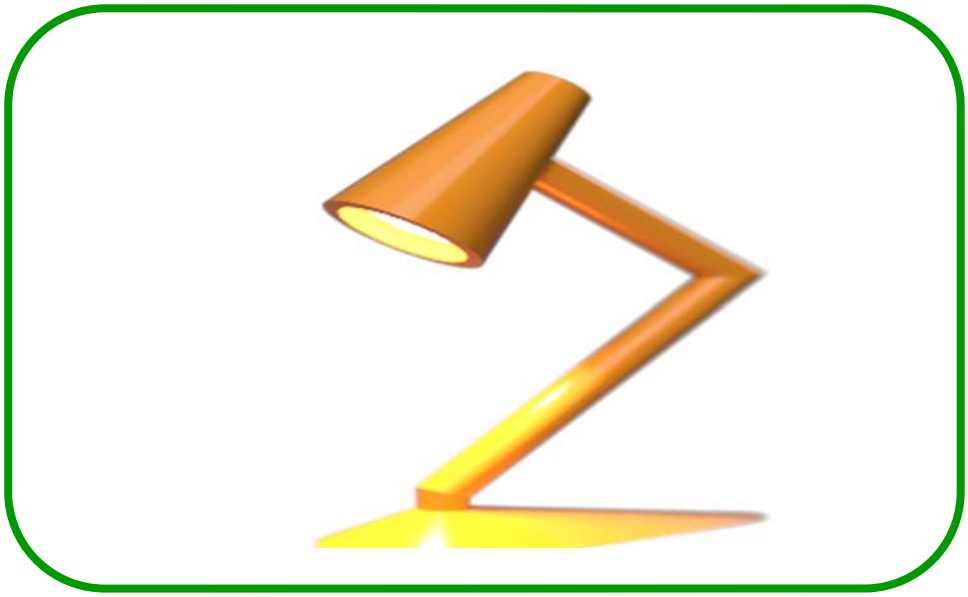
DENFORD®

CAD/CAM Projects

USB Lamp Project



CAD SUPPORT BOOKLET

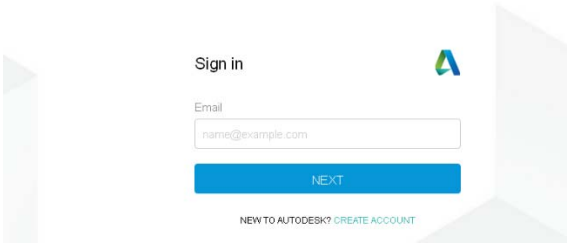
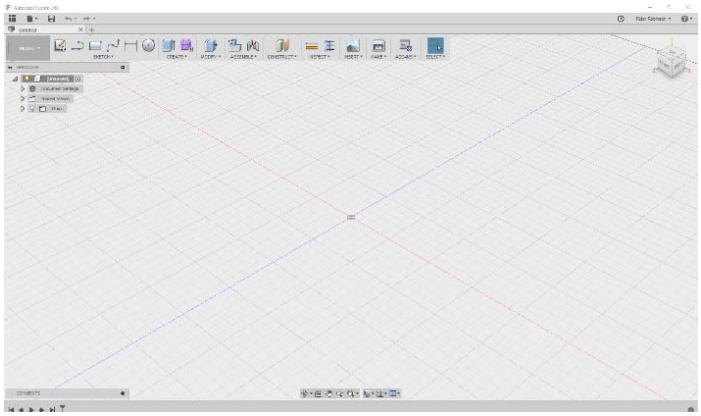
Denford: USB Lamp Project - Denford CNC Router/Vacuum Forming Machine

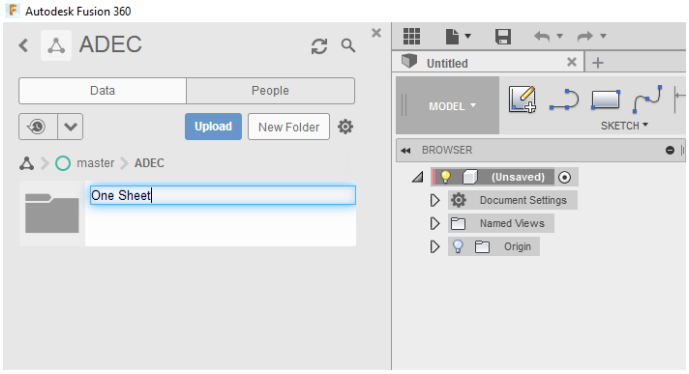
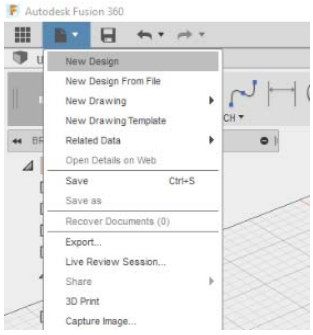
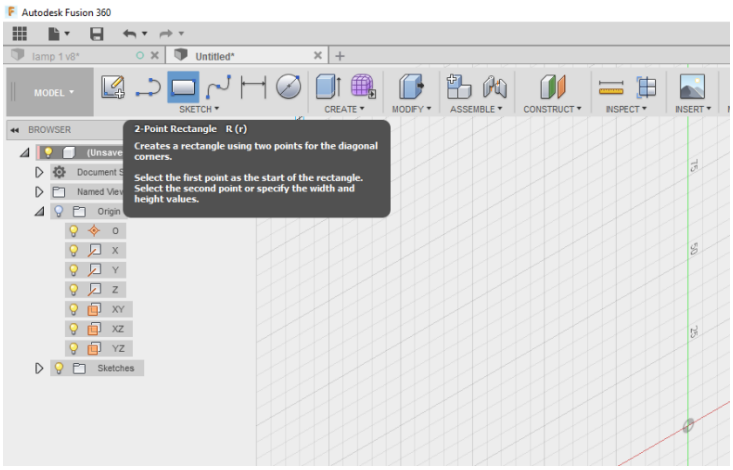
CAD Support Booklet

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

The process will be as follows:

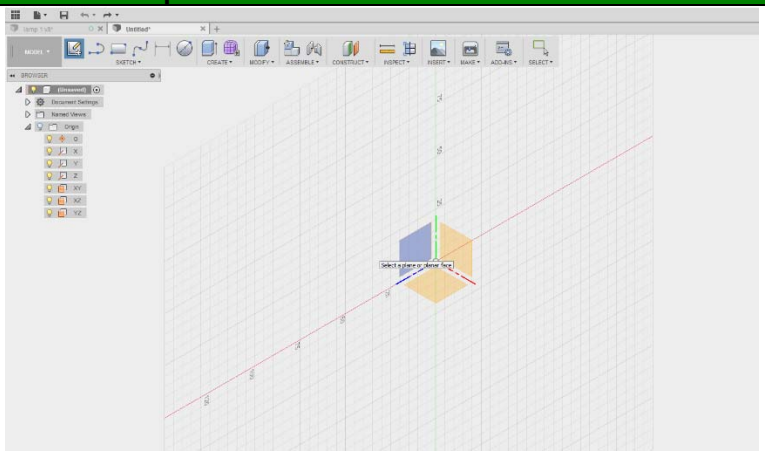
1. Draw an outline for the maximum size of the Lamp
2. Draw the Lamp Base
3. Draw the stem
4. Draw the Lamp shade
5. Import the circuit to check fit
6. Split the body into two halves and add the two halves to the base
7. Export the STL file for machining
8. Set-up the QuickCAM Pro file for machining

Drawing an outline for the maximum size of the Lamp	
Log into Fusion. Launch Fusion 360. Use your Autodesk ID to log into Fusion. If you do not have an Autodesk ID, you can create an account free of charge.	
In front of you is the main work area. During this tutorial you will learn about its various functions.	

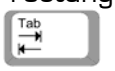
Drawing an outline for the maximum size of the Lamp	
Open the Data Panel. Click to display the Data Panel. The data panel slides open on the left.	
Create a new project. Click on "**New Project**". Enter **USB Lamp** in the Name field. **Double click** on the new project you have created to start working in this folder.	
Create a new design. Click and select **New Design**. A new empty design is created	
Press **Rectangle**	

Drawing an outline for the maximum size of the Lamp

Now **select** the workplane you will be working in. This should be the XY (right) workplane.



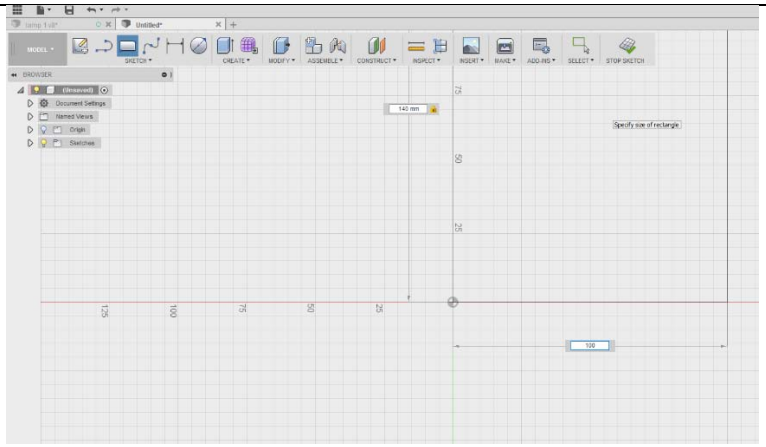
Starting at the origin point **left click** and move out the mouse to draw a rectangle. Type in: **100,**

press 

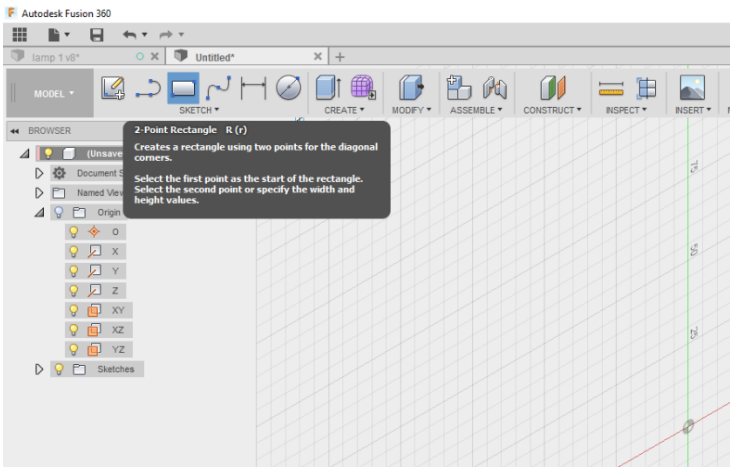
then type: **140**

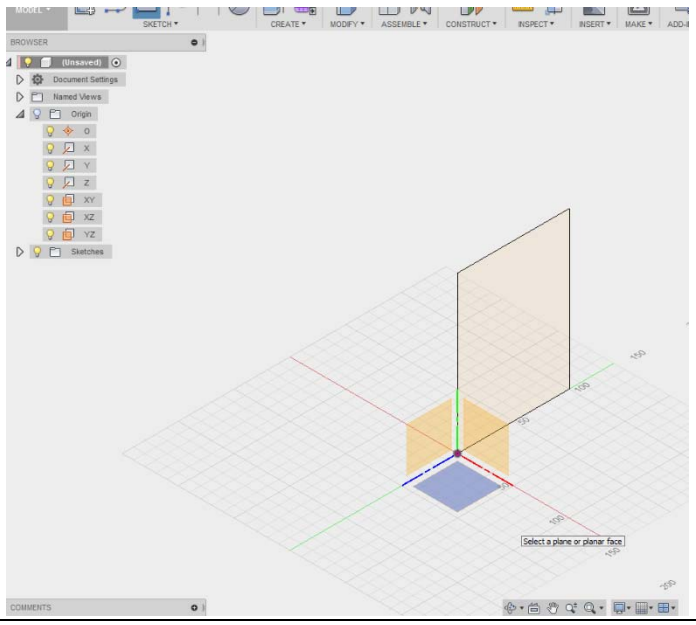
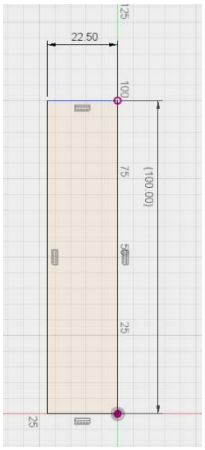
Finish the rectangle by pressing **Enter** on your keyboard.

Click **Stop Sketch.**



Once again press **Rectangle**

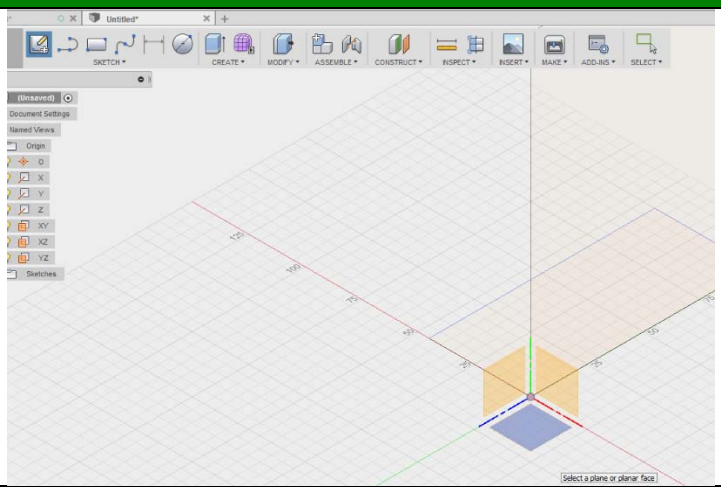


Drawing an outline for the maximum size of the Lamp	
This time select the **XZ (right)** workplane.	
Starting at the origin point **left click** and move out the mouse to draw a rectangle, **22.5mm** wide by **100mm** high. Click **Stop Sketch**	

Drawing the Lamp Base

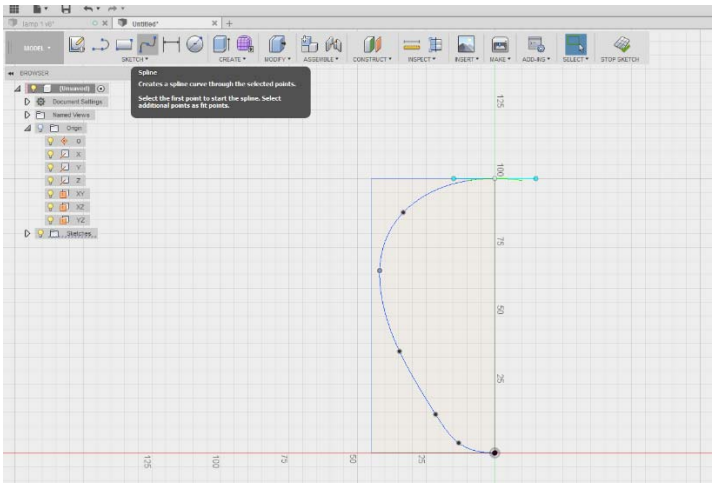
We are going to construct the base by drawing two profiles and lofting between them.

First Click **New Sketch** and select the **top workplane**.



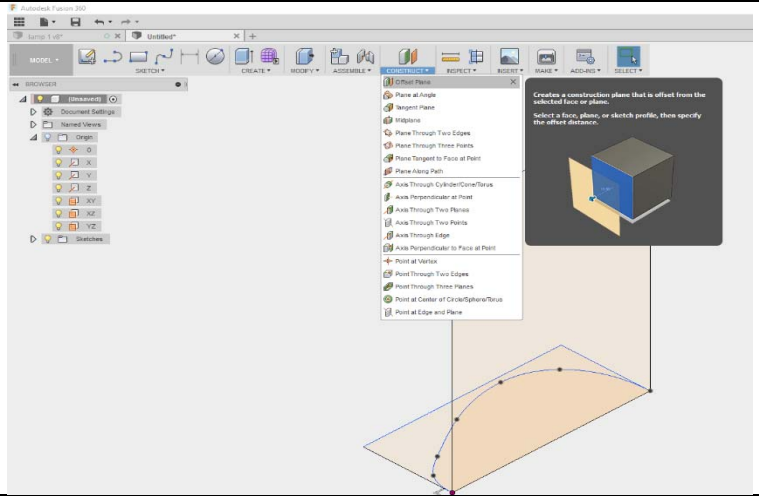
Using the **Spline tool**, draw out a profile sketch for the half of the base of the lamp, as opposite. Complete the shape by adding a **vertical line between the two end points.**

Click **Stop Sketch**



Click the **Home button** near the view cube at the top right hand corner of the screen.

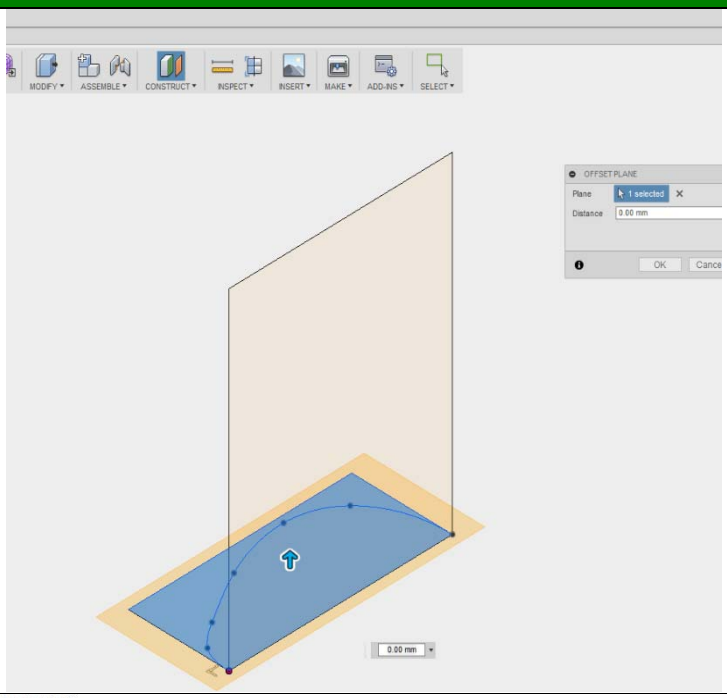
Click **Construct – Offset plane** and select the bottom workplane.



Drawing the Lamp Base

Drag the Blue Arrow up **10mm** (or however high you wish your base to be) and click **OK** to accept.

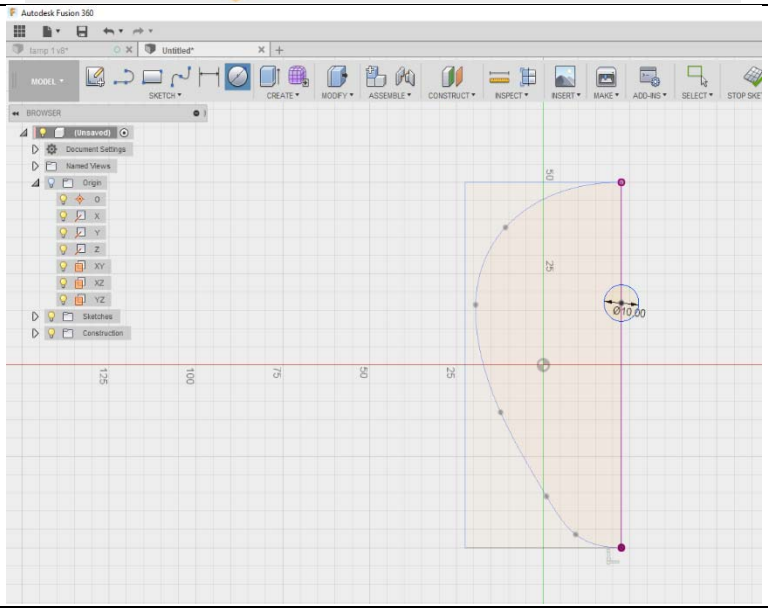
This has now added a new workplane for you to draw a top profile sketch on to.

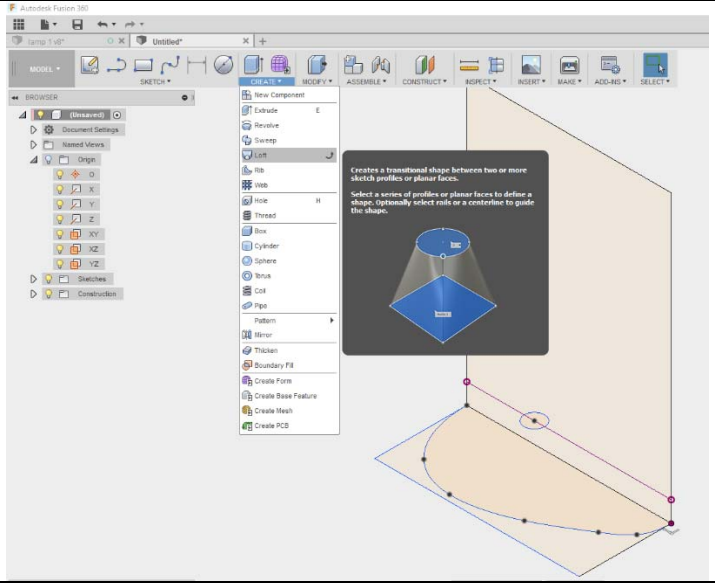
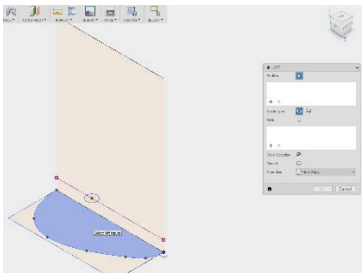
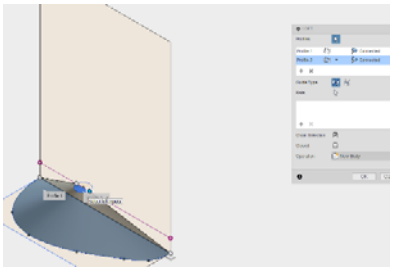
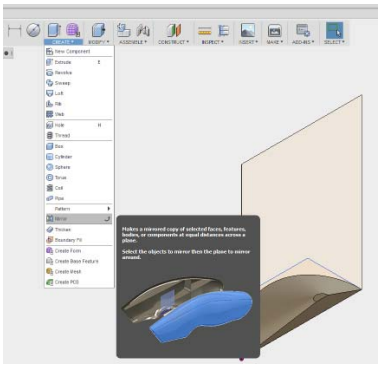
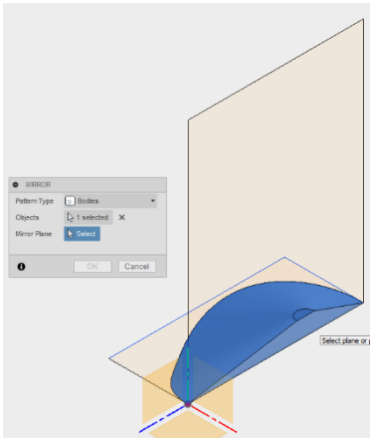


Click **Create Sketch** and select the new workplane.

Now sketch a profile for the top of your base.

Mine will be a **10mm circle**



Drawing the Lamp Base	
Click **Create - Loft**	
Select the **bottom profile** shape, and then select **one-half of the top profile shape**. Your solid shape should appear. Click **OK** to accept.	 
We are now going to finish the base by mirroring the body you have just formed to the other side. Click **Create – Mirror** In the dialog box click the following: Pattern Type : **Bodies** Objects : **Select the body** Mirror Plane : **Select the Flat Face of the body** Click **OK** to accept.	 

Drawing the Lamp Base

As a final step we are going to make the two halves into one body.

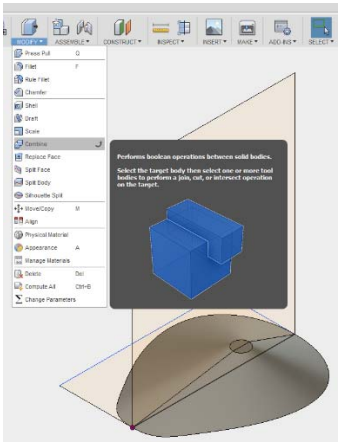
Click **Modify – Combine**

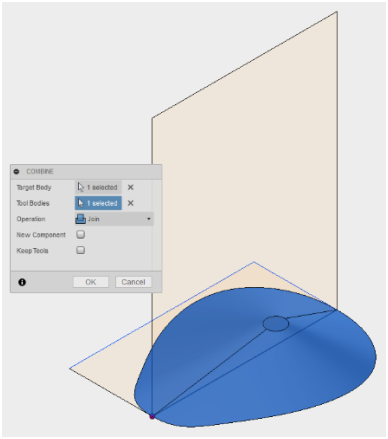
In the dialog box click the following:

Target Body : **Select one half**
Tool Bodies : **Select the second half**
Operation : **Join**
New Component : **Not Ticked**
Keep Tools : **Not Ticked**

Click **OK** to accept.

Now **save** your design.

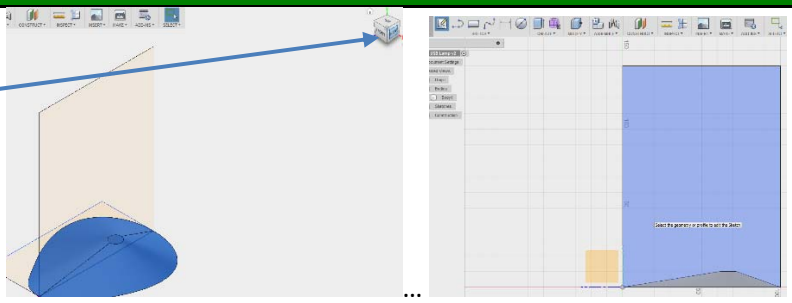




Drawing the Stem

Click on the right side of the View Cube to allow you to draw on the mid plane.

Click **Create Sketch** and select the **Right Plane**

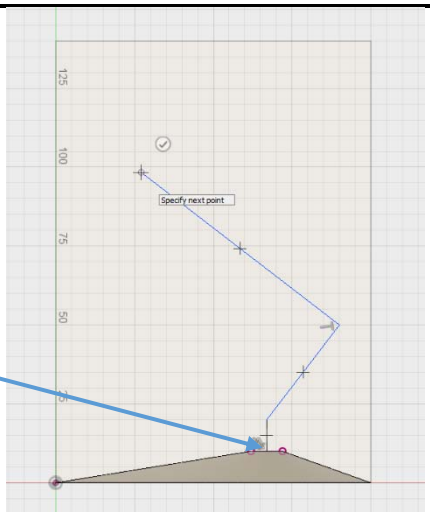


Using the **Line Tool**, or **Spline Tool**

Draw an outline shape for your stem.

NOTE: You MUST start this sketch from the top edge of the base

When you are happy with the shape, click **stop sketch**.

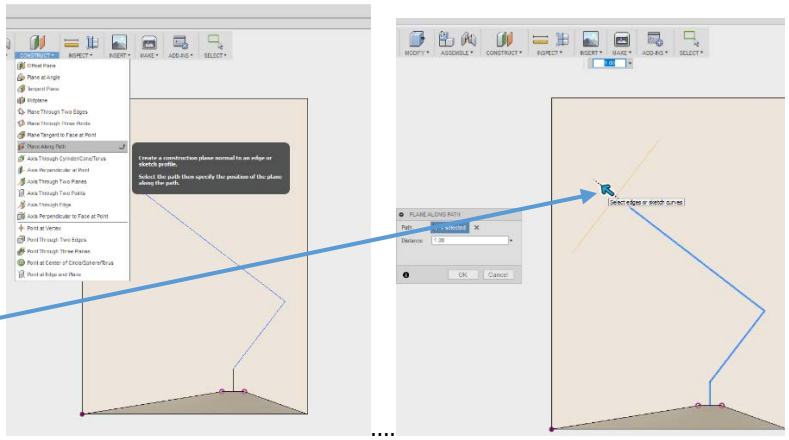


Next we are going to add a workplane on the end of this sketch. This will allow us to sweep a shape along the sketch to create a 3D object.

Click **Construct – Plane along a Path**

Select the sketch, then move the blue arrow to the end of the line as pictured.

Click **OK** to accept.



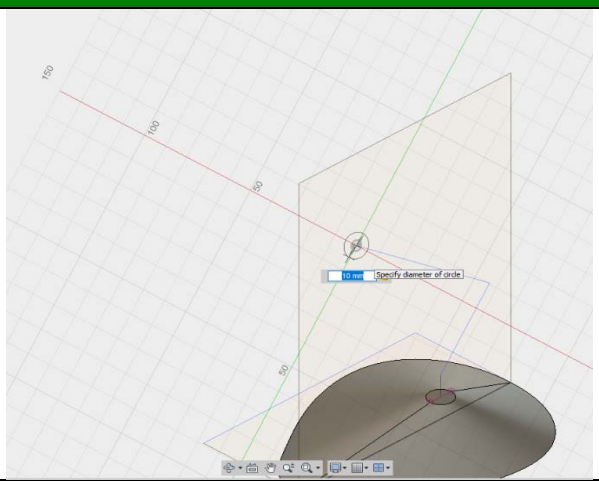
Drawing the Stem

Now click on this new workplane you have created and click **Create Sketch**

You will now need to draw your profile for the stem. This could be an ellipse, circle or square?

For this tutorial I will be drawing a **10mm** diameter circle.

Now click **stop Sketch**



Click **Create – Sweep**

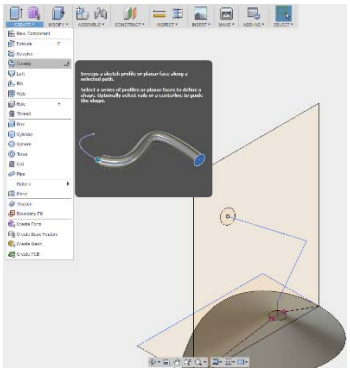
On the Dialog Box do the following:

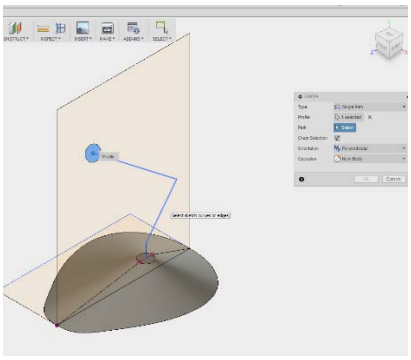
Tube: **Single Path**
Profile: **click the circle**
Path: **click the sketch for the stem**
Operation: **Join**

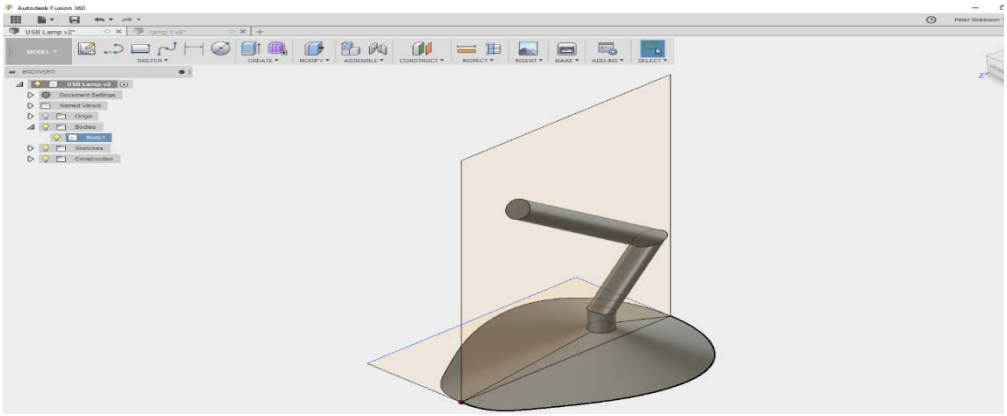
Keep all other settings the same.

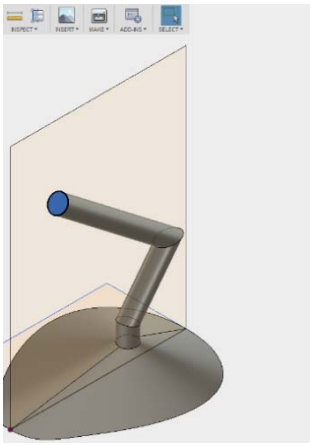
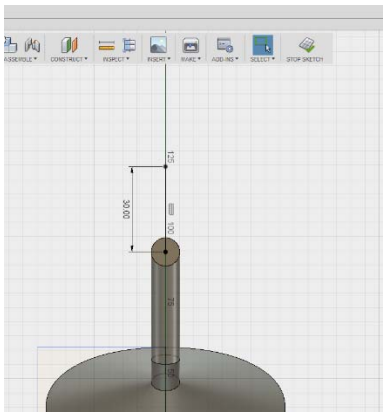
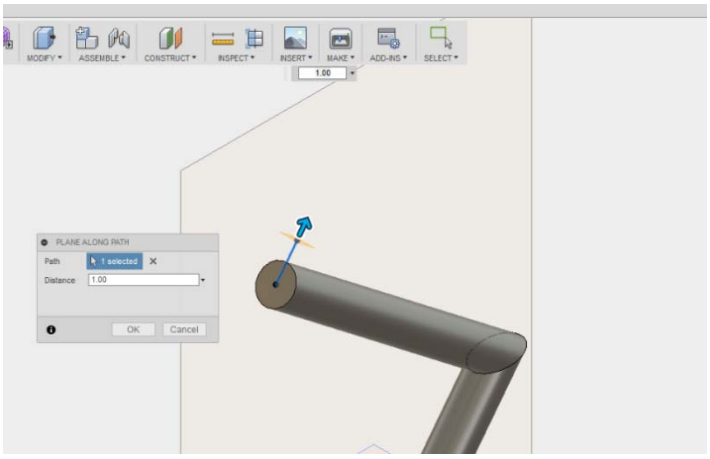
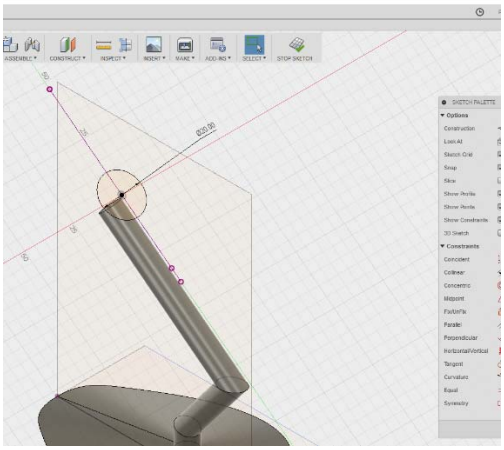
Click **OK** to accept.

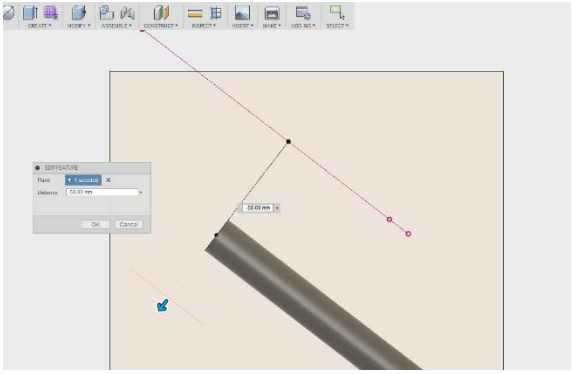
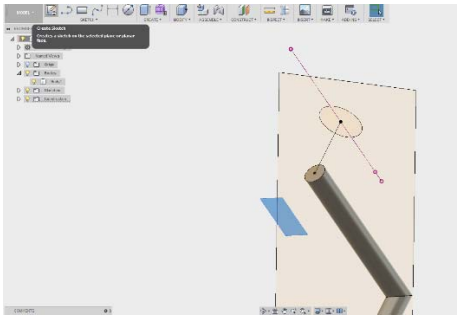
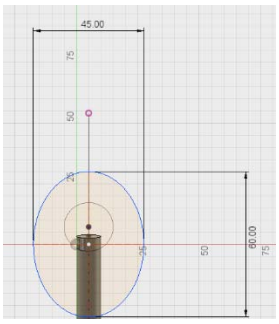
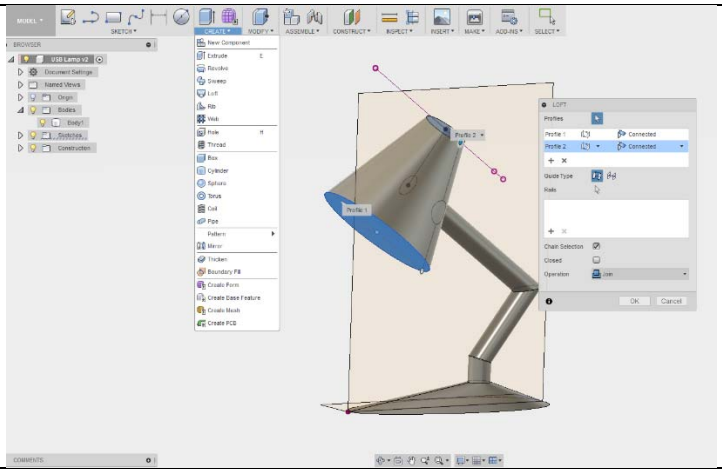
You should now have the stem as a 3D Body showing on your screen.

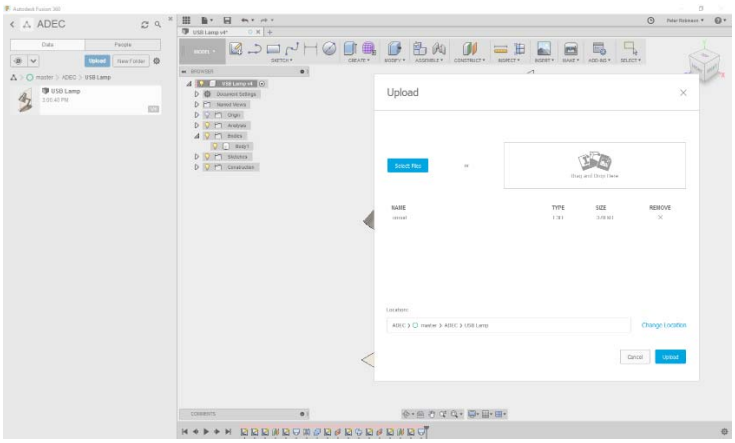
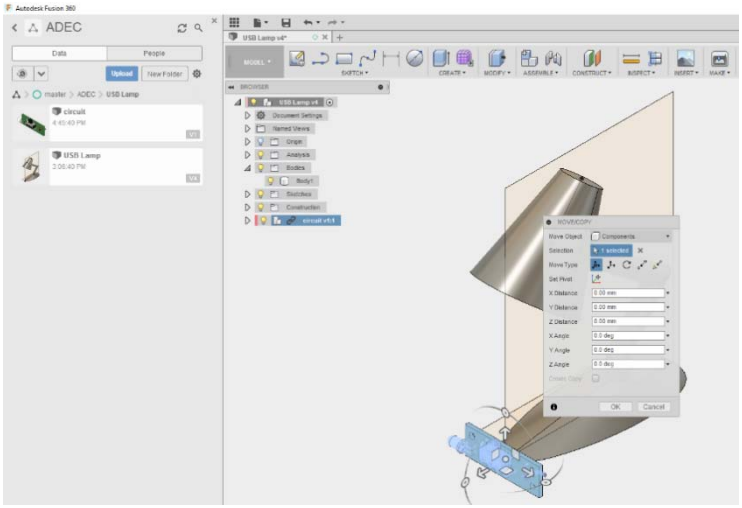






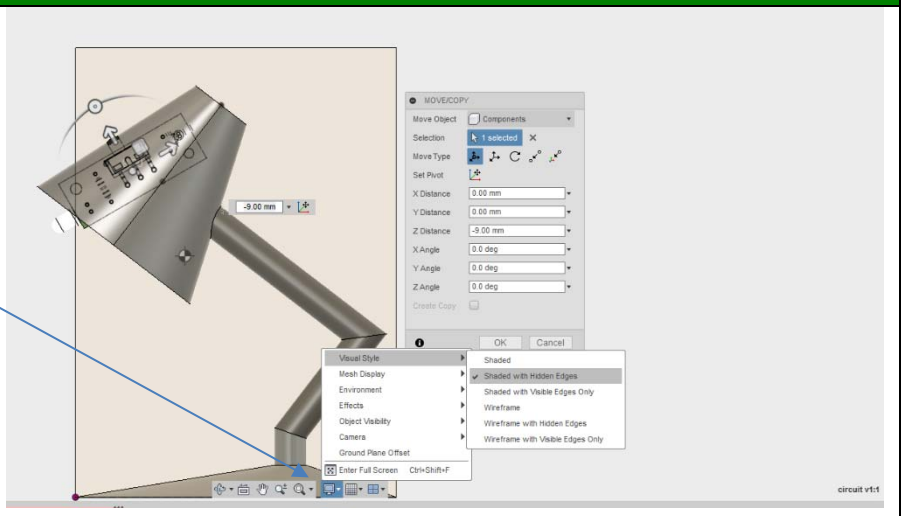
Drawing the Lamp Shade	
Click the top surface of the stem and click **Create Sketch** Draw a vertical line from the centre of the top face of the stem. This will help locate the back of the shade. For this exercise I will make it **30**mm.	 
Create a workplane at the end of the line by clicking **Construct – Plane along a path**. Select the line and ensure the plane is at the far end of the line by moving the blue arrow. Click **OK** to accept.	
Click **Create Sketch** Draw a **circle** on the newly created workplane. I have made mine **20mm** Click **Stop Sketch**	

Drawing the Lamp Shade	
Click **Construct – Offset plane** Select the Circle then drag down the blue arrow **55mm** ***Note: this will need to be 45mm minimum to accommodate the LED circuit board.*** Click **OK** to accept.	
Click **Create Sketch** and select your new offset plane. Now draw a new shape for the front of the shade. I have drawn an ellipse **45mm** wide by **60mm** high. ***Note: the width of this will be limited by the size of the material you will be making the lamp from.*** Click **Stop Sketch**	 
Click **Create – Loft** Select both the front and rear profiles of the shade. Ensure the operation is **Join.** Click **OK** to accept. Now **Save** your Design.	

Importing the circuit to check fit	
For this next part, we are going to import a pre-drawn circuit - the same as the real one you will be using to make your lamp. Open your file Browser on the left hand side and click the blue **upload** button. In the pop-up dialog box, click **Select Files**, and browse for the file **circuit.f3d** Now click **Upload** Once it says upload complete, press **close**, you should then have a copy of the circuit in your folder.	
Now import the Circuit into the Lamp drawing by **clicking and dragging** the picture of the circuit from the browser into the drawing area of the lamp. Here you can now use the manipulation tools to move the circuit into place in the lamp.	

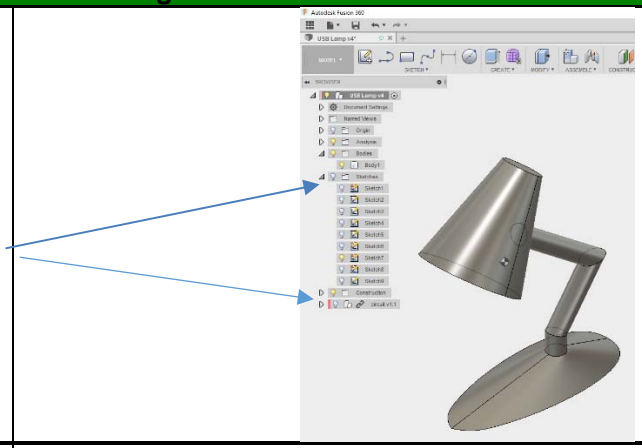
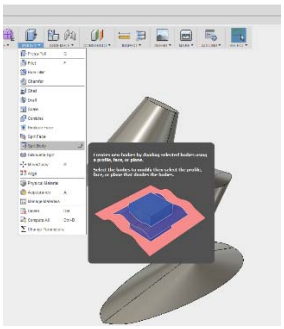
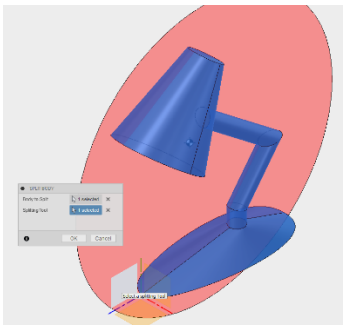
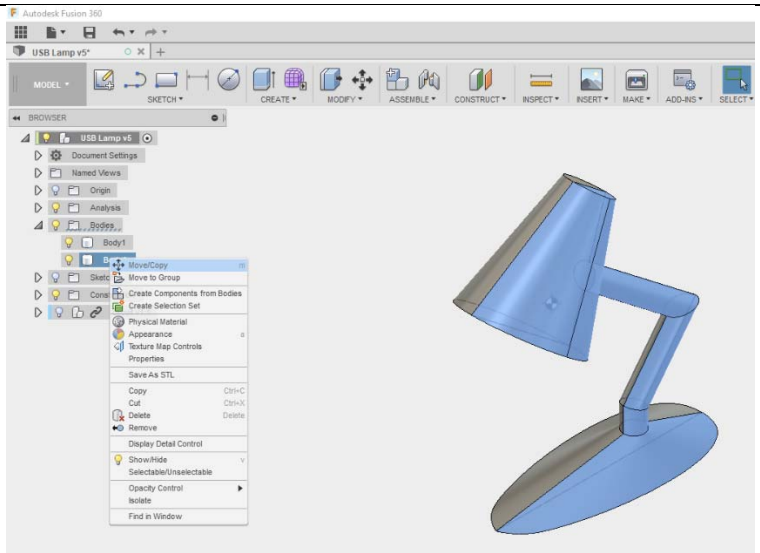
Importing the circuit to check fit

You can change the visual style of the drawing, to check if it fits, by clicking on the screen button at the bottom of the screen and clicking:
Visual Style – Shaded with Hidden Edges



Here, you can see that the circuit fits perfectly, with the switch piercing through the top, and the LED protruding from the front.

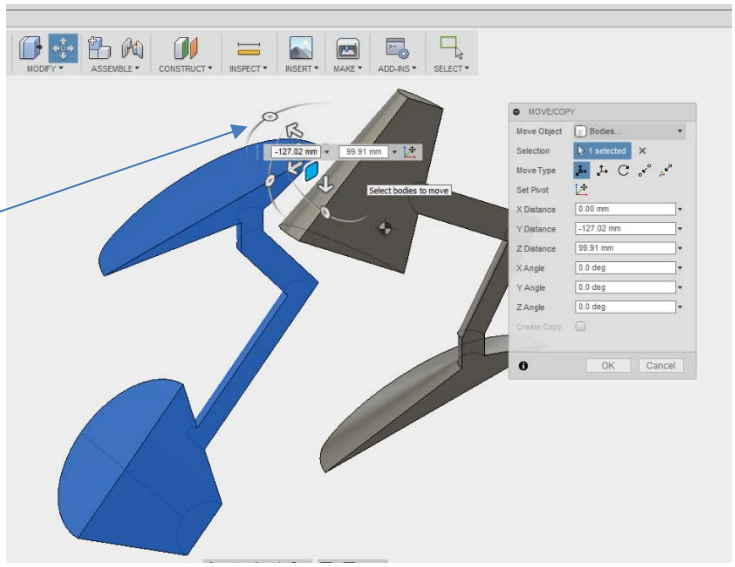
If, at this point, you find your circuit does not fit, you can edit parts of the drawing using the history buttons at the bottom of the screen.

Splitting the Body into two halves and adding the two haklves to the base to create a mould	
Using the Browser tree on the left hand side of the screen, toggle off the sketches and circuit by clicking on the lightbulb next to them.	
Now click **Modify – Split Body** In the Dialog box: Body to Split: **click on the full lamp body** Splitting Tool: **click on the Right plane**. Click **OK** to accept	 
If you check under Bodies, you should now have two bodies: one for each half of the lamp. We will now move one half, so that it is next to the other.	
In the Browser tree on the left hand side of the screen: **Right Click** on **Body2** and click **Move/Copy**	

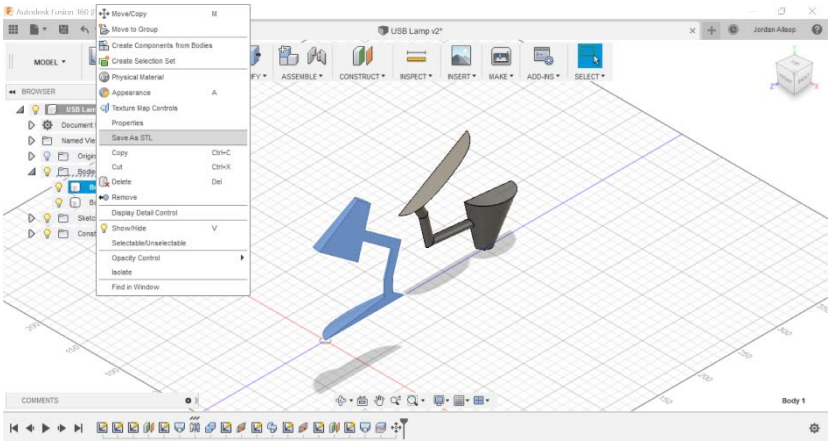
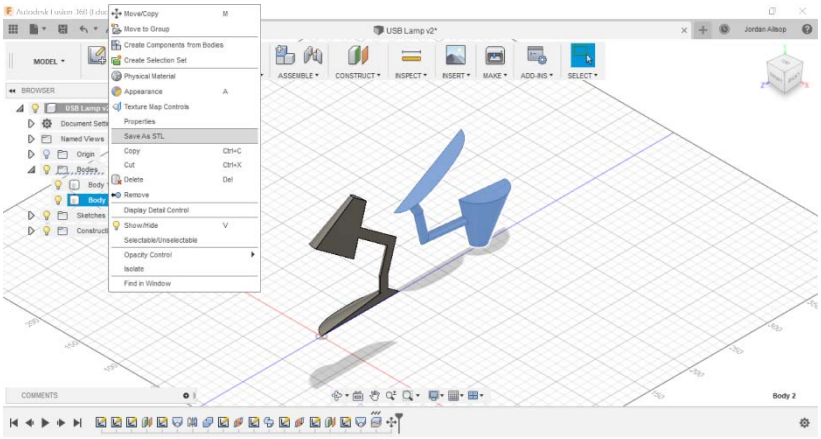
Splitting the Body into two halves and adding the two haklves to the base to create a mould

Using the manipulators, move Body2 so it is level with and next to Body 1.

When you are happy with the position Click OK to accept.



Exporting the STL file for machining

With your design open in Fusion 360, under the Bodies section **right click** on Body 1 and click **save as STL.** At this stage, you need to name your files: **YOURNAME_Body1.stl**	
Repeat the step above, but this time for Body 2. **YOURNAME_Body2.stl**	
Now Open the program "**QuickCAM PRO**"	

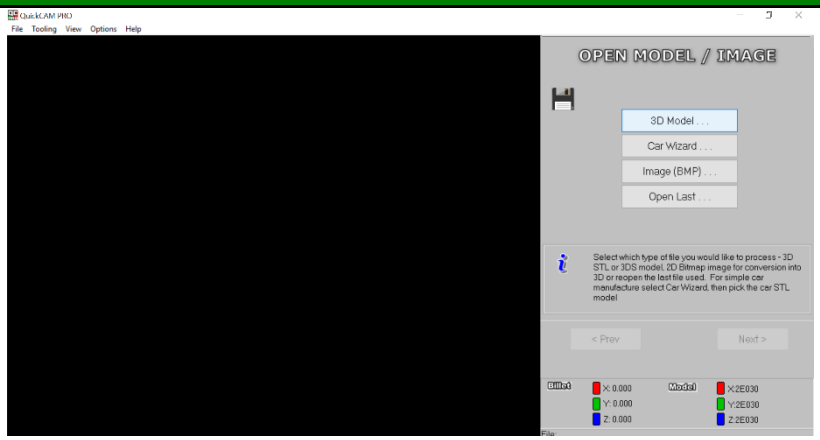
Exporting the STL file for machining

Once open, the first screen you will be faced with is the **OPEN MODEL/IMAGE** screen.

Click "**3D Model**"

And select the first STL file you just created **YOURNAME_Body1.stl** And click "**Open**"

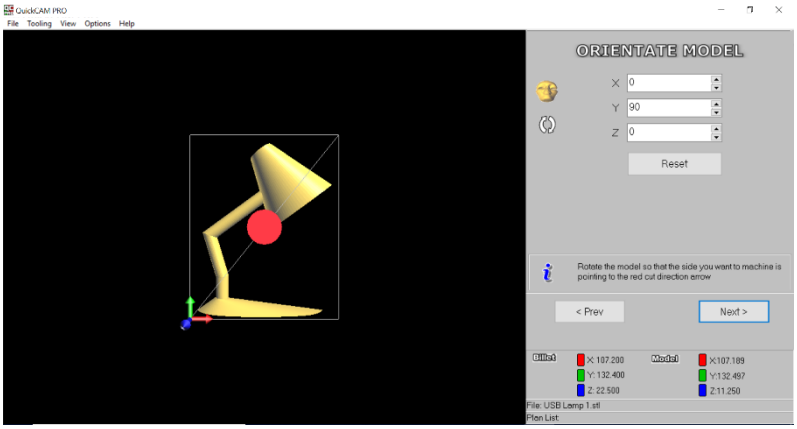
Now click **Next**.



Your next step will be to **orientate the model,** so you are machining on the correct side of the Block, with the Origin in the correct place.

Using the **X,Y, and Z** button on the side, rotate the model so the Origin is in the bottom left hand corner and the tool (big pink arrow) is above the face as shown.

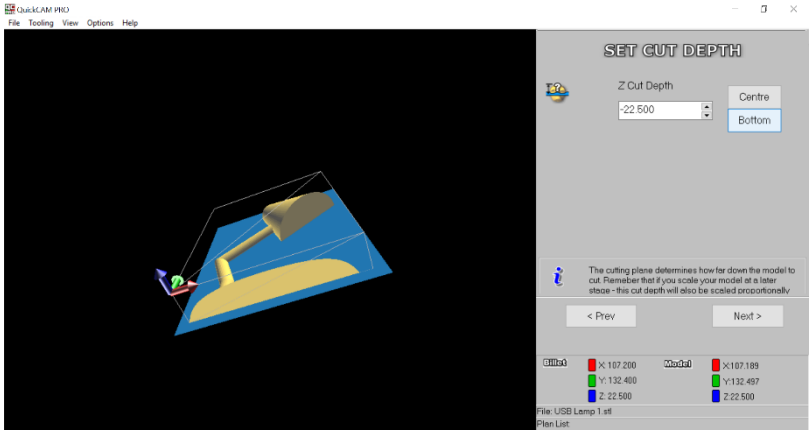
Click "**Next**"



Set Cut Depth

Type in the depth so the blue surface is level with the flat surface. This should be **-22.5mm** and click "**Next**"

This will ensure that when your part is machined, the tool will not machine through all of the material.



Exporting the STL file for machining

Set Billet Size

Set the Billet to the following dimensions:
X: **110**
Y: **150**
Z: **50**

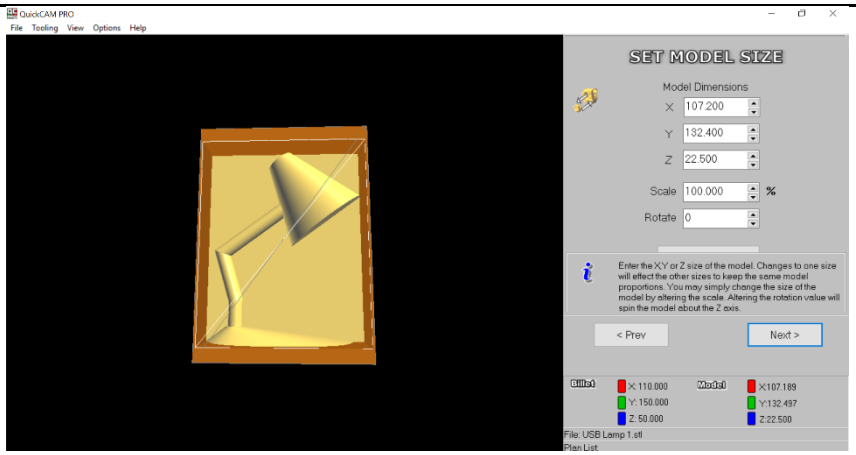
Click "**Next**"

Set Model Size

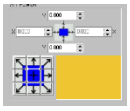
On this screen, there should be no need to change anything, as you would have already drawn your product to match the dimensions.

Just check that the scale is **100%**

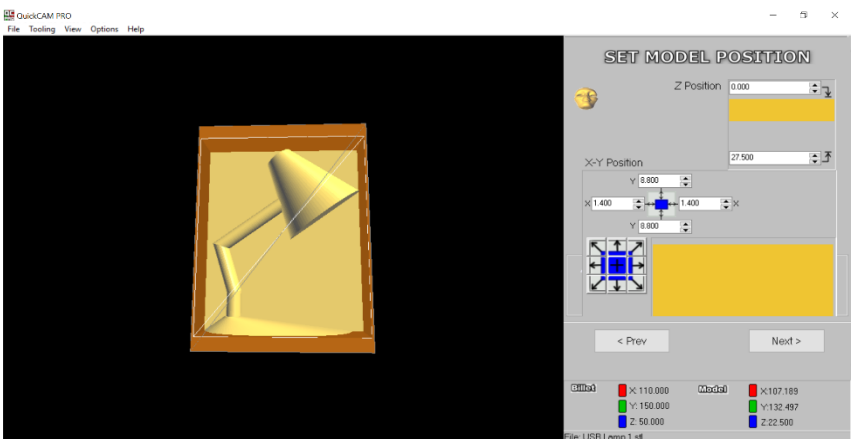
Click "**Next**"



Set Model Position

Your model should fit perfectly within the billet and should be centralised,
If not, click on the **centre blue block**  to centralise your model in the billet.

Click "**Next"**



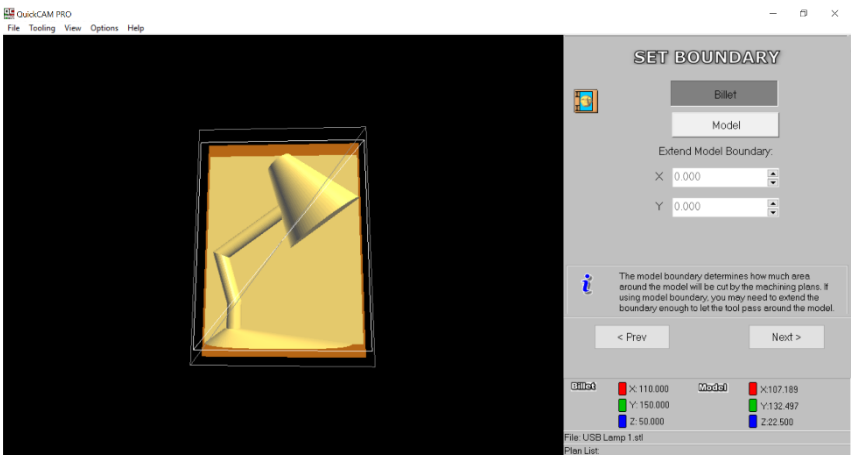
Exporting the STL file for machining

Set Boundary

Ensure you click **Billet** to remove any excess material from the billet.

This will ensure your tool will be able to machine around the full perimeter of the block.

Click "**Next**"

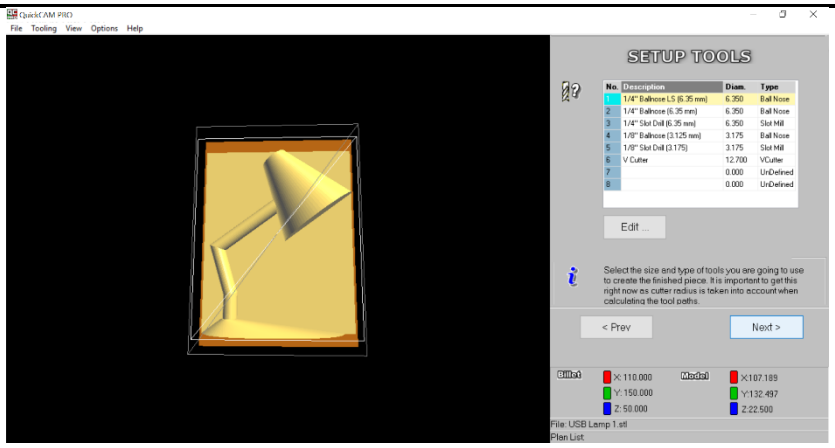


SET-UP TOOLS

This section will allow you to set-up the tool you will be using to machine this model.

You will only be using the **¼" Ball nose cutter (6.35mm).** Ensure this tool is showing.

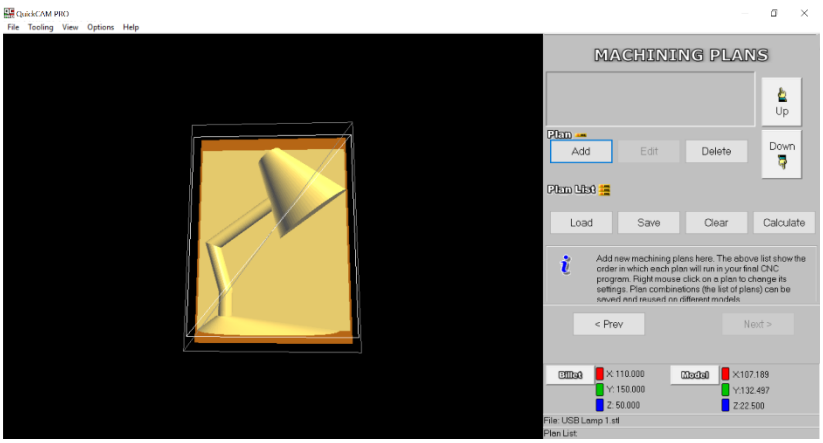
Click **"Next"**



MACHINING PLANS

In this section we will set up two machining plans: one will be a rough machining process, and the other will be a finishing pass to ensure a good quality outcome.

Click "**Add**" to open the next screen.



Exporting the STL file for machining

Selecting Machining Plan

Click "**Raster Roughing**"

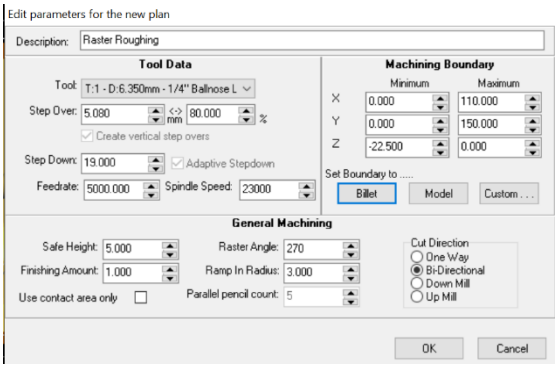
You should not need to make any edits in the top section. However, just check the following:
Tool Data
Step Over: **80%**
Feedrate: **5000mm/min**
Spindle Speed: **23000 RPM**

In the bottom section make the following edit:

General Machining
Finishing Amount **1mm**

(This will leave 1mm of material on the part to be machined using the second plan.)

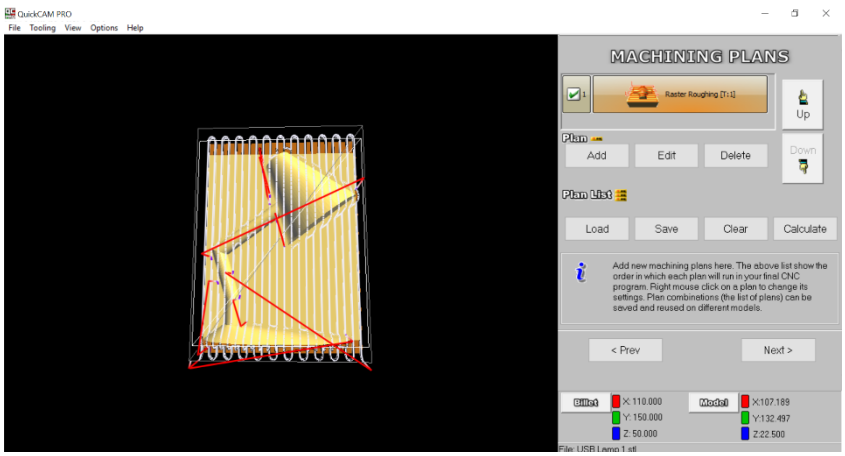
Click "**OK**"



This will take you back to the **Machining Plan Screen**.

You will see the program start to calculate machining paths from the set-up you just created.

Click **Add** once again to add the finishing pass.



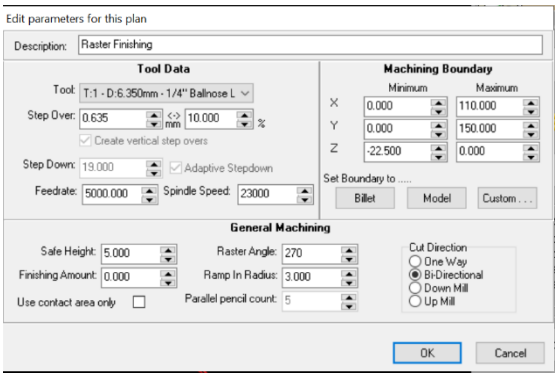
Exporting the STL file for machining

Selecting Machining Plan

Click "**Raster Finishing**"
Now make the following edits
Tool Data:
Step Over: **10%**

General Machining
Raster Angle: **270**

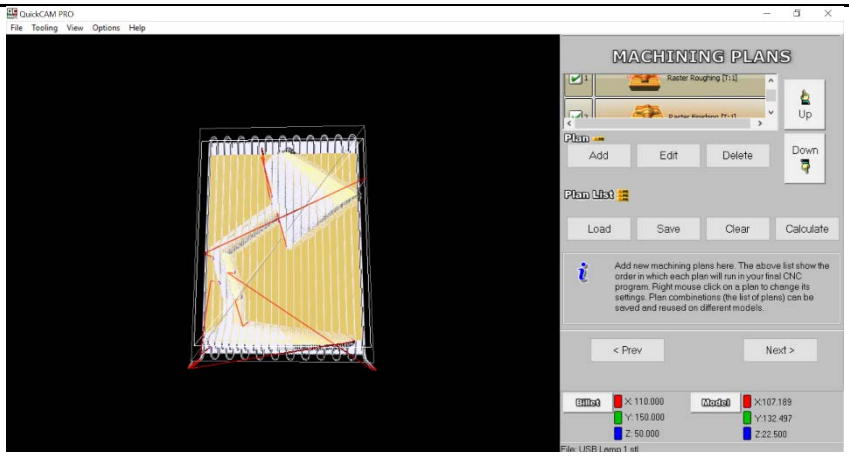
Click "**OK**"



This will take you back to the **Machining Plan Screen**.

You will see the program start to calculate your second machining paths from the set-up you just created.

Click **Next** to move to the next screen.

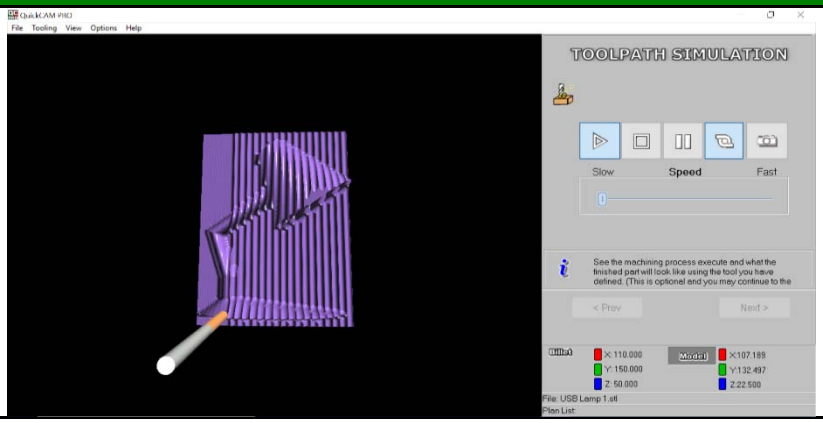


Exporting the STL file for machining

TOOLPATH SIMULATION

Press the large **Play** button to run a simulation of how your part will be machined.

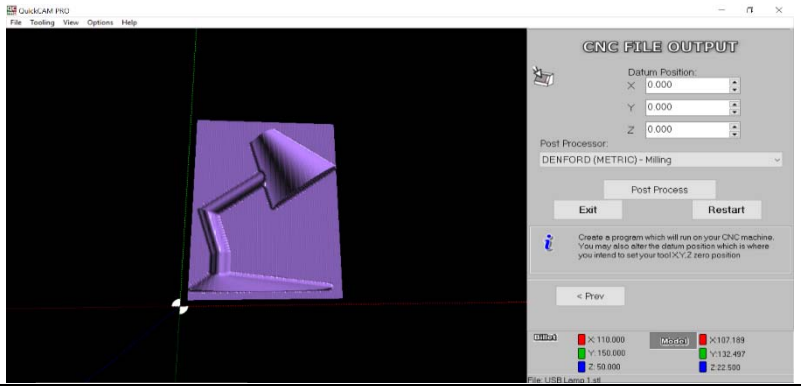
Click "**Next**"



CNC FILE OUTPUT

This screen is the final screen on this software.

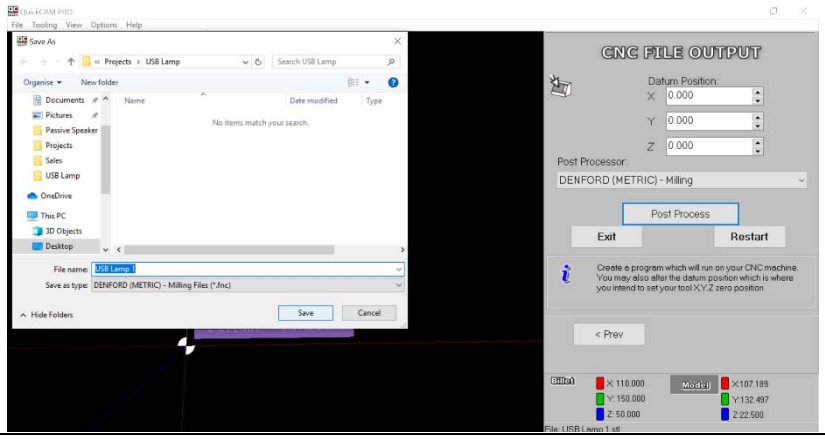
Ensure the Datum Position boxes are all set to Zero. This will match the datums of the machine and the foam.



Ensure the post processor is set to **DENFORD (METRIC) - Milling**

And click "**Post Process"**

Ensure you know where the file is being saved and press **Save**



Repeat the QuickCAM PRO steps for Body 2

The next stage is to use your moulds to vacuum form the Lamp.

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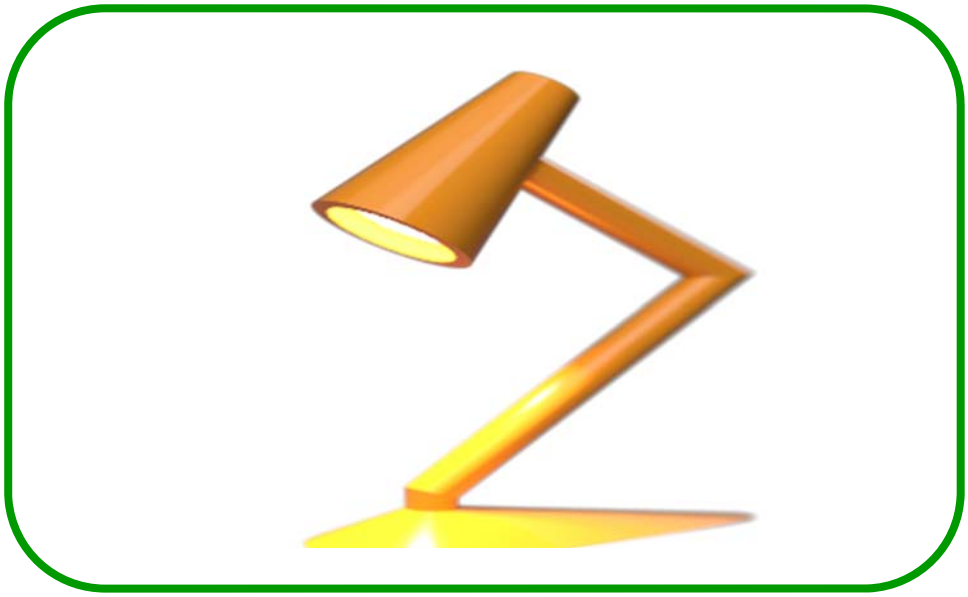
CAD/CAM Projects

USB Lamp Project



TEACHER SUPPORT GUIDE

Denford: USB Lamp Project - Denford CNC Router/Vacuum Forming Machine

Teacher Support Guide

Students are tasked with designing and making a USB powered LED Lamp. The Lamp will be manufactured by vacuum forming, the mould for which will be designed in Fusion 360 and machined using high density modelling foam on the Denford CNC Router. The design must be made in two halves and glued together when finished.

Project Timeline Summary

It is anticipated that learners will spend a total of approximately 8 hours in producing the work for this project (each session being around 1 hour). 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	Product Research & Design	Understanding the context of the product. / secondary research of products / start to generate an idea.
2	Designing & Developing a Final Solution	Finish the sketch idea to satisfy the requirements of the problem / sketch final design in 3D form, both assembled and exploded.
3	CAD Design	Using Fusion 360 generate a CAD version of the design. The final design will need to be split in half and attached to a flat body to create a vacuum form mould.
4	Finish CAD Design / CNC Post Processing	Using Fusion 360, complete and CAD work, then export STL's for manufacture.
5	QuickCAM Pro Post Processing	Import files into QuickCAM Pro and set up files for importing into VR Milling.
6	CNC Manufacture & Vacuum Forming	Using VR Milling, and the Denford Router, machine out the mould, prepare the mould for forming and vacuum-form the part.
7 & 8	Assembly & Finishing	Trim the product, prepare the electronic circuit and assemble the final product.

Project Delivery Session Detail

Session	Focus	Session Content
1	Product Research & Design	Ask students to read through the design problem, then working individually to carry out research on the internet and make rough sketches of products they like the look of. Ask them to annotate any design features they particularly like. Now get students to take a look at the design considerations page and talk them through the constraints such as fitting the circuit board and the overall size of the product and how the product will be manufactured. Explain to the students that we will not be going through the traditional designing process, and the development of the design will happen within the designing of the CAD model. Ask students to draw a concept sketch of a lamp they would like to bring forward using inspiration from their mood board.
2	Designing and Developing a Final Solution	Students will need to finish their concept idea if they have not already done so, then move straight into the layout of the mould and sketch out how their design will look split in half onto the sheet of foam.

Session	Focus	Session Content
3	CAD Design	Using Fusion 360, follow the CAD support booklet to draw their own lamp design. The process will start by students drawing a complete sketch, 150mm x 110mm x 22.5mm, This will represent the maximum size for one half of the design. Then, through a series of drawing operations, create the overall shape of the lamp. Students will then add a circuit assembly to the lamp to check for fit, and edit their design as necessary. Finally, students will split the block into two halves and move the bodies to create a vacuum form mould ready for manufacture. Students can use the CAD support booklet to support them in self-guided learning.
4	Finish CAD Design / CNC Post Processing	Students will need to use this session to firstly complete any outstanding CAD work and secondly export their file for use in QuickCAM Pro. This will need to be in STL format. Follow the CAD support booklet to ensure the file is ready and exported correctly.
5	QuickCAM PRO Post Processing	Using the CAD Support Booklet, Students will need to import their drawing into QuickCAM Pro and set-up machining programmes for the mould. Within QuickCAM, it will be necessary for students to set-up a roughing and finishing pass, to ensure a good quality finish.

Session	Focus	Session Content
6	CNC Manufacture & Vacuum Forming	The Teacher/Technician will need to set up the router to hold the pre-cut sheet of foam. Import the **.fnc** file into VR CNC Milling V5, and machine using the long series cutter. Also Offsets for this project will need to be pre-programmed, all students can then use this ***– This will be covered in training; however, you can also check your Denford VR CNC Milling V5 Training guide for guidance on how to do this.*** Once the mould is machined, students will need to prepare the mould for vacuum forming, by lightly sanding any imperfections and coating it with the mould release agent. Students will then vacuum form their designs, using a HIPS vacuum forming sheet.
7 & 8	Assembly & Finishing	Once formed, students will then need to remove the mould as best they can. Do not worry if you cannot get all of the mould out. Now using the trimmer, cut around each profile so the students have two separate halves. Students will now need to assemble the USB LED circuit. Place the circuit into the shade of one of the halves of the lamp and trim the shade where the switch and LED are and glue in the circuit. Finally, glue the two halves together and keep them held in place with masking tape, until set and dry.

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CAD/CAM Projects

USB Lamp Project

Project Duration: 8 Hours

STUDENT WORKBOOK

Denford: USB Lamp Project - Denford CNC Router/Vacuum Forming Machine

Student Workbook

DELIVERY PLAN (HOURS)

1. Product Research & Design
2. Designing & Developing a Final Solution
3. CAD Design
4. Finish CAD Design / CNC Post Processing
5. QuickCAM Pro Post Processing
6. CNC Manufacture & Vacuum Forming

7 & 8. Assembly & Finishing

DESIGN PROBLEM

Students are tasked with a project to design and manufacture a USB powered Work Lamp.

The design will follow the design principles, found from researching products, such as clean lines, basic shapes and bright colours.

MACHINES

Denford CNC Router
Router Tooling Package – BI00846
Vacuum Forming Machine

MATERIALS

2 x 110mm x 150mm x 50mm High Density Modelling Foam – (Pack of 50) BI03508A

HIPS Vacuum Forming Sheet

ADHESIVE

HIPS Glue

FINISHING MATERIALS

Glass Paper for finish
Craft Knife

EXTRA INFORMATION

The final product will be made by vacuum forming. However, the mould will be manufactured using high density foam on a Denford CNC Router, machining both halves at the same time.

Required as part of this product is a USB powered LED PCB with a mounted slide switch. Positioning of this will need to be considered in your designing!

Product Research

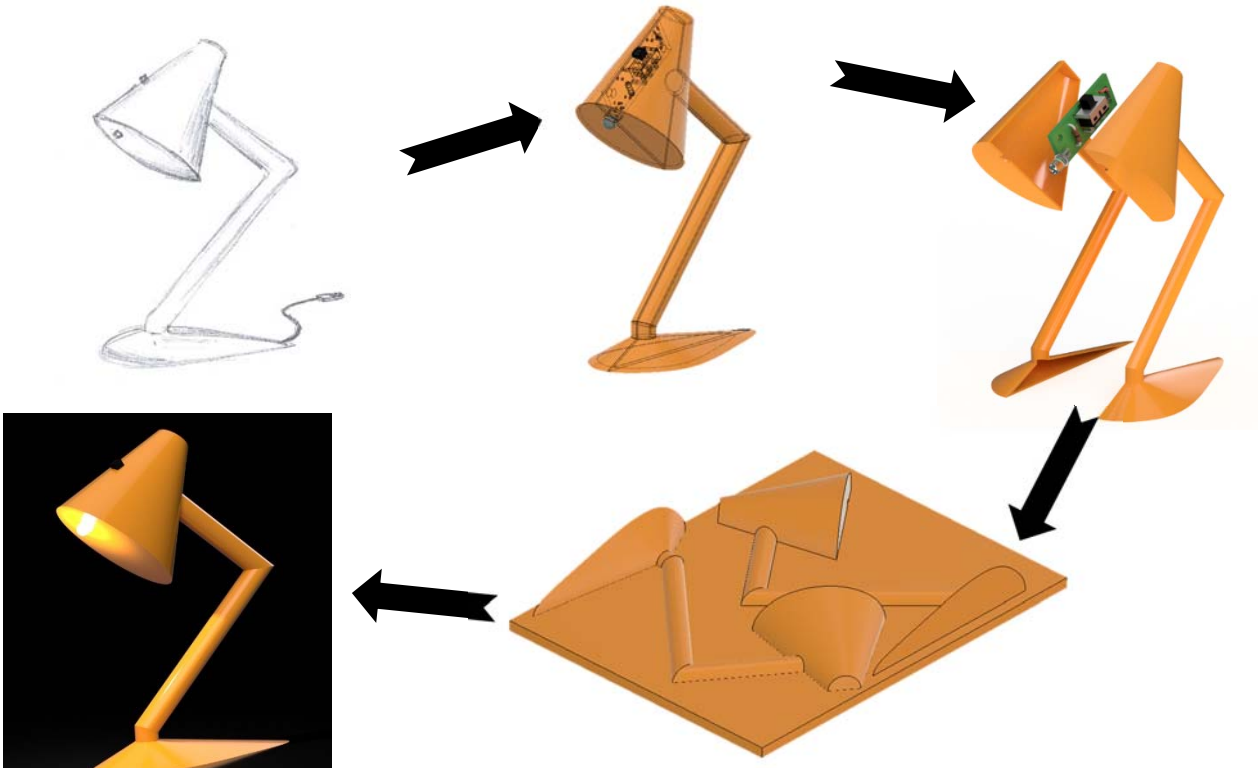
Primary research task.

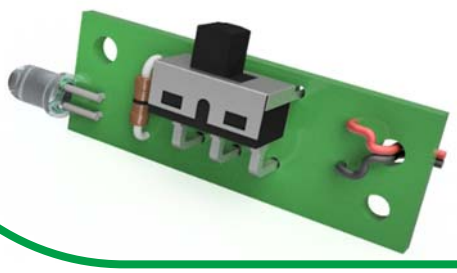
Take a look at available work lamp designs on the internet. Sketch some of the products which capture your interest. These will form the basis of your

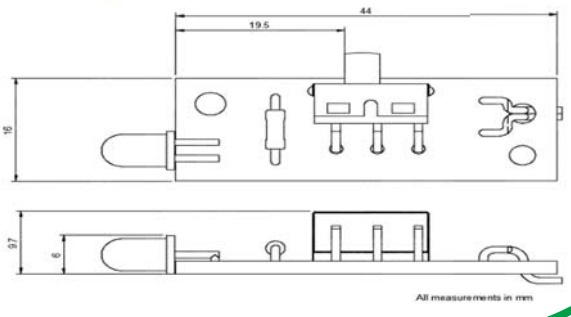
Design Considerations

You have some limitations on your design creativity within this project:

1. The size of lamp. The overall size of your lamp cannot be any larger than 140mm x 100mm x 45mm.
2. Your lamp shade MUST be able to incorporate the circuit board shown at the bottom of this page. You must also ensure you have allowed access to the switch. Dimensions have been included to aid your designing.
3. The final product will be made using the Vacuum Forming process. Therefore, ensure that the design has no indentations, to allow the mould to be easily removed.







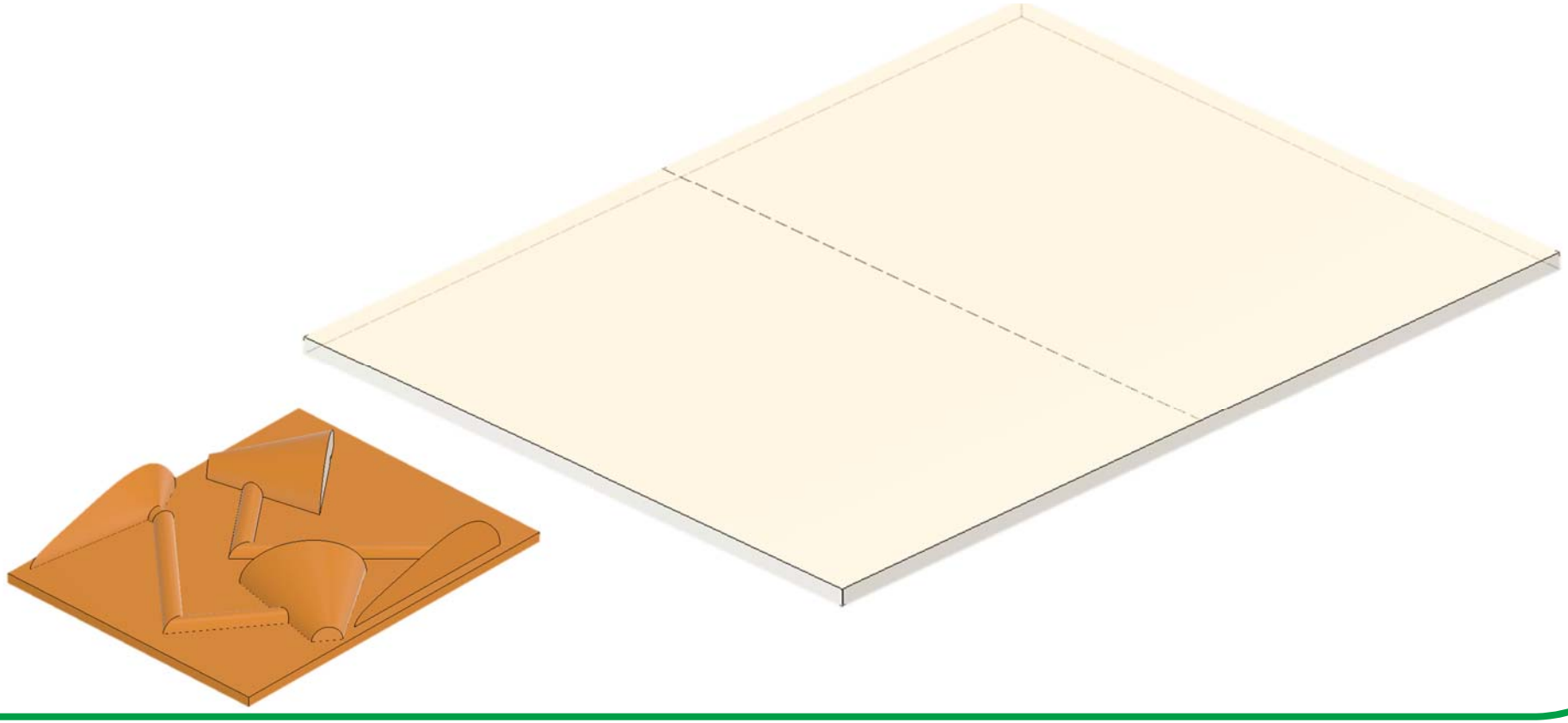
Most of your design development for this project will be done on CAD. At this stage, we would like you to produce just one sketch idea. This may not end up being your final idea, as your CAD design develops.

Concept Sketch

Working with the previous sheet as guidance and your design research sketches as inspiration, draw out an idea you would like to take forward. Think about how the lamp will balance, as well as where the circuit can be placed.

Mould Layout

Sketch out how your design will be laid out on the sheet of foam. Note the centre line for both halves – an example can be seen below:



CAD Design

Follow the tutorial guide to produce a CAD version of your final design on Autodesk Fusion 360. An example can be seen below:

Your Final CAD Design

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

Product Assembly

Once you have machined your mould on the Denford CNC Router, you will need to assemble the rest of the product by doing the following:-

1. **Preparing your Mould:** Once machined, if you have any material sticking proud of the mould, please have this trimmed off on a band saw. It may also be necessary to lightly sand any imperfections at this stage. Finally, in a well-ventilated area, give the mould a light covering of mould-release spray to aid with removal after vacuum-forming.

2. **Vacuum-Forming:** Following your teacher's guidance, vacuum-form your lamp using the HIPS sheet. Once formed, remove the mould and use the craft knife to trim around each side of your lamp.

3. **Building the Circuit:** Using the instructions, build your circuit for the LED light.

4. **Attaching the Circuit:** You will need to offer up your circuit to the lamp, and make a small cut-out for the LED: one for the switch, and one for the lead to exit the lamp. Once these are glued in place, fix the circuit board using the adhesive supplied. Ensure your USB lead is exiting in a suitable place.

5. **Final Assembly:** Using the adhesive supplied, glue the two halves of the lamp together. Hold together with masking tape until the glue has set. Once set, remove the tape and carefully trim and sand the bottom of the base to ensure it sits level.

6. **Test the Lamp:** Plug your USB lamp into any USB power source to try it out.

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