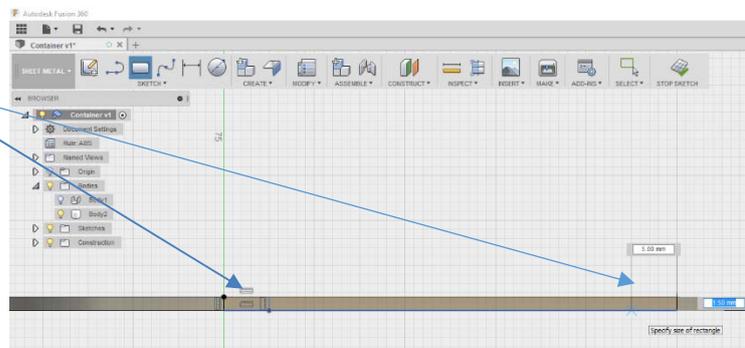
Adding the sides to the Body

Select the top surface of the side part (highlighted blue) and click **Create Sketch**.



Draw two rectangles, **5mm x 1.5mm** at each end of the flat face.

Click **Stop Sketch**



Click **create - Extrude**

Select the two rectangles just drawn and click the following in the dialog box:

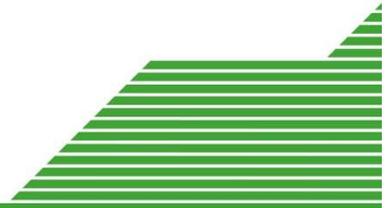
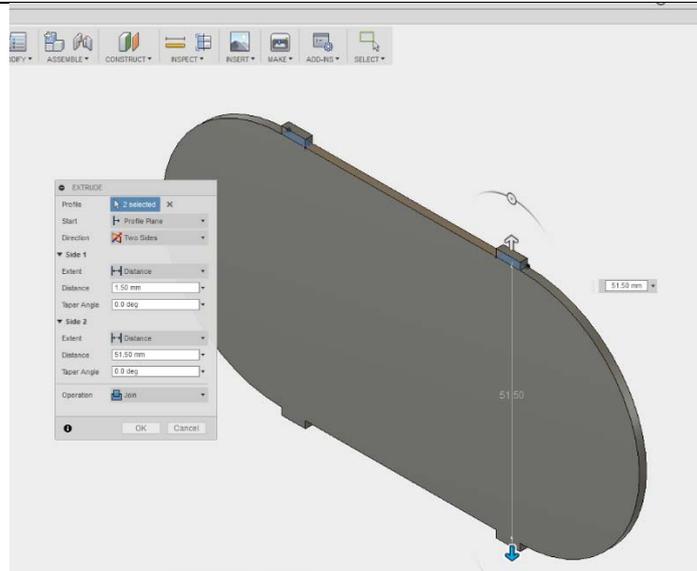
Direction: **Two Sides**

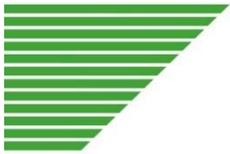
Side 1 Distance: **1.5**

Side 2 Distance: **51.5**

Ensure operation is set to **join**

Click **OK** to accept.



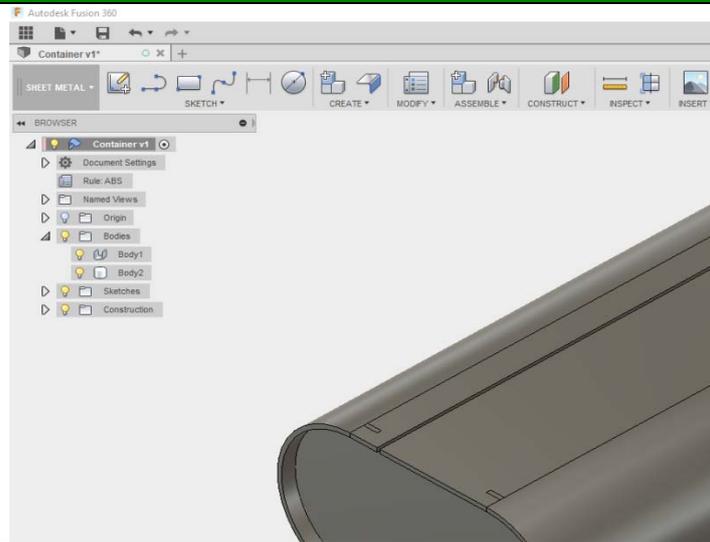


Adding the sides to the Body

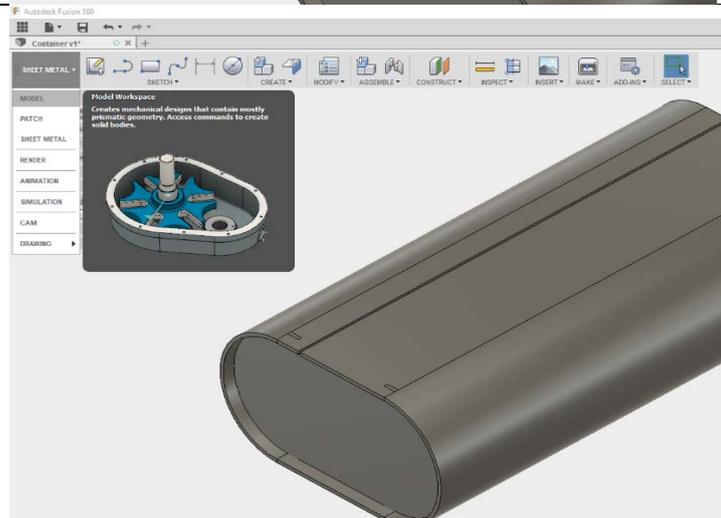
Now turn **Body 1** back on to see the side in place.

You will see the Tabs showing through. These will need cutting out for final machining. We will do this in a second.

Firstly, we need to replicate this side to the other side.



Change the workspace back to the **Model** workspace



Click **Create - Mirror**

When the dialog box appears select the following:

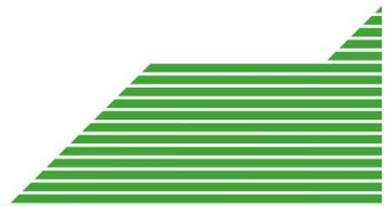
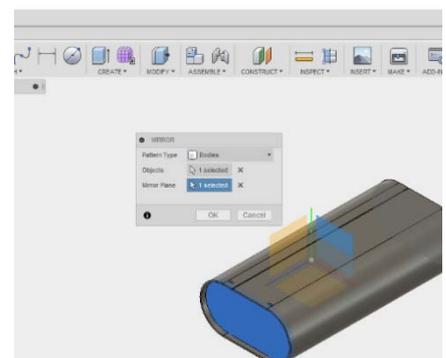
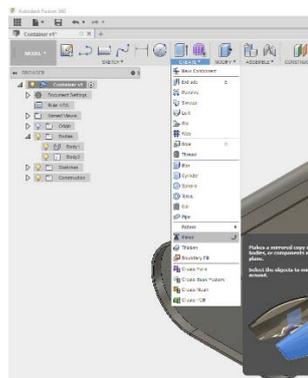
Pattern Type: **Bodies**

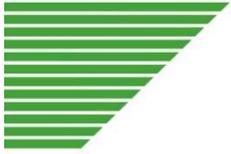
Objects: **select body2 (the side)**

Mirror plane: **select the right origin plane as pictured.**

Click **OK** to accept.

You should now have two sides to your container.





Adding the sides to the Body

Now to cut out the slots in the main body:

Click **Modify – Combine**

Select the following:

Target Body: **Body1 (the main body)**

Tool Bodies: **Body2 and Body3 (the two sides)**

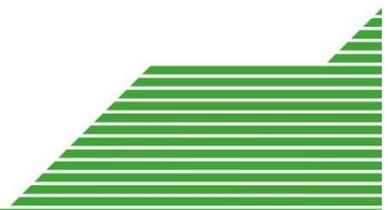
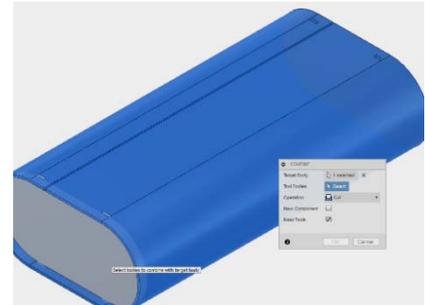
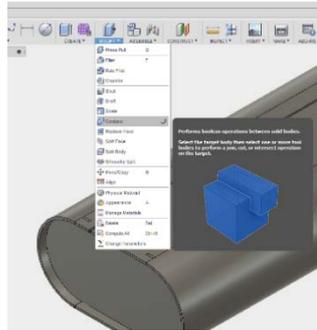
Operation: **Cut**

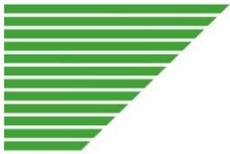
Keep tools: **tick**

Click **OK** to accept.

You will now have cut-outs, which exactly match the side tabs.

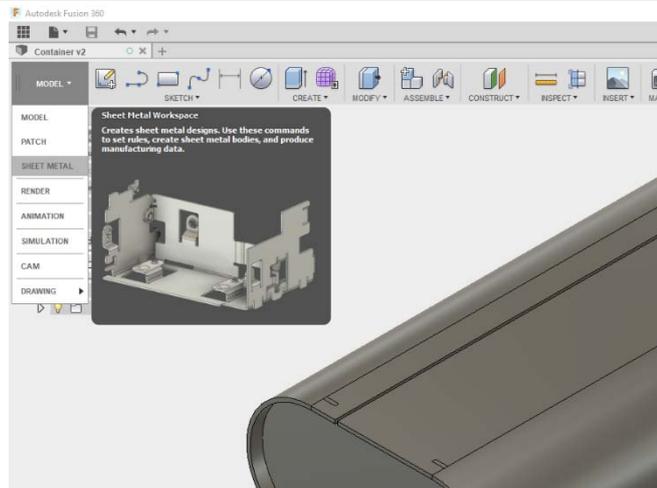
Now **save** your Design.



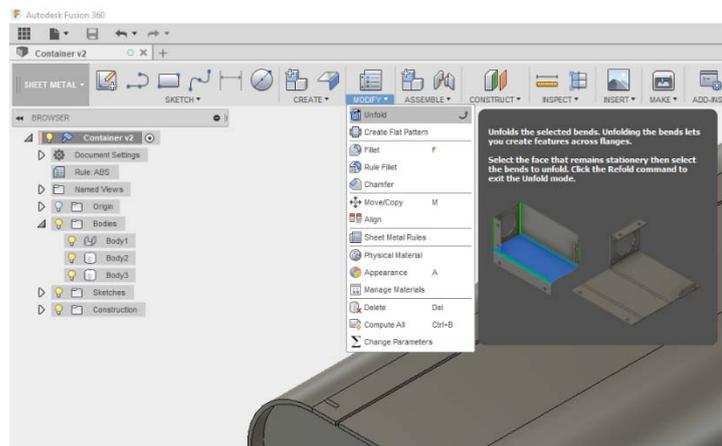


Unfolding your Design

Return to the **Sheet Metal Workspace**



Click **Modify - Unfold**

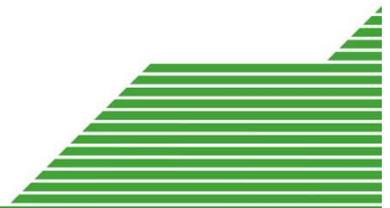
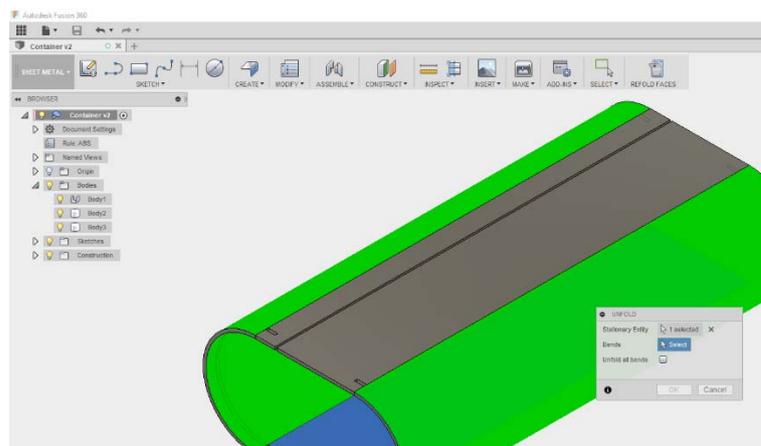


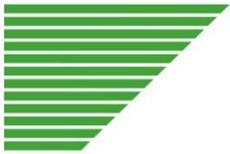
Select the following in the Dialog Box:

Stationary Entity: **Select the bottom surface**
Bends: **click on both curved sections of the body**

Click **OK** to accept.

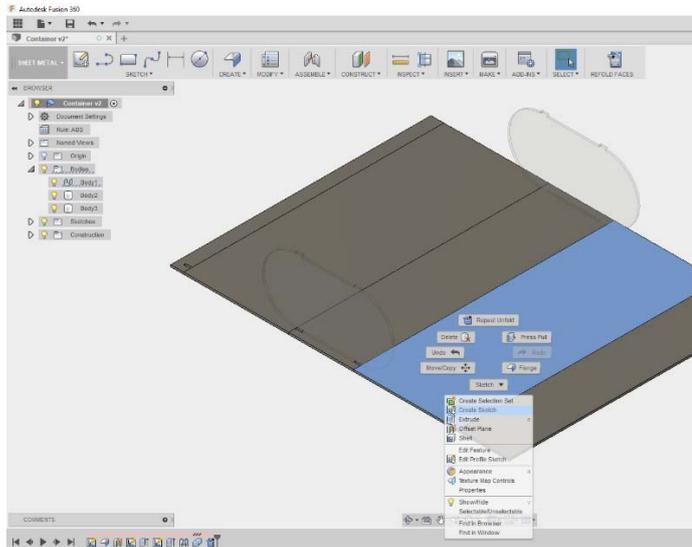
You will now have a net of the body.
We will now work on applying the living hinge to the curved sections.





Designing your Living Hinge

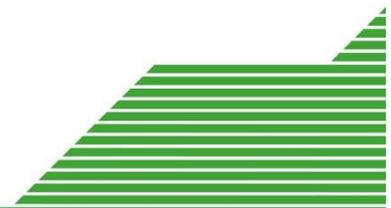
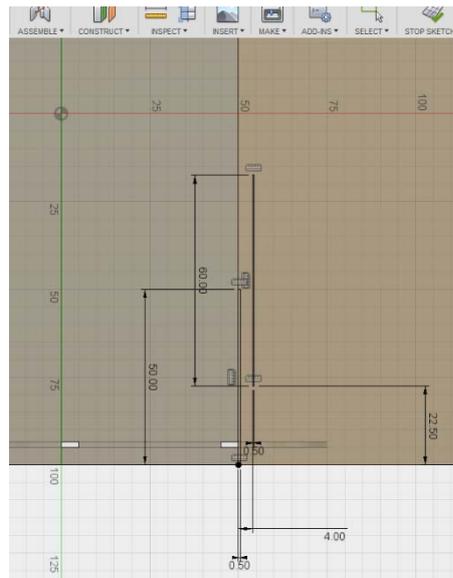
Right click on one of the flattened curved surfaces and click **Create Sketch**

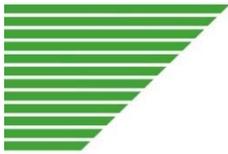


We are now going to draw the first sections of the first two rows of the hinge.

Using the **rectangle** button under the sketch menu, draw a rectangle **50mm x 0.5mm** in the bottom left hand corner of the section. Draw a second rectangle **60mm x 0.5mm** and dimension **22.5mm** from the bottom and **4mm** from the first rectangle.

Click **Stop sketch**





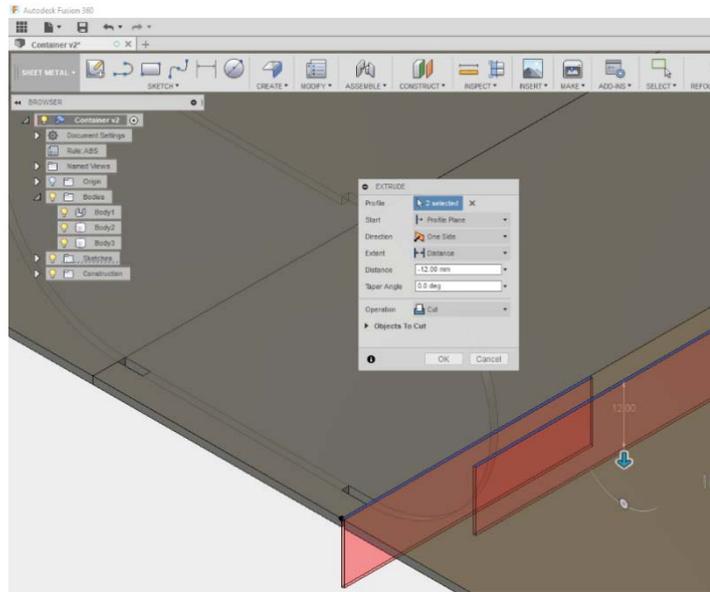
Designing your Living Hinge

Now extrude cut these rectangles by clicking **create - extrude**

Select both rectangles

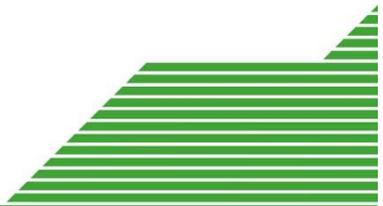
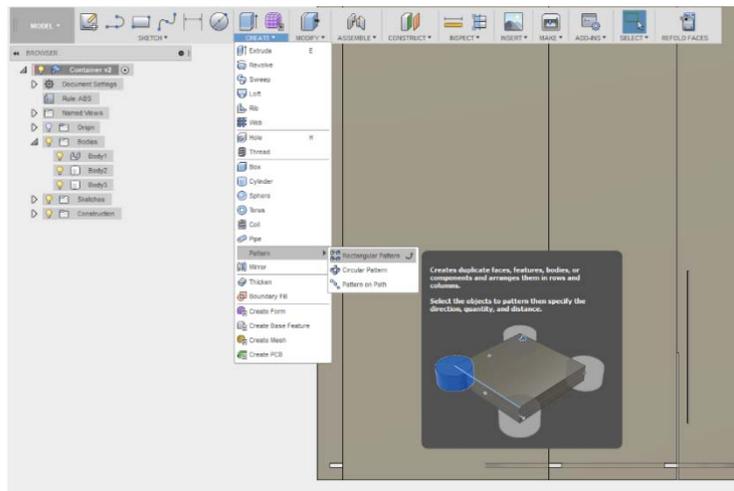
Drag the **blue arrow** down through the body.

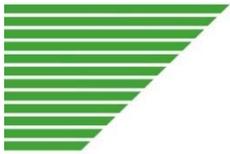
Click **OK** to accept.



Return to the **Model Environment**

Click **Create - Pattern - Rectangular Pattern**





Designing your Living Hinge

Select the following:

Pattern type: **Features**

Objects: **select your extrusion from the timeline at the bottom of the screen.**

Directions: **select a vertical and horizontal edge of the body**

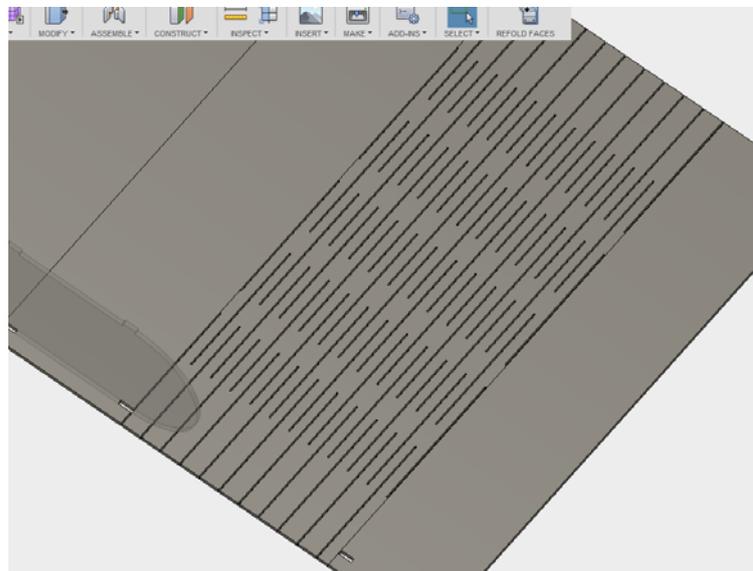
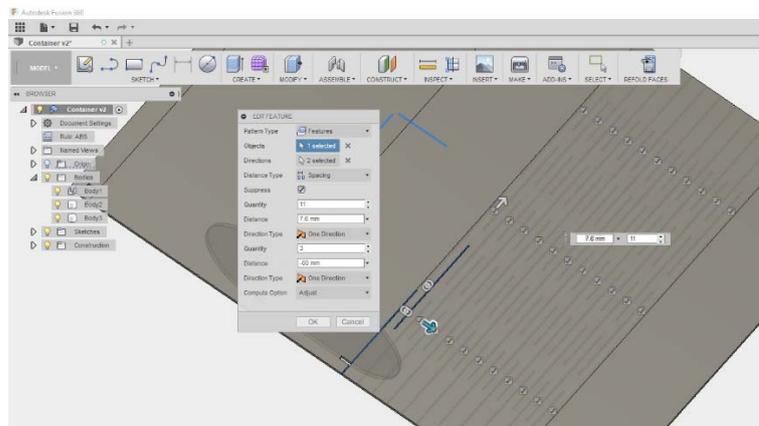
Direction Type: **Spacing**

Now **drag out the two arrows** to the edges of the surface we are adding the hinge to.

In the quantity boxes change to **11 and 3** respectively.

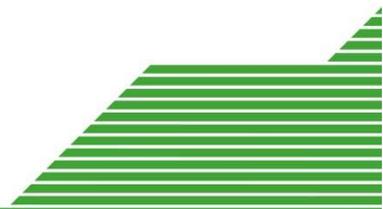
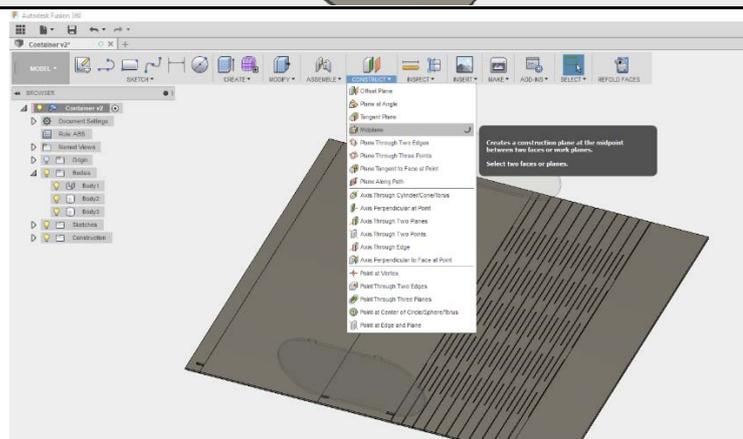
In the **distance boxes**, experiment to ensure your grid fills the entire surface.

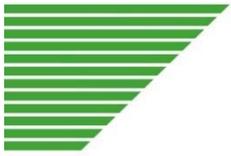
Once happy with the grid and it looks similar to the one opposite, click **OK** to accept.



We will now copy this pattern across to the other curved side. We will be using the mirror function to do this, but will first need to create a plane to mirror this with.

Click **Construct - Midplane**

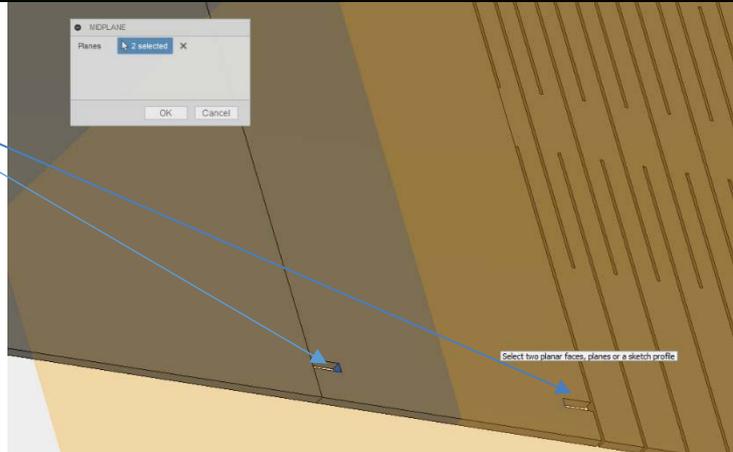




Designing your Living Hinge

Select the two opposite inside faces of the cut-outs from the side part. You may need to toggle off the side to access these surfaces.

Click **OK** to accept.



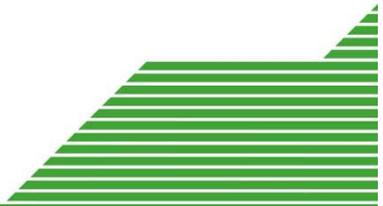
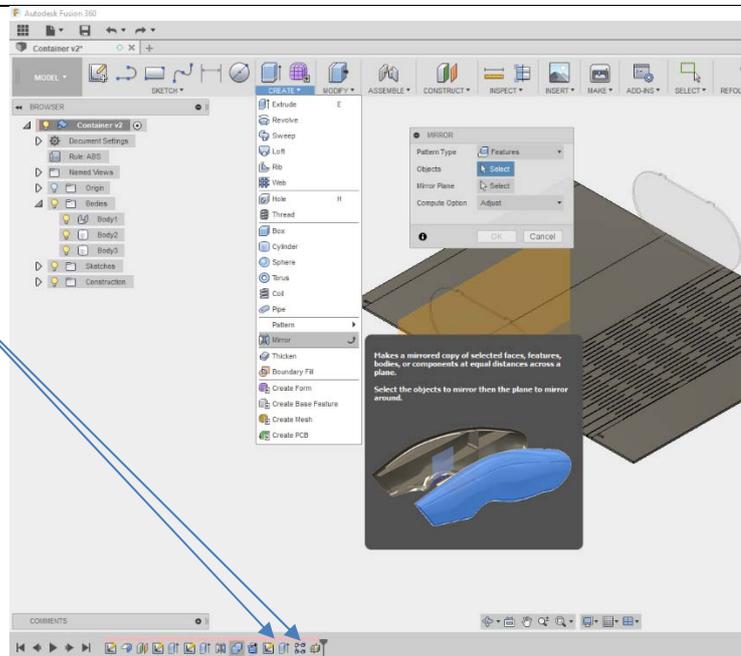
Click **Create - Mirror**

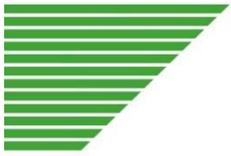
On the dialog box select the following:

Pattern Type: **Features**
Objects: **Select the Pattern and extrusion from the timeline**
Mirror Plane: **select the plane you have just created.**

Click **OK** to accept.

You should now have the grid pattern on the opposite side.



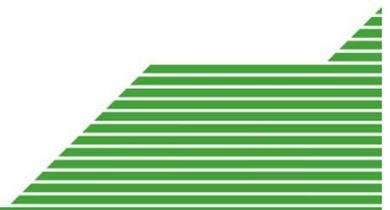
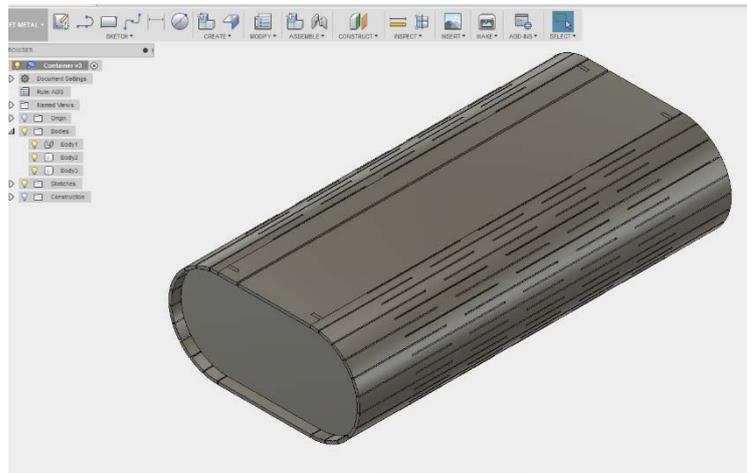
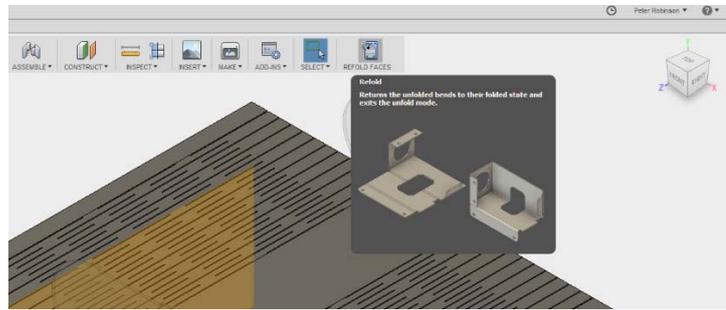


Designing your Living Hinge

Return to the **Sheet Metal Workspace**

Now click **Refold Faces**

Your shape should return to the original 3D shape, but with the Living Hinge.



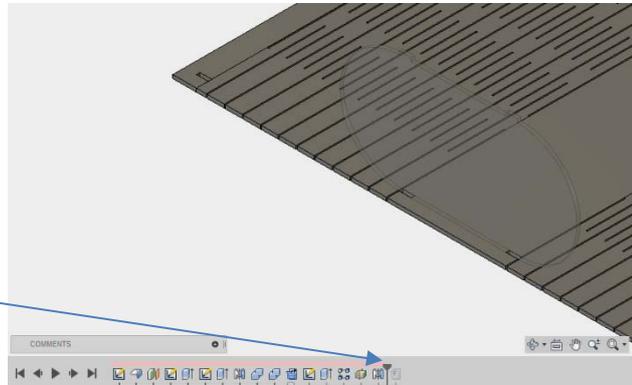


Adding detail to your Design

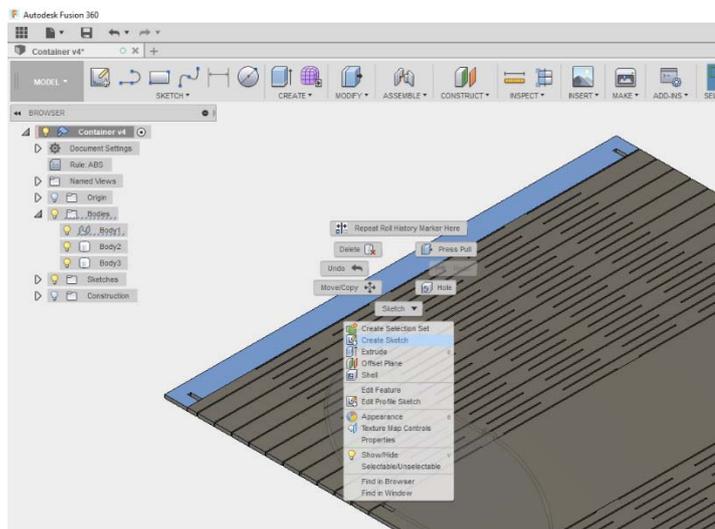
We will now add a cut out for fingers to enter.

Before we do this we need to roll back our history to before we refolded the faces.

Simply **drag the marker** on the timeline before the last function. This will “spread out the shape”.



Select the surface to cut out. In this case, it will be the short edge. **Right click** and select **create sketch**.

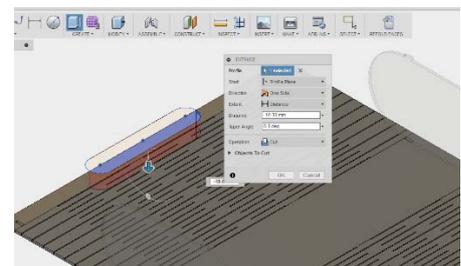
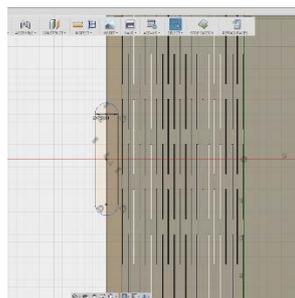


For the cut out I have opted for a Slot. You may wish to use a different shape?

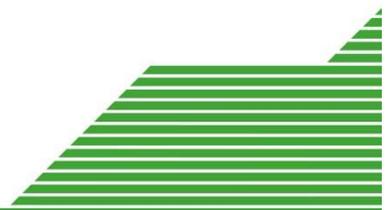
Once you have drawn your enclosed shape, click **Create - Extrude**

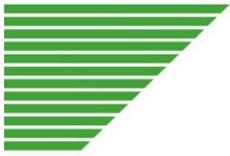
Select the shape area and **Drag** through the part to cut out.

Click **OK** to accept.



At this stage, your box is complete and ready for manufacture. You may wish to add text to the design. However, any text added in this software will not be added to the machining file.



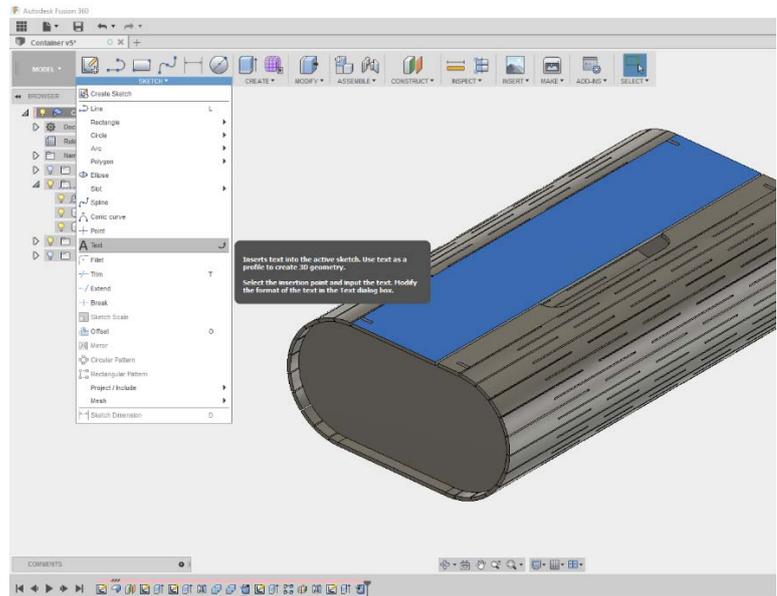


Adding detail to your Design

To add text:

Roll forward the marker on your timeline to the end of the line so the body is back in 3D form.

Now **click the top flat surface** of the body and click **Sketch – Text**.



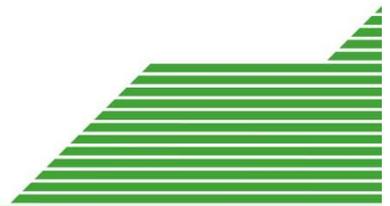
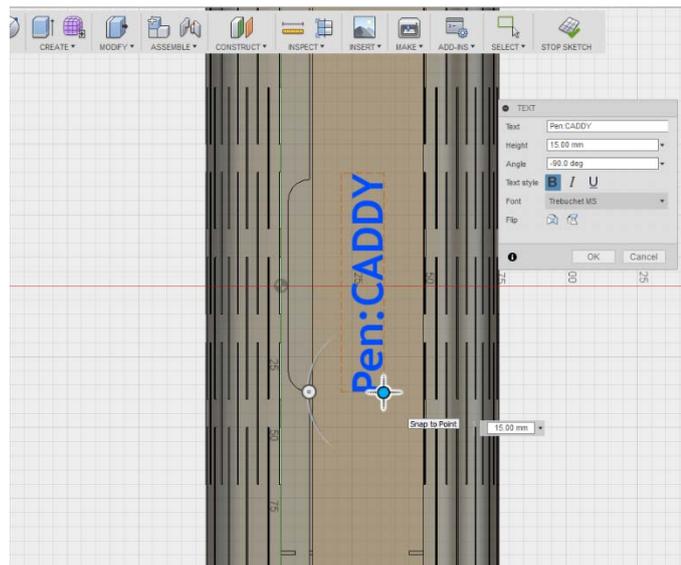
Click in the Text section of the dialog box and **type in the text of your choice**.

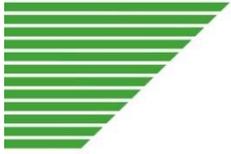
You can edit the height of the text (I have chosen **15mm**).

Change the angle to be in line with your requirements (**-90 deg**)

Choose your font.

Click **OK** to accept.





Adding detail to your Design

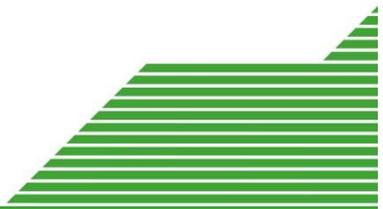
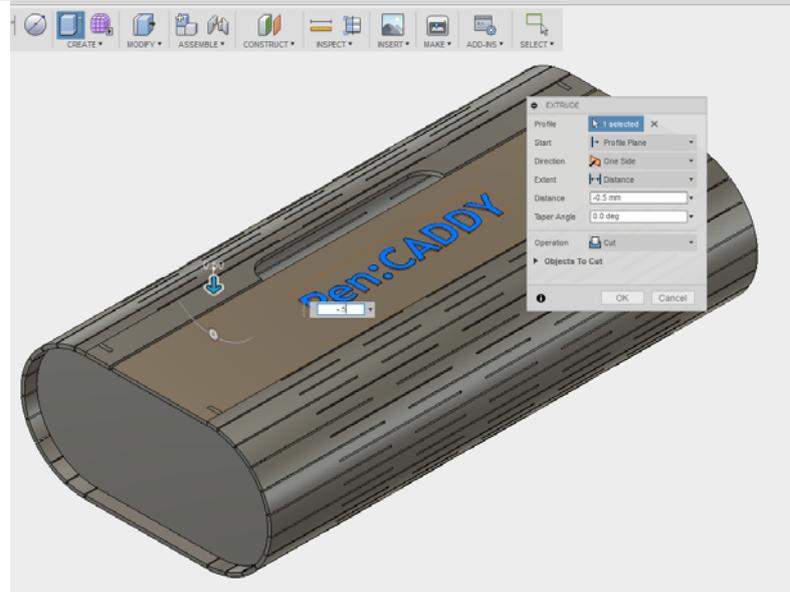
The text is only laid on the top of the surface, so you will need to cut it in by doing the following:

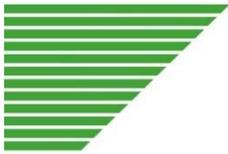
Create – Extrude

Select the text

Cut into the surface by 0.5mm

Click **OK** to accept.

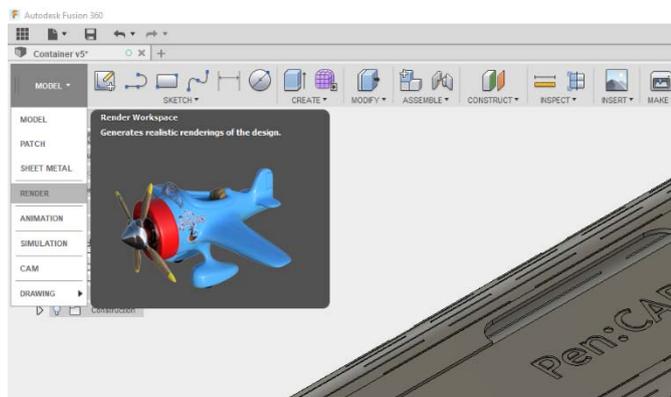




Rendering your design to produce a photo export

To enter the modelling workspace

Click **Model - Render**

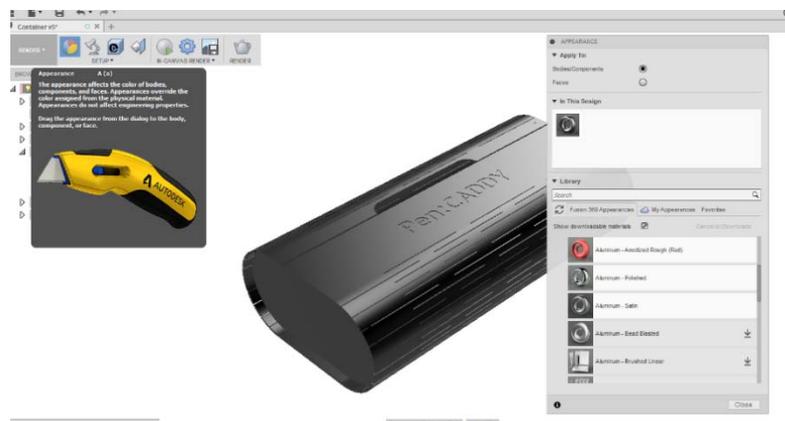


To begin with we will work on the appearance.

Click **Appearance**  This will open a dialog box which will allow you to apply different material looks to different bodies.

Click the arrow next to **metals** and browse for **Aluminium - Satin**.

Drag the thumbnail on to each body and it will apply that material to the surface.



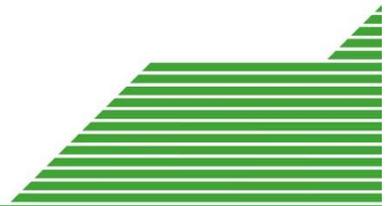
You can also apply different materials to different surfaces, such as in the text, by selecting the **faces** option at the top of the dialog box.

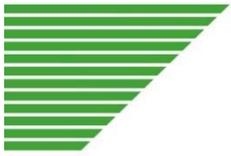
Now browse for **plastic - Opaque** - **Plastic-Glossy (Black)**

Drag this thumbnail on to the individual letters, one at a time.

Apply this also to the inside of the material surface.

Once finished press **Close**





Rendering your design to produce a photo export

Next Press **Scene Settings** 

Here we can set-up how the overall picture looks, such as background, and lighting.

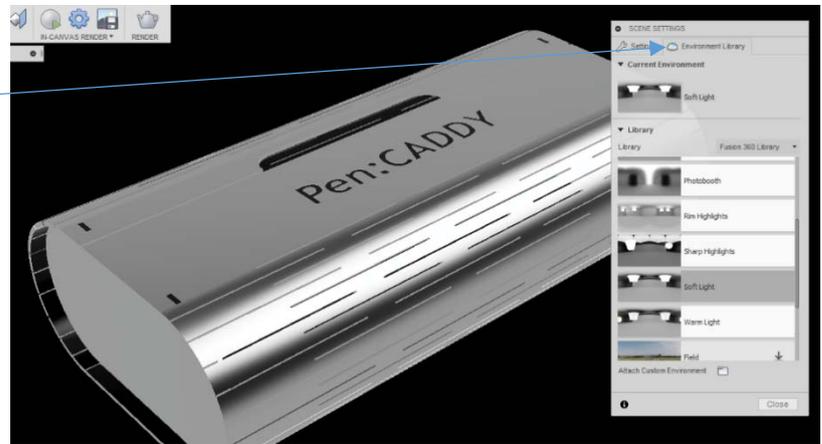
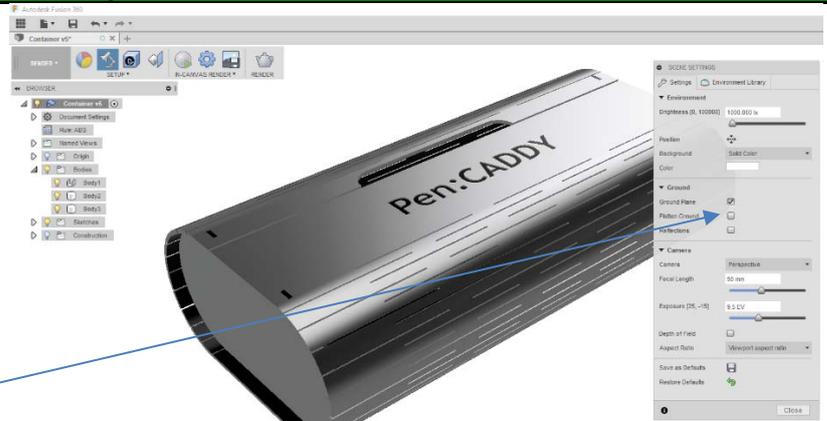
Click on the **Colour swatch** and select **Black**, click **Apply** and **OK**. This will change the background colour to black.

Under the **Ground section**, click **Reflections**.

Next click on the **Environment Library Tab**

Scroll for **Soft Light** and drag the thumbnail on to the background. You will see the lighting change. You can, if you wish, try some of the other lighting profiles here by dragging them in.

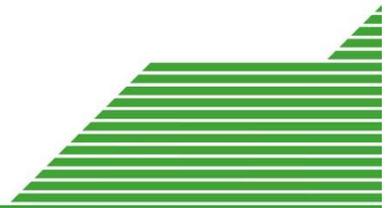
Click **Close** to finish.

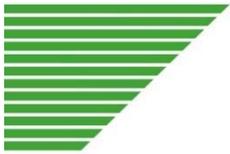


All that is left is to start the render.

Click the **In-Canvas Render**  button. This will start to render your design for presentation.

Once it has completed the render on screen you can press the **In-Canvas Save**  button and save this in a suitable location on your computer.





Rendering your design to produce a photo export

The above method of rendering uses your computer memory and can take some time. There is an alternative method, which is to use Cloud Render.

Click on the **Cloud Render** button, select the size of image you wish to have and click **Render**.

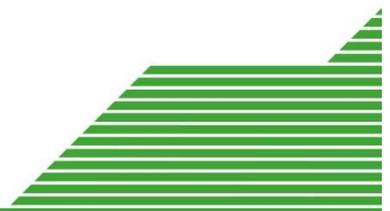
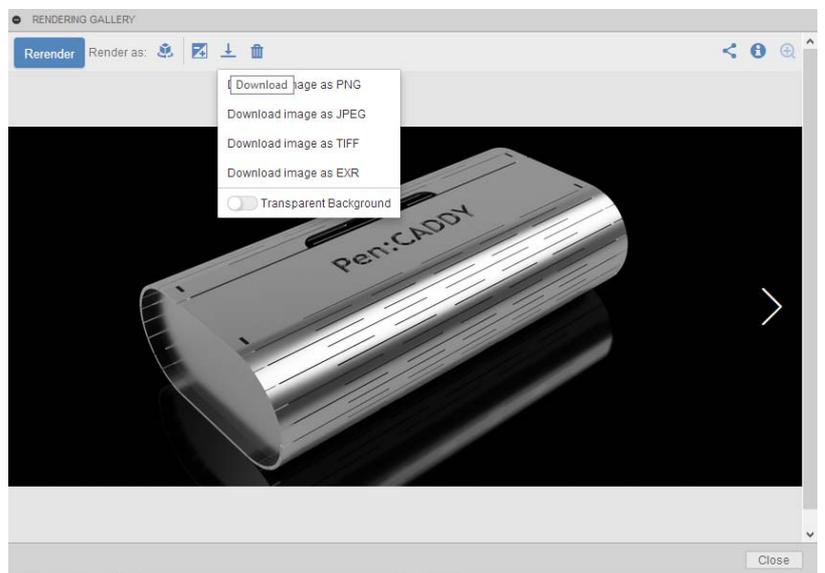
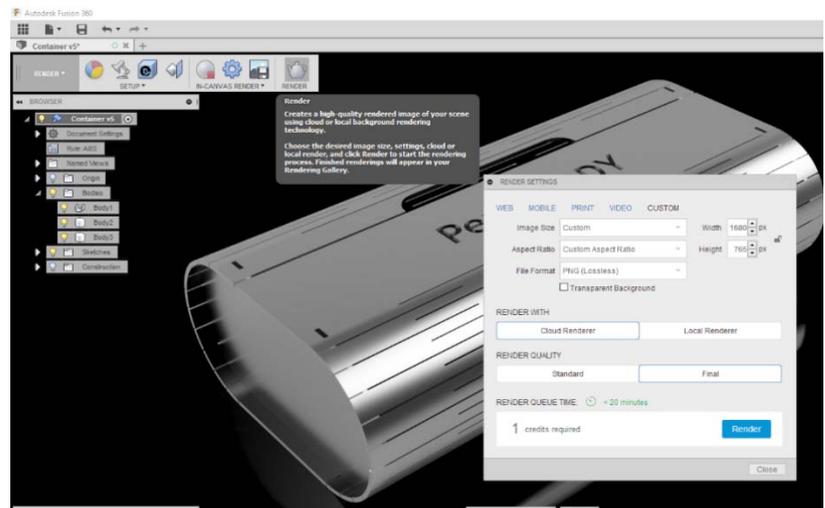
The file will then be sent to Autodesk's Cloud servers to render your file.

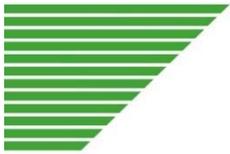
You will then find a thumbnail at the bottom of the render screen. When your file is ready there will be a picture for you to click on.

Simply open that thumbnail and you can **download** that picture to your computer.

Press **Close** when downloaded and return to the **Sheet Metal Workspace**.

Note: The cloud rendering keeps working when you are not logged on the computer. It is therefore good practice to use this function at the end of the day, and pick up the next day.

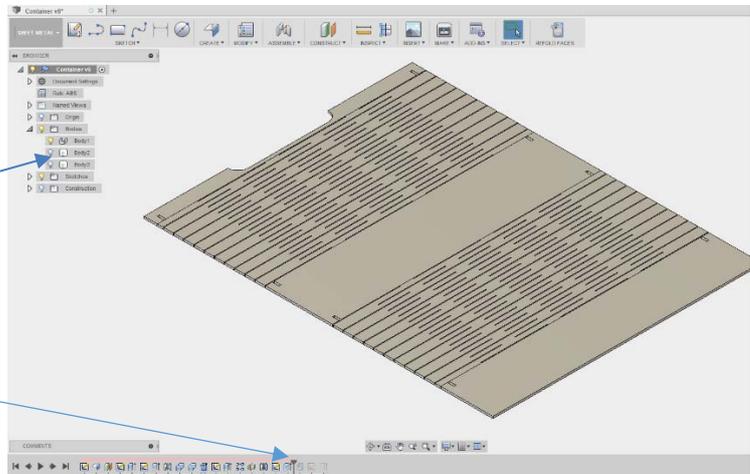




Exporting your design for Manufacture

You cannot export the 3D body to the Laser Cutter, so we need to create a sketch of the surface to export as a DXF and import into LaserCAM.

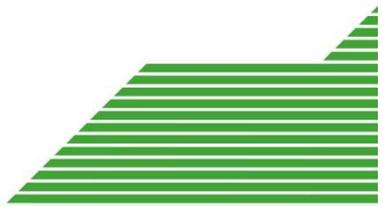
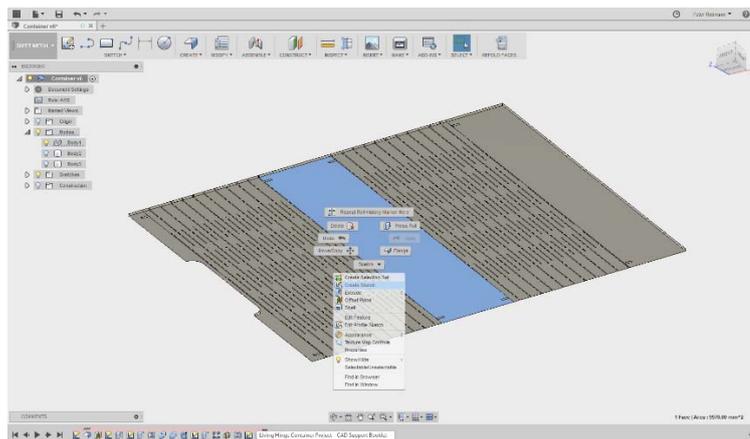
In the sheet metal workspace, **Toggle off** the two side piece bodies and **roll back** the marker on your history timeline so the net is laid flat.

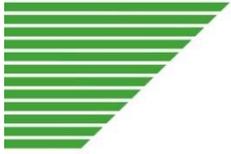


Move the view cube so you can see the underside of the body.

Click on a flat surface and Right Click to access the floating menu.

Click **Create Sketch**





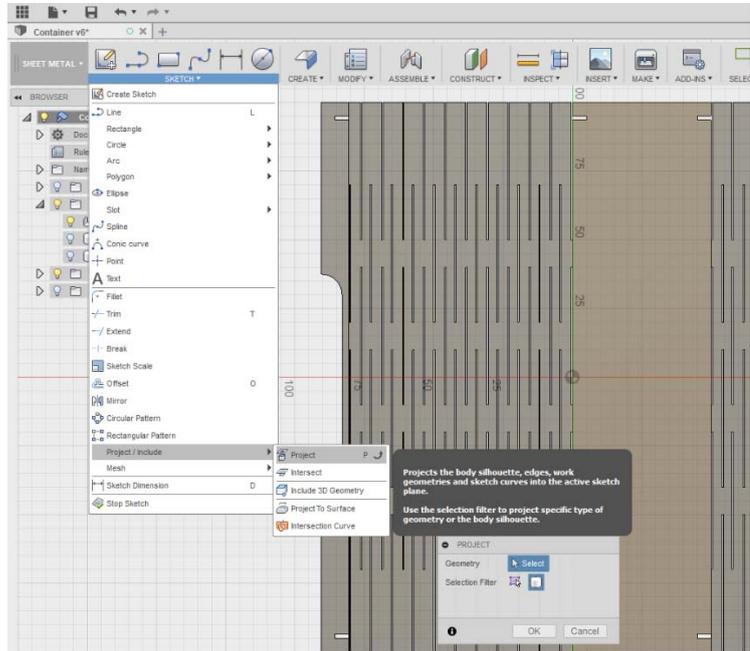
Exporting your design for Manufacture

Click **Sketch - Project/Include - Project**

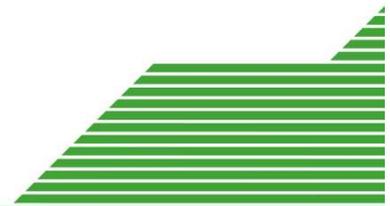
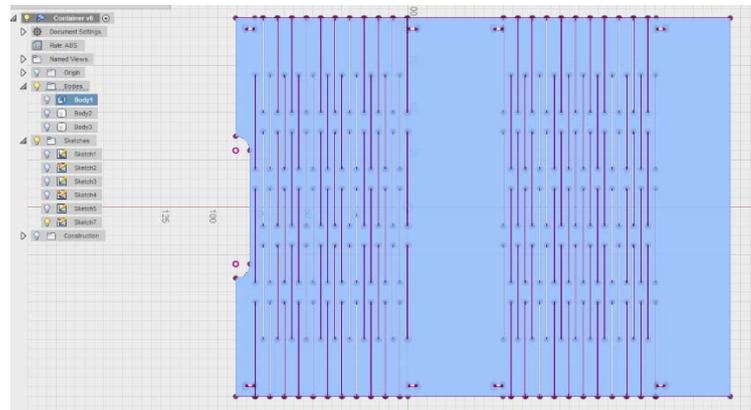
In the Dialog box ensure the selection filter is set to **Bodies**

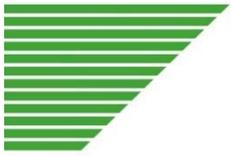
Now **click the Body**

Click **OK** to accept.



Toggle off the main Body. You should now see the sketch you have just created.





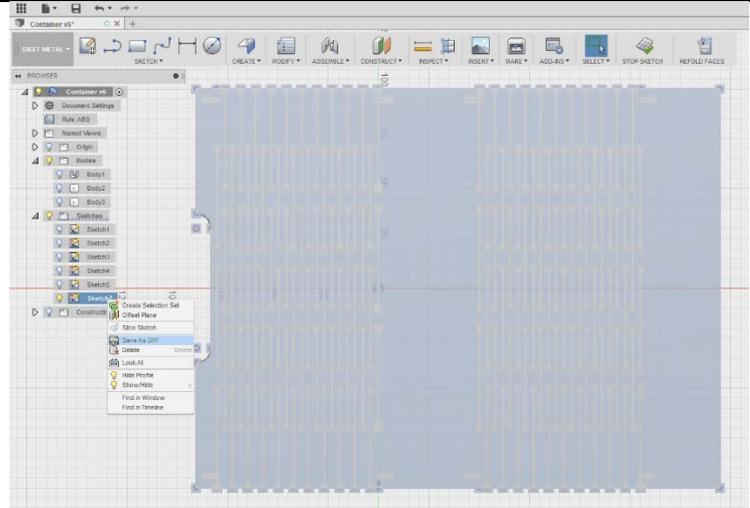
Exporting your design for Manufacture

On the Browser tree on the left hand side, **expand** the Sketches tree and you will find your current sketch with the bulb on.

Right click on that label and click **save as DXF**

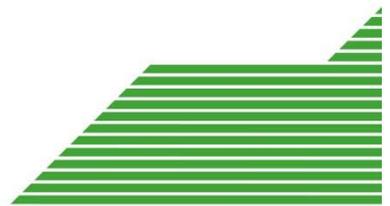
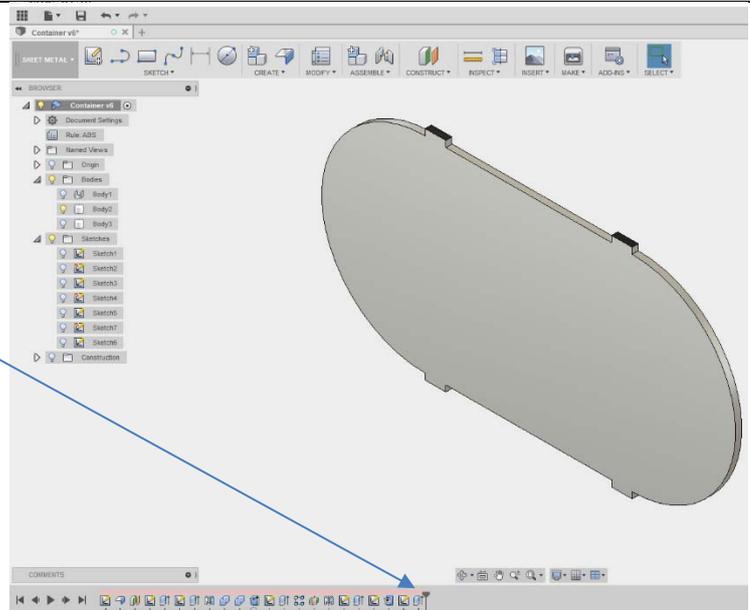
Save in a suitable location using a filename you can easily remember such as **NAME_maincasebody.DXF**

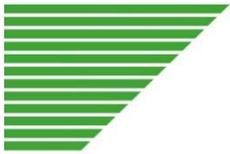
Now click **Stop Sketch**



We now need to export the side profile as a DXF File too.

Roll forward the timeline to the end and toggle off the last sketch and toggle on one of the side bodies

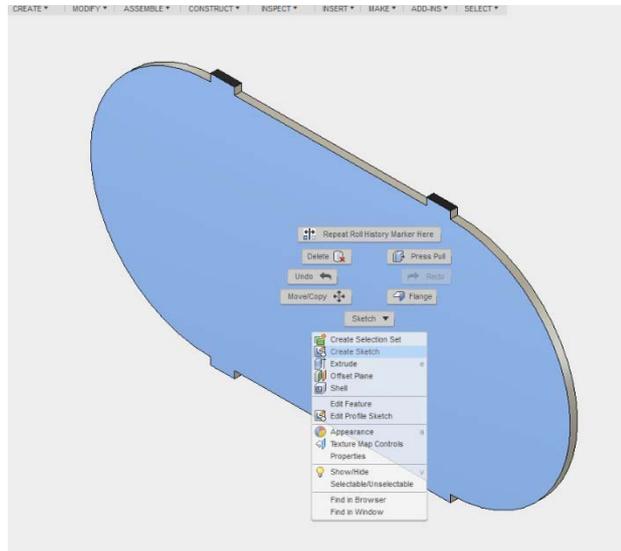




Exporting your design for Manufacture

Right click on the front surface.

Click **Create Sketch**

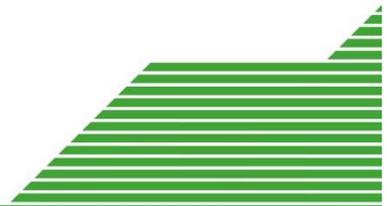
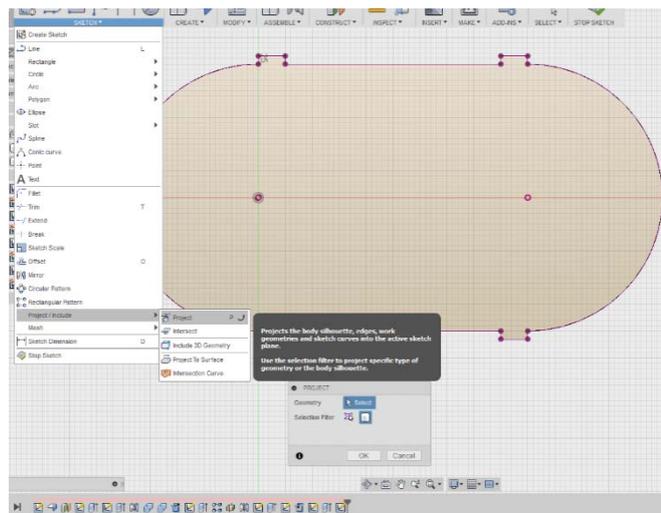


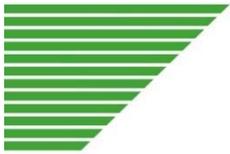
Click **Sketch - Project/Include - Project**

In the Dialog box ensure the selection filter is set to **Bodies**

Now click the **Body**

Click **OK** to accept.





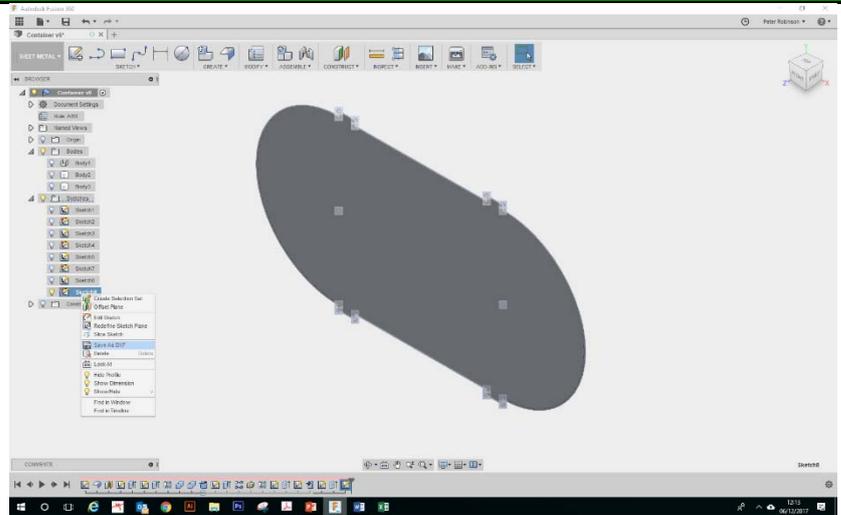
Exporting your design for Manufacture

On the Browser tree on the left hand side, **expand** the Sketches tree and you will find your current sketch with the bulb on.

Right click on that label and click **save as DXF**

Save in a suitable location using a filename you can easily remember such as **NAME_sidebody.DXF**

Now click **Stop Sketch**



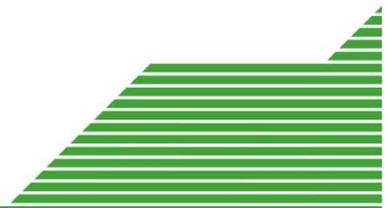
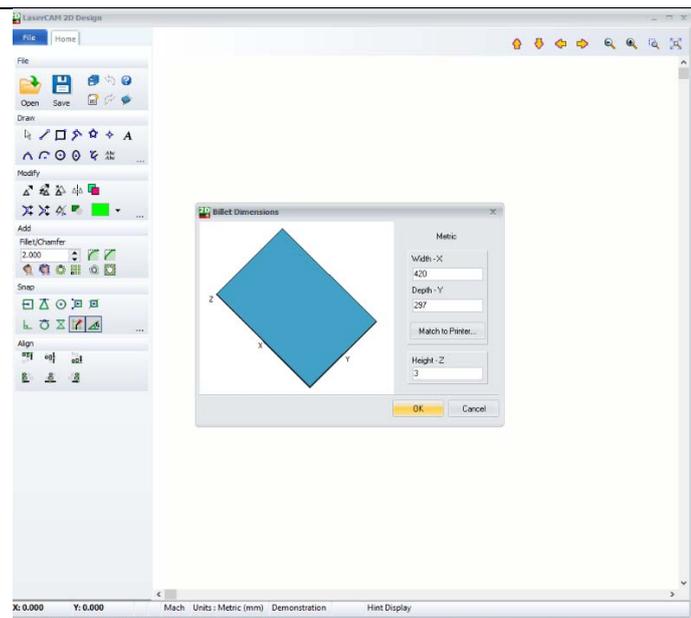
Now Open the program **"LaserCAM 2D Design V2"**

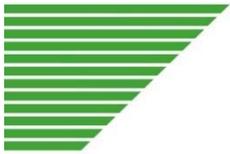
Name	Date modified	Type	Size
LaserCAM 2D Design V2	15/11/2017 20:21	Shortcut	3 KB
QuickCAM PRO	22/11/2017 14:43	Shortcut	3 KB
VR Milling V5	22/11/2017 14:58	Shortcut	3 KB

As soon as it opens, it may give you a "Tip of the Day". Just **cancel** the box.

You will now have a screen which says Billet Dimensions.

Change to the following dimensions:
Width: **420**
Depth: **297**
Height: **1.5**

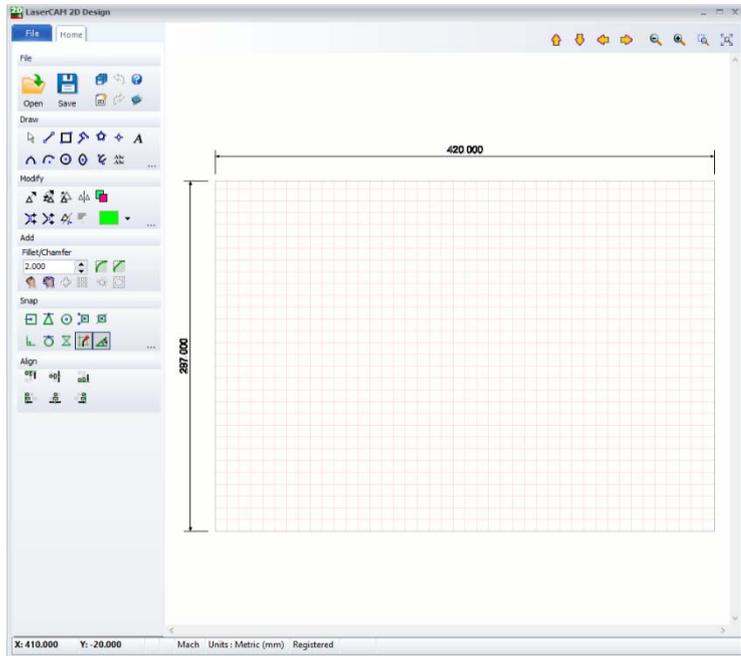




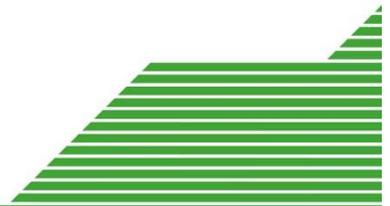
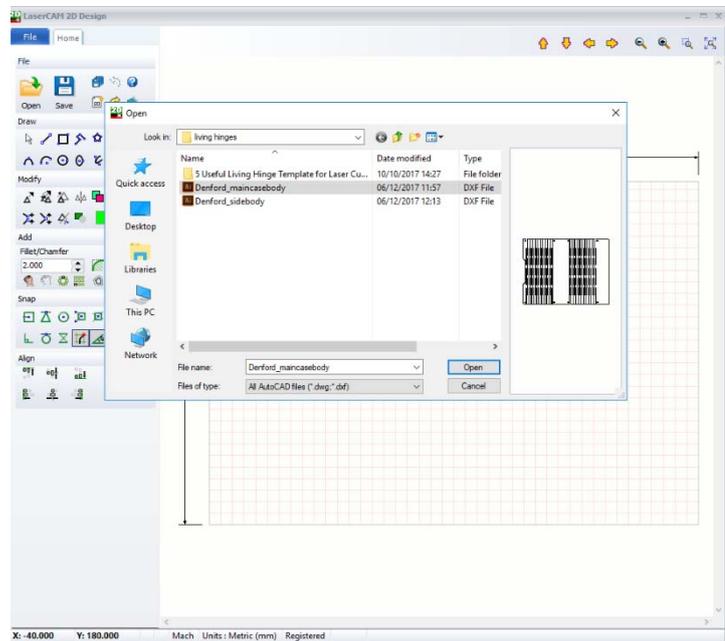
Exporting your design for Manufacture

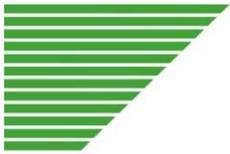
You will now be presented with this screen.

Now click “file” and “import”



Browse for the **YOURNAME_mainbody.dxf** file you exported from Fusion 360 and click **open**.



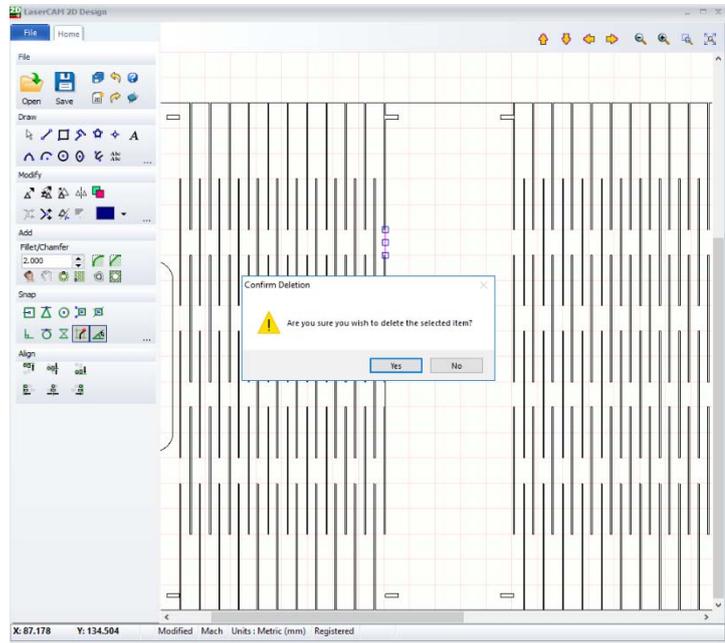


Exporting your design for Manufacture

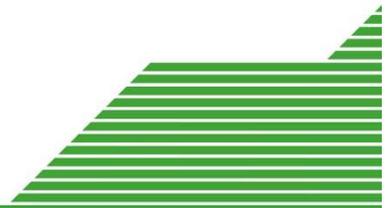
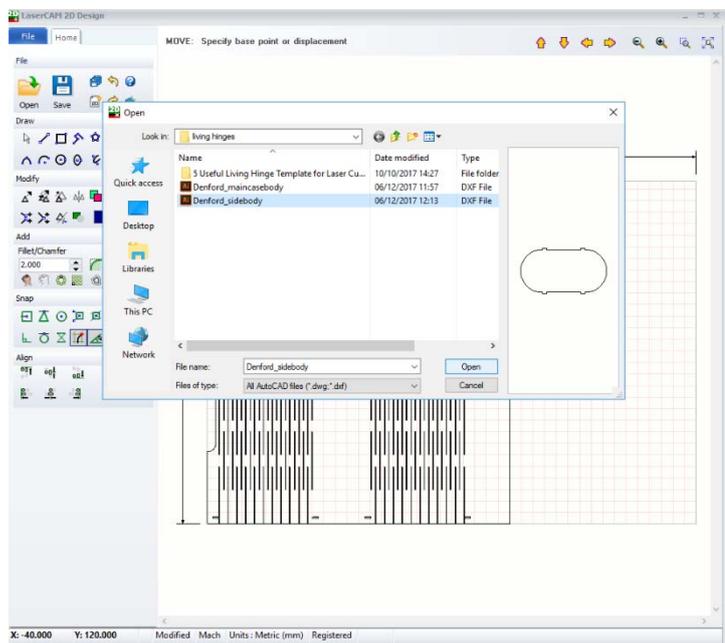
You should now have an exact replica of the sketch you made on Fusion 360 on your screen.

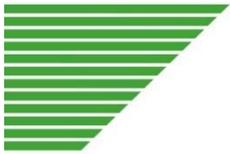
You may find that Fusion has exported some odd construction lines. You can select these and simply **delete** them.

Note: It may be necessary to right click and “explode” lines, so you do not delete the lines you wish to cut.



Browse for the **YOURNAME_sidebody.dxf** file you exported from Fusion 360 and click **open**.



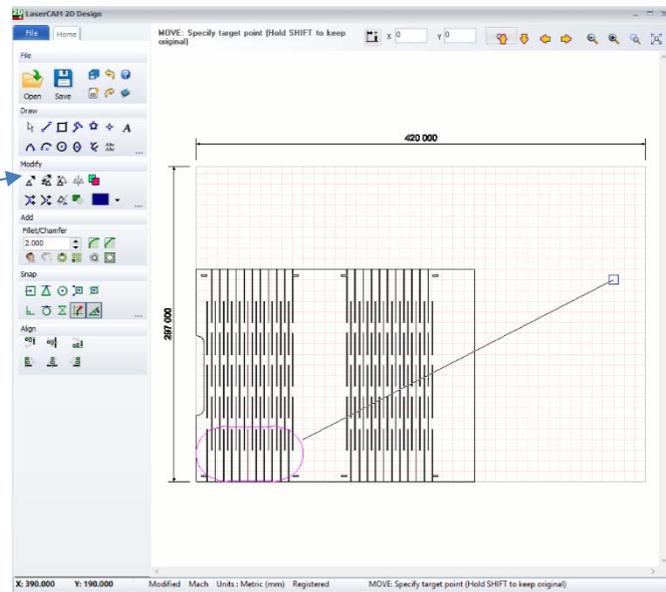


Exporting your design for Manufacture

The side body will import into the same area as the main body.

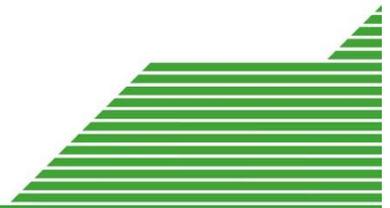
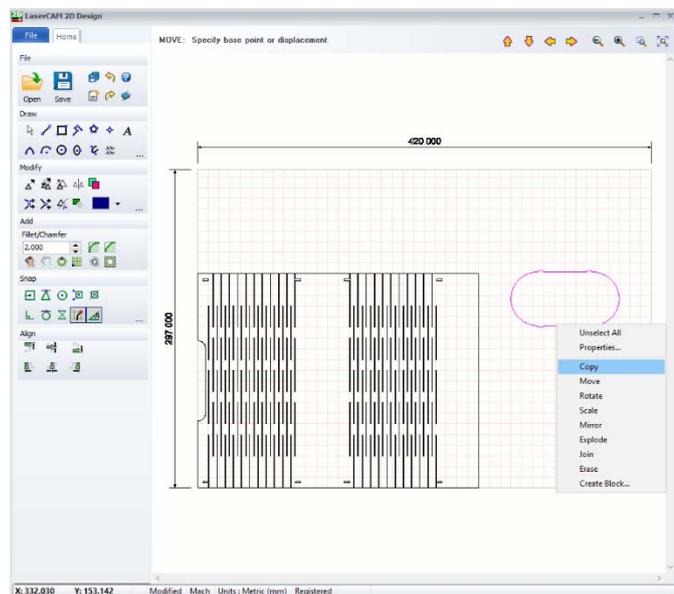
Click **Modify - Move**

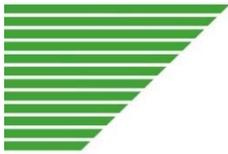
Select the pink sketch and click to another blank part of the sheet. It will then move the sketch.



We will need two side pieces, so **right click** on the side piece and click **Copy**

Click on your mouse to place the object.





Exporting your design for Manufacture

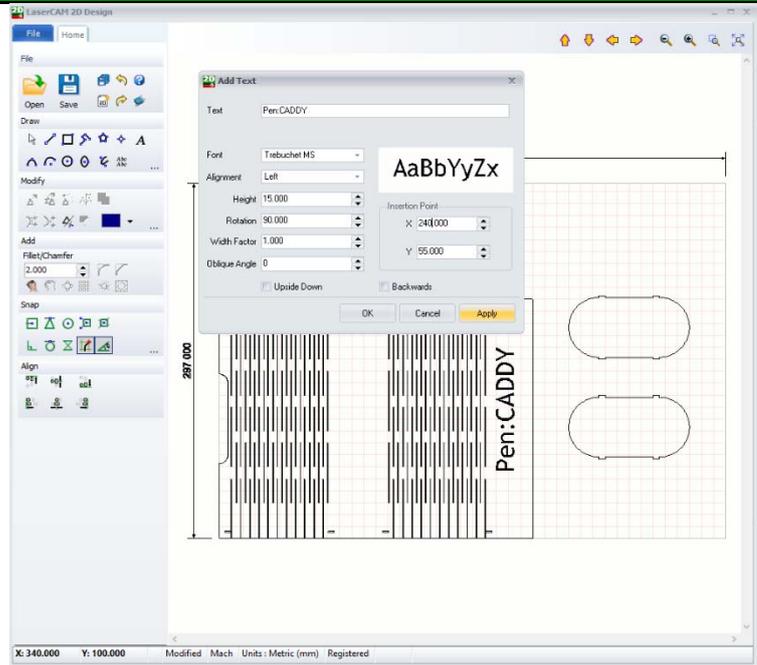
Next we are going to add the text logo onto your design.

LaserCAM has a **text tool**  using this tool, add your logo to your design to be engraved.

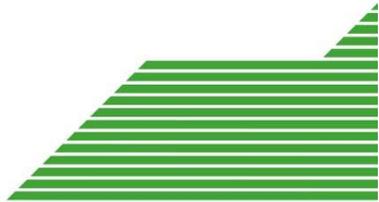
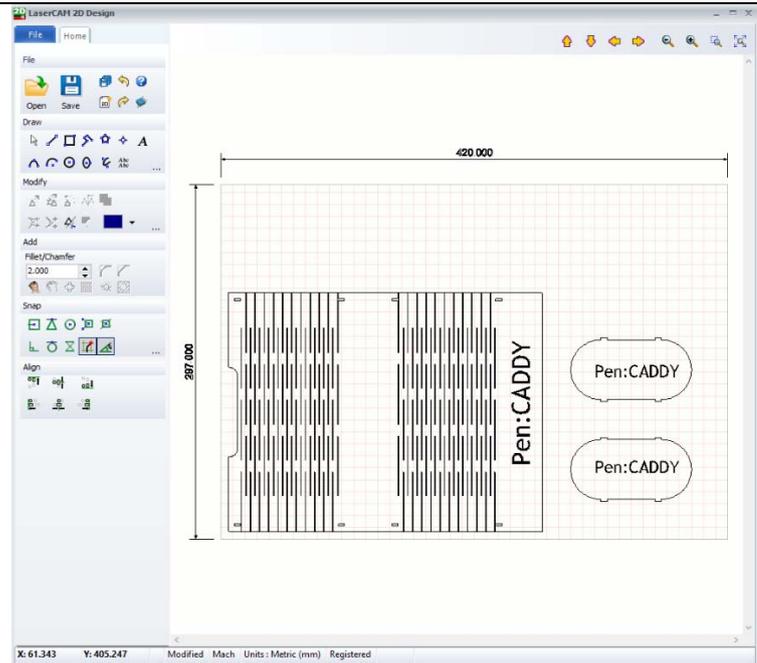
Click **Apply** to check position and edit as necessary.

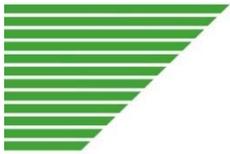
Click **OK** to accept.

If you wish to move and change your text after you have clicked OK there are blue manipulators you can use to scale or rotate your text.



Add any extra text you wish to have on the design.





Exporting your design for Manufacture

Next, we need to set up how each of the lines would be used on the Laser Cutter - i.e. which ones would be engraved and which would be cut. The Laser Cutter reads RGB colours to instruct it how to "print/cut" your design.

Red

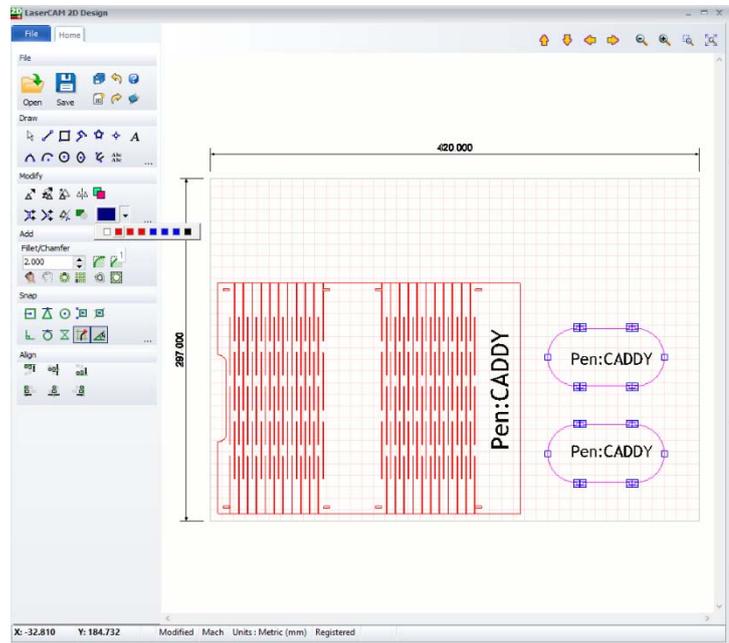
Red is for vector cutting. This is for thin vector lines only.

Blue

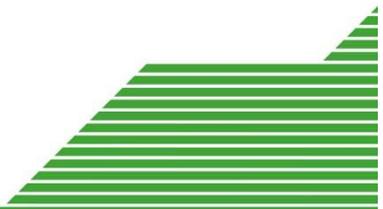
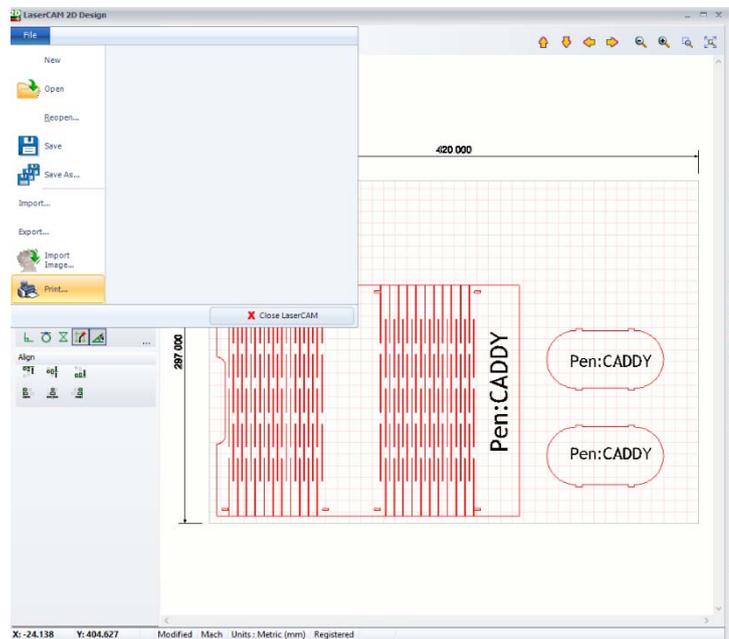
Blue is for vector engraving. This is for thin vector lines only.

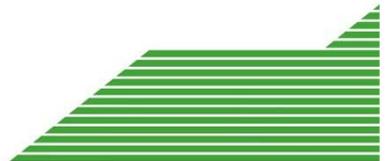
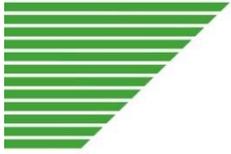
Black

Black and greyscales are for raster engraving.



The design is now ready for output to the VLS Series Laser





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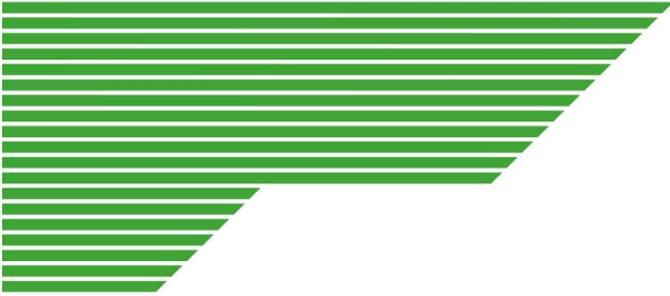
Denford Limited Armytage Road Brighouse West Yorkshire HD6 1QF England



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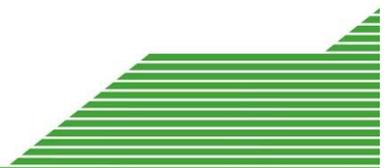
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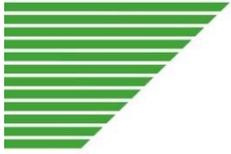
CAD/CAM Projects

Living Hinge Container Project



TEACHER SUPPORT GUIDE





Denford: Living Hinge Container Project – VLS Series Laser Cutter

Teacher Support Guide

This Project takes the form of the design and manufacture of a saleable product. Students are tasked with designing and manufacturing a container using living hinges to hold a product from a range given. It is expected that the product will be glued, assembled and will be a saleable item. The final product will be designed using CAD and will be manufactured on a VLS Series Laser Cutter using an aluminium-effect ABS laminate.

Students have one sheet of A3 1.5mm ABS aluminium-effect laminate sheet to make their designs from, so they will need to ensure that all of the parts for their designs will fit on the sheet.

As a guide, students should not make their containers any larger than 300mm x80mm x60mm.

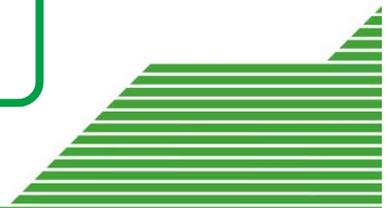
Project Timeline Summary

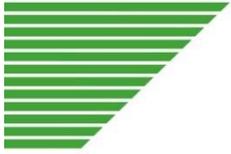
It is anticipated that learners will spend a total of about 8 hours in producing the work for this project. Learners will be expected to have had previous experience in using Autodesk Fusion 360.

Throughout this project, learners will need to apply problem-solving and designing skills developed in earlier years, in order to achieve a successful outcome.

Session	Focus	General Content
1	Understanding the Problem	Analysis of the Design Problem and understanding the context of their product.
2	Designing	Producing a concept sketch idea of their storage container.
3	Developing the Solution	Modelling of the idea and producing a final 3D sketch and layout sketch.
4	CAD Design	Using the sheet metal function in Fusion 360, generate a model of the design which can be “flat packed”.
5	CAD Design	Produce a rendered output of the file as a design concept.
6	CNC Post Processing	Export DXF files of all parts for use on the LaserCAM software.
7	Manufacture, Finishing & Assembly	Cut the final design out on the laser cutter, sand and finish edges, assemble and glue the final product.
8		

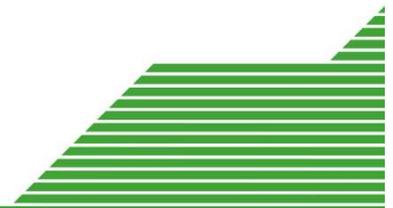
Note: This project uses AutoDesk Fusion 360. However, the project can be adapted to be used with other 3D Design Software Packages.

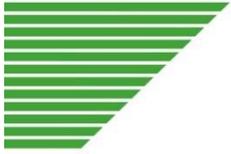




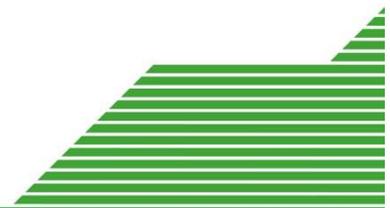
Project Delivery Session Detail

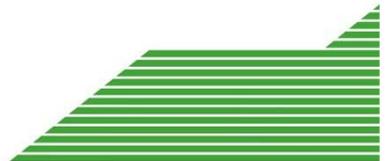
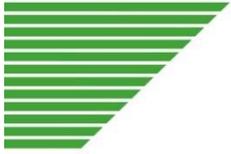
Session	Focus	Session Content
1	Product Research & Design	<p>Ask students to read through the design problem, then, working individually, ask them to visit the Alessi website and make rough sketches of products they like the look of. Ask them to annotate any design features they particularly like.</p> <p>Now tell students to look at the key information page and talk them through how living hinges work.</p> <p>Introduce students to the Design Considerations - Primary Research page. Students have the option of four different products to choose from. Once chosen, ask students with the same products to group together.</p> <p>Introduce the overall size of the product and explain how the product will be manufactured.</p> <p>Ask students to start some primary research as a group, listing the size and shape of the products they will be storing. Get them to note this down on the sheet (this may need to be finished at home).</p>
2	Designing	<p>Students will need to draw a concept idea.</p> <p>Encourage the students to keep to basic shapes when designing, but remind them they will be using living hinges as part of their design to “wrap around” their product.</p> <p>Once complete, ask students to discuss/present their idea to another group, and note any relevant feedback on to their own sketch.</p> <p>Note: To fasten the product, students can choose a mechanical tab, as shown on the example. Alternatively, they can use magnetic tape, along with steel washers for the tape to stick to. These would need to be glued on to the product.</p>





Session	Focus	Session Content
3	Developing the Solution	<p>Students will now test their idea, by drawing and cutting out their design idea on A3 Paper.</p> <p>Once happy with their net, ask students to produce a final “quick” sketch and layout of the idea.</p>
4	CAD Design	<p>Students will use the next two sessions to draw their idea in Fusion 360.</p> <p>Students are encouraged to draw their design whilst working through the CAD Support Booklet for this project, where there is a space for the design to be drawn. However, students will need to independently draw their own idea using the same principles.</p>
5		<p>By the end of the two sessions, students should have a 3D model ready for manufacture and rendered drawing.</p>
6	CNC Post Processing	<p>Explain to the students the difference between DXF files and STL files and which CNC machines utilise each.</p> <p>Students will need to export DXF’s of “faces” of their model. For my example, there is one main face and two identical side faces.</p> <p>Once exported, ask students to import these into LaserCAM and prepare the drawing for manufacture on the Laser Cutter</p>
7	Manufacture, Finishing & Assembly	<p>The next two sessions will be used to programme and cut out objects on the Laser Cutter.</p>
8		<p>Once machined, students will need to glue their models together using ABS glue and masking tape to hold until set.</p>





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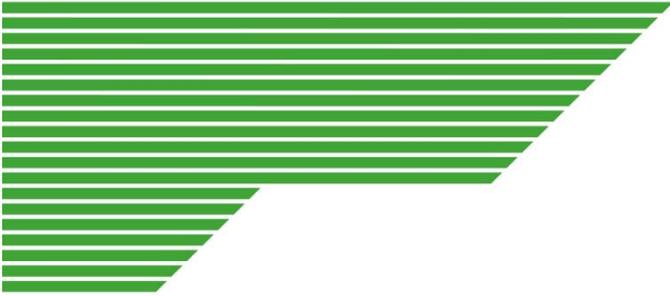
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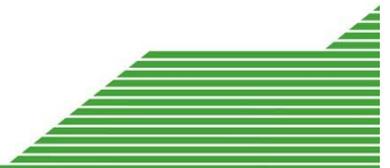
CAD/CAM Projects

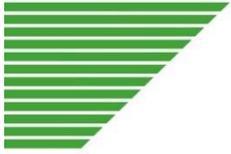
Passive Speaker Dock Project



Project Duration: 8 Hours

STUDENT WORKBOOK





Denford: Passive Speaker Dock Project - Denford CNC Router Student Workbook

DELIVERY PLAN (HOURS)

1. Product Analysis & Design
2. Designing & Developing a Final Solution
3. CAD Design
4. Create the Cutter Paths
5. Post Processing
- 6-8. Manufacture, Assembly & Finishing

DESIGN PROBLEM

Students are tasked with a project to design and manufacture a passive speaker phone/MP3 player dock.

The Design Emphasis will focus on designing to ensure good acoustics.

MACHINE

Denford CNC Router
Router Tooling Package -
B100846

MATERIALS

2 x 160mm x 100mm x 20mm
Maple Wood Blocks -
B103509G Class Pack of 50

ADHESIVE

PVA Glue

FINISHING MATERIALS

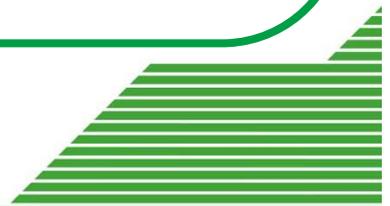
Glass Paper for finish
Wax to finish

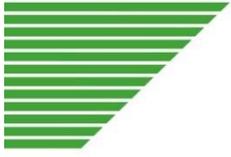
EXTRA INFORMATION

The product is one complete unit and will be manufactured using the Denford CNC Router in two separate halves, then glued together to complete the dock.

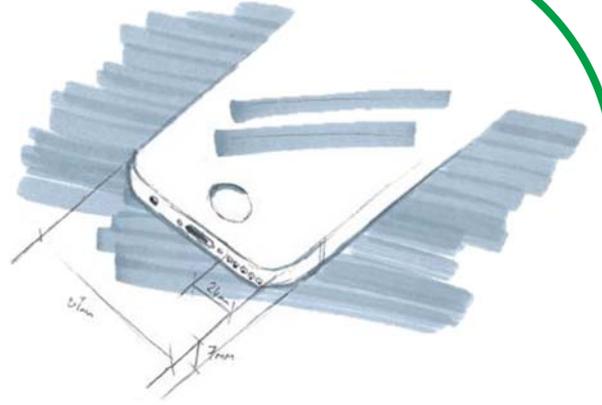
It is expected that the dock will be finished with wax. However, there are opportunities to embellish the design with accent colours.

IMPORTANT - *When machining you can only machine one face of each block. This MUST be considered in your designing.*





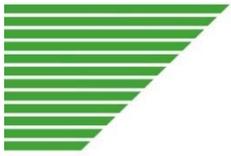
Where does your sound come from?



Take a look at your mobile phone and MP3 player. Do they have an inbuilt speaker? If so, where is it?

What are the critical dimensions of your device? Show your findings above.

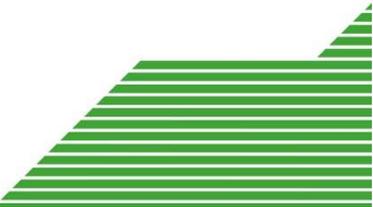




What's out there already?

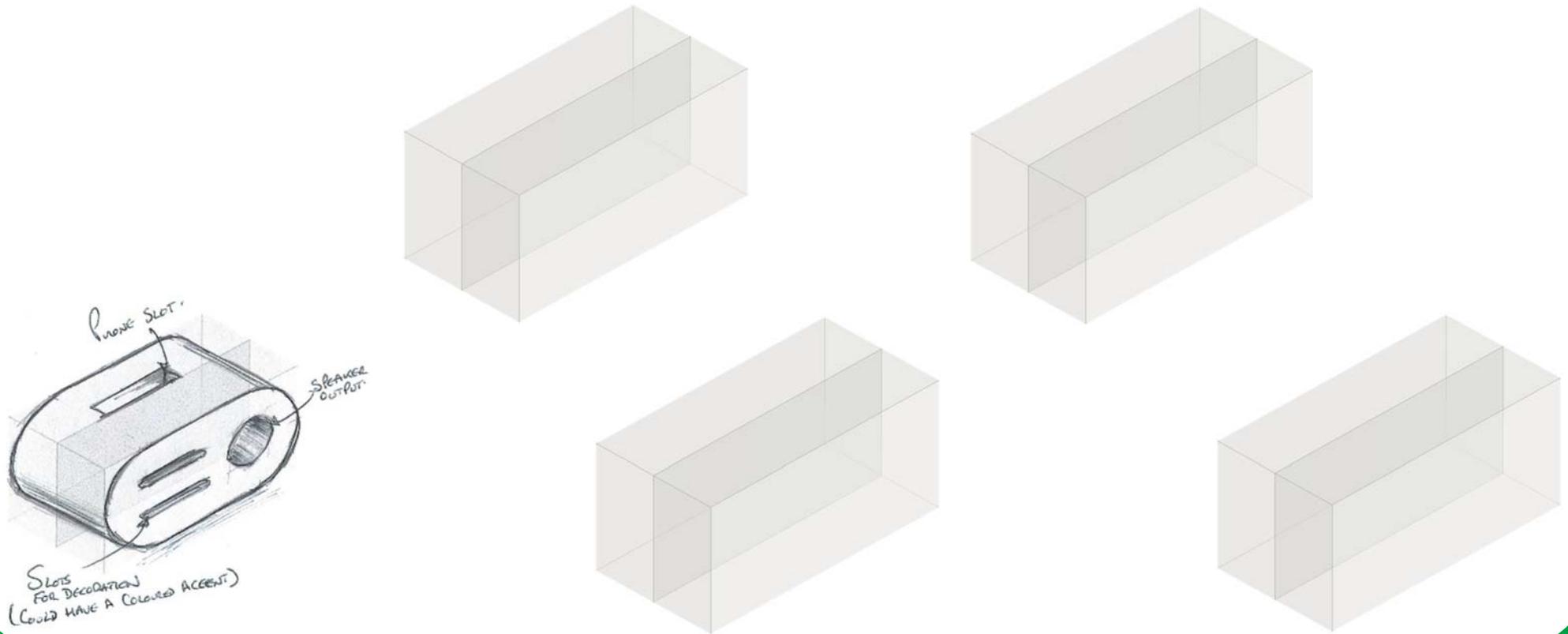


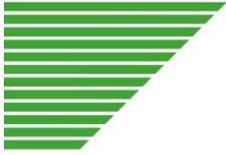
Above are research images that are associated with the function of this product. These are to help you in your designing.



Design

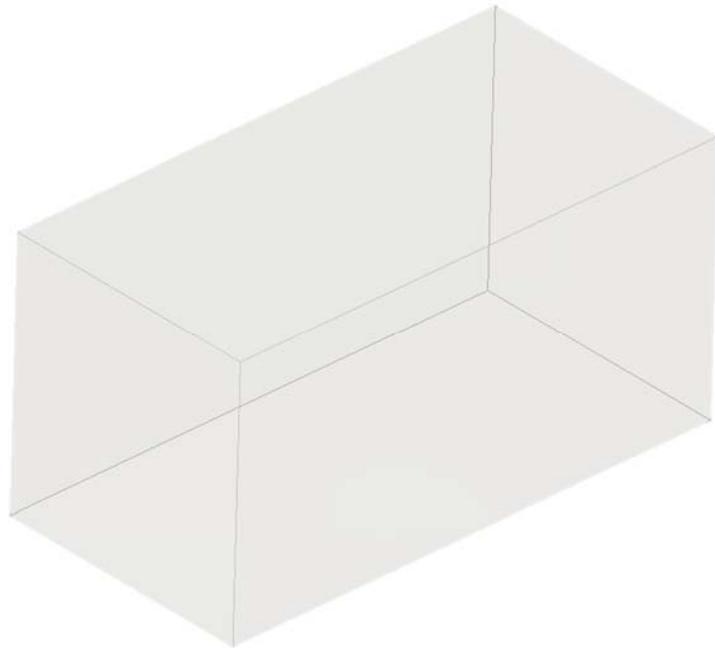
Using 3D Sketching generate some ideas to resolve the problem.
Sketch your ideas over the shaded blocks like the example below. Add annotation to help explain them!



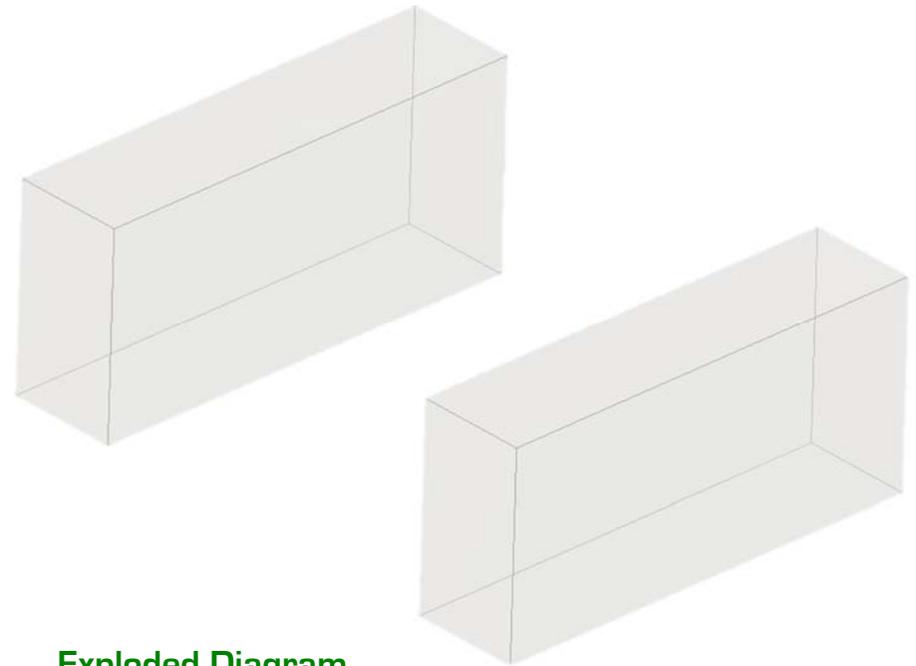


Design Realisation

Bring your favourite idea forward from the previous page and produce two separate sketches, one final 3D sketch, and one exploded sketch showing the two halves of the dock to be manufactured. Add dimensions to help with your CAD drawing



3D Sketch

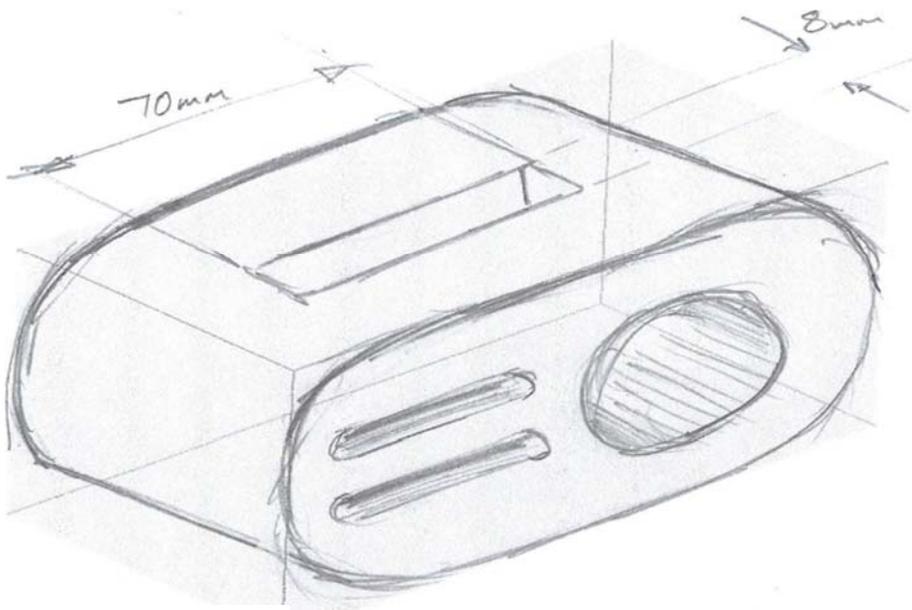


Exploded Diagram

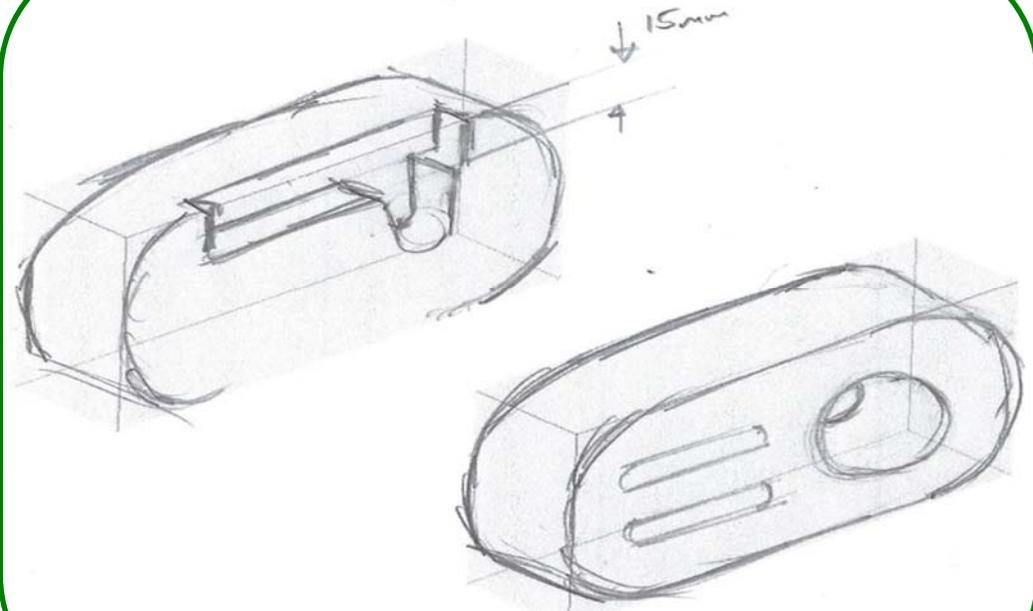


Design Realisation - Worked Example

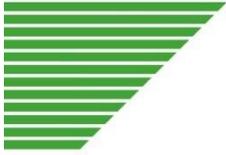
Bring your favourite idea forward from the previous page and produce two separate sketches, one final 3D sketch, and one exploded sketch showing the two halves of the dock to be manufactured. Add dimensions to help with your CAD drawing.



3D Sketch



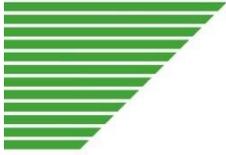
Exploded Diagram



Your Final CAD Design

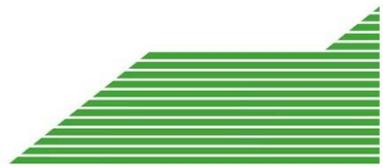
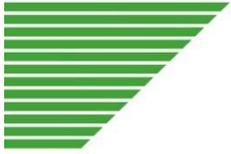
Export a final copy of your CAD Design and attach to this sheet.





CAD Design – Finished Product Example





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