MFG463552-L

From Scribble to Sketch to Solid to Swarf—Having a Go at the Full Design-to-Manufacture Process in Fusion 360

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Learning Objectives

- Learn how to take a product through the entire design and manufacture process in Fusion 360.
- Explore workspaces other than “Design” in Fusion 360.
- Learn how design changes can be cascaded to other subsequent processes in Fusion 360.
- Learn about the different ways of collaborating in Fusion 360

Description

Fusion 360 software is a complete product-design tool that enables users to go from concept to production with integrated CAD, CAM, and CAE technology. We hear that a lot, but in reality, what does it actually mean? Is it the Swiss Army knife of design software? Can it do everything? This class will take attendees through the complete design and manufacturing process, from initial concept scribbles all the way through to generating files for 3D printing and toolpaths for a computer numerical control (CNC) job. We'll kick off with bringing some drawings into the Fusion 360 environment and using them as template. Then we will move on to some basic simulations, carrying out a design change and generating some communication visuals with custom materials. Finally, we'll hit up the Manufacture workspace to generate some toolpaths for CNC machining.

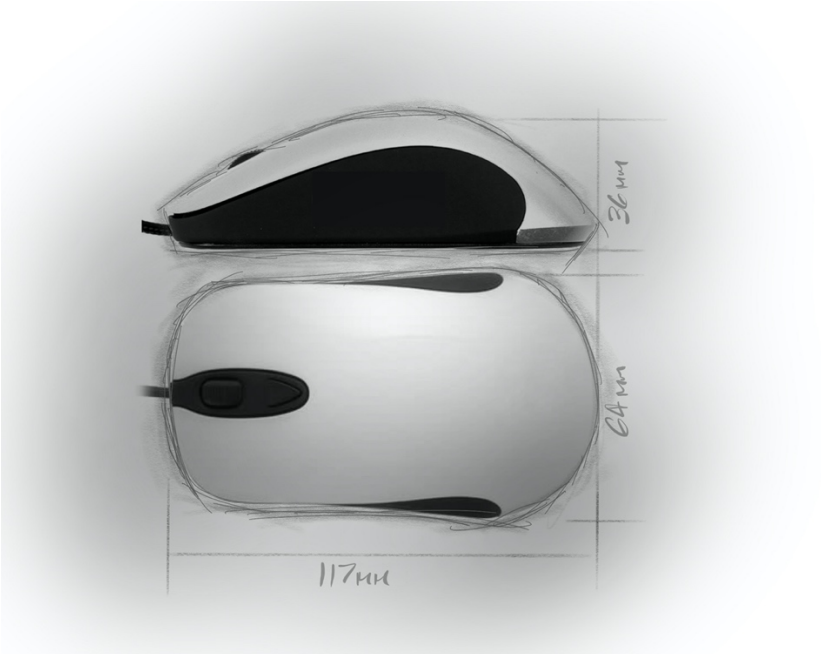
Speaker(s)

Simon is an Associate Professor of Engineering in School of Engineering at the University of Warwick. He teaches core Engineering Design to all first year students as well as supervising third and fourth year design projects. He is academic lead for Warwick’s Engineering Build Space and head of the Digital and Material Technologies Research Lab.

Ollie is a 4th Year Manufacturing and Mechanical Engineering Student, working towards his master’s degree at the University of Warwick, UK. As a community manager on the Autodesk Ambassador Hub, he helps to ensure that the hub runs smoothly and helps to create content for the Autodesk education community. Ollie first learnt Fusion 360 in a university class in his first year and after 3 years of progression, planned, created and delivered a set of Fusion 360 tutorial videos that were used to teach all first-year engineers at the University. He also works teaching Fusion 360 for Teach3D, an Autodesk Learning Partner in the UK, delivering online video tutorials.

The design concept

The first step in our process is to get our concept drawing. This concept drawing has been made in Autodesk Sketchbook. Don't worry about creating a concept drawing, it has been provided for you in the class data set.



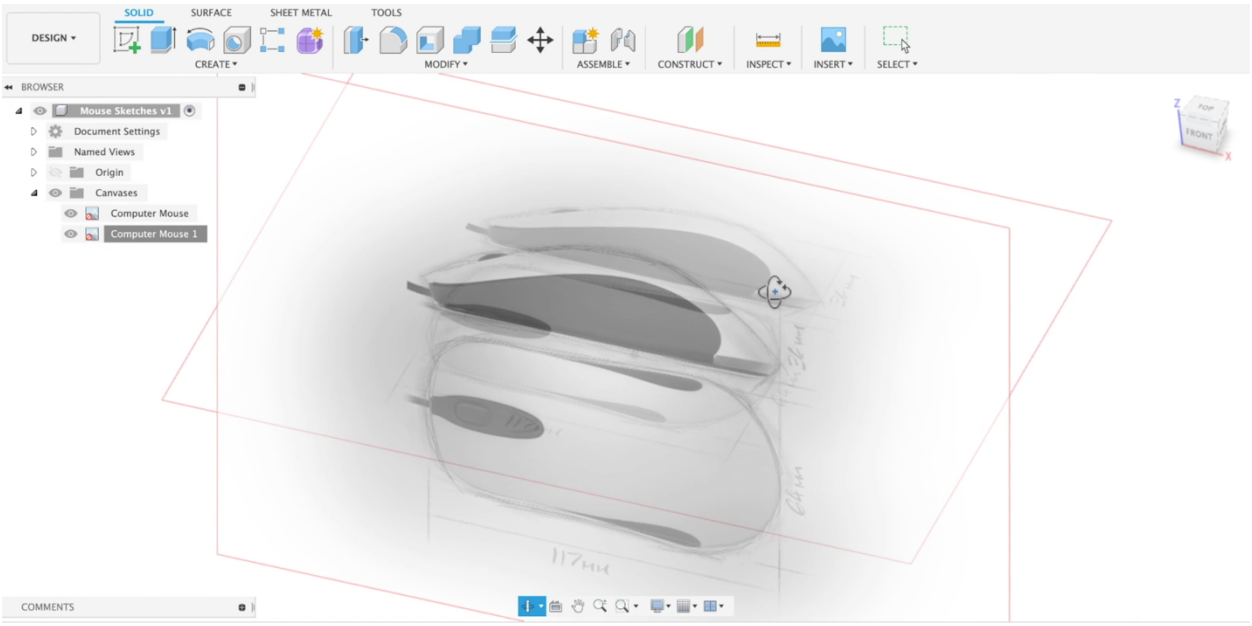
A CONCEPT SKETCH FOR A COMPUTER MOUSE

Setting up the canvases in Fusion 360

Now open Fusion 360. We are going to bring in our concept drawing and use it as a canvas off which to base our design in Fusion. In the Fusion data panel, select upload to bring the computer mouse concept drawing into Fusion. We can now insert it as a canvas. In the solid modelling tools, select "Insert", and then "Canvas" from the toolbar You can now select the file you want to bring in and the plane on which you want to place it. Don't worry about the size yet, we can resize it properly in a minute. It is worth saving your file often as you are working. You can give the file and name and save it into a Fusion project (which you can access in your data panel on the left hand side of the screen). Once you have inserted the canvas, right click on the canvas in the part browser and select "Calibrate". Using calibrate, you can select two points on a canvas and define the real distance to scale the canvas to the correct size.

Aligning the second canvas

We can now repeat this process with a second copy of the mouse drawing to overlay the plan and side views within the workspace as shown below. You can modify the position of a canvas at any time by right clicking on the canvas in the part browser and selecting "Edit Canvas". It may take some time to get the two copies of the canvas perfectly aligned, but there is no rush. Spending some time getting them aligned now will make for a better 3D model as we go along.



ALIGNING THE CANVASES

Modelling the mouse

In order to model the mould for the mouse, we must first create a pattern. A pattern is a replica of the part you wish to model. For example, in sand casting, the pattern is put in the centre when packing the sand so that, when it is removed, the space remaining in the sand gives the shape you wish to mould. Here we will be modelling a virtual pattern in order to model our mould and create the CNC toolpaths for it.

Creating the initial form

To model the mouse, we will be using the form workspace. The form workspace allows for much greater control and flexibility over the organic shape of your model. In this case, since the mouse will have curved surfaces to be ergonomically comfortable, the design workspace would not give us the ability to manipulate the surfaces as we need.

"Create Form" from within the design workspace.

This can be found within the create menu and has a distinctive purple colour. This will bring up a new workspace that contains all of the tools required to create and manipulate the form body.

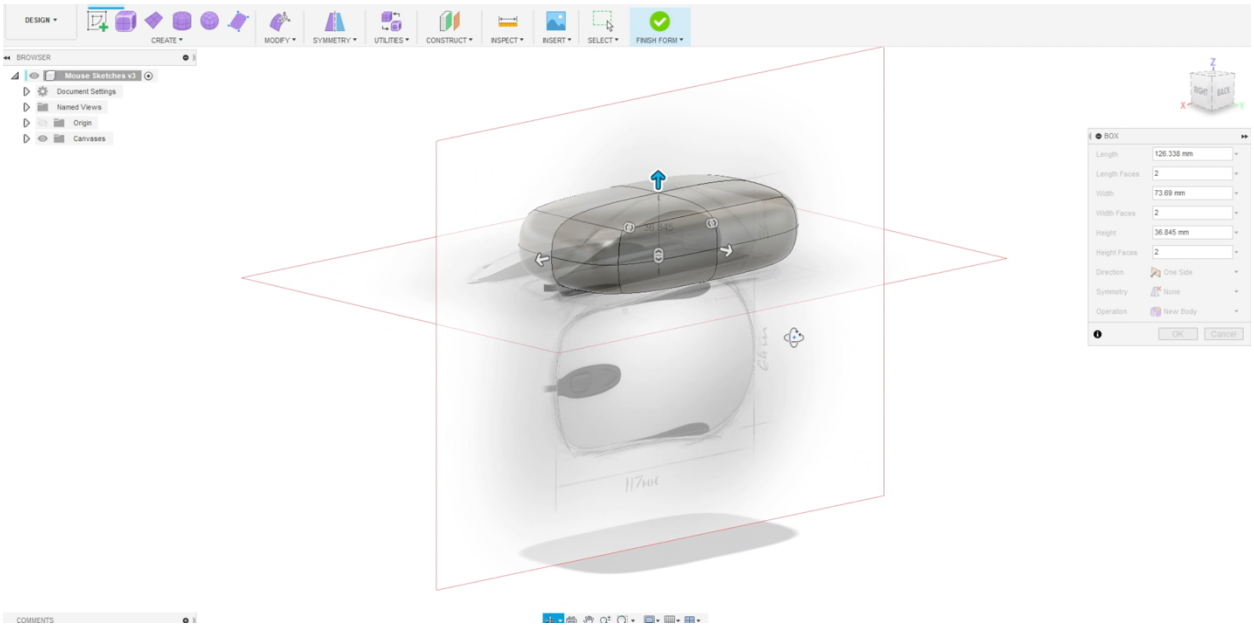
Create a "Box" from within the create menu.

Select the XY plane as the drawing plane for the X & Y dimensions of the box. The box tool creates a three dimensional shape by drawing two dimensions in the axis parrelel to the plane selected, and then once clicked, brings it into 3D with a third entry.

Draw the centre rectangle from the origin point

Draw the rectangle so that it closely surrounds the mouse in the canvas. Specific dimensions are not required as the shape will be manipulated later in the process. Click once the correct outline is drawn. A 3D shape should appear that should fit the side profile of the mouse fairly closely, you can adjust your view using the view cube in the top right hand corner or by using Shift and the middle mouse button. If the box's height does not roughly match the mouse's, you

can adjust the height of the box before clicking “OK”. Remember the box will be manipulated later so it does not need to be exact.



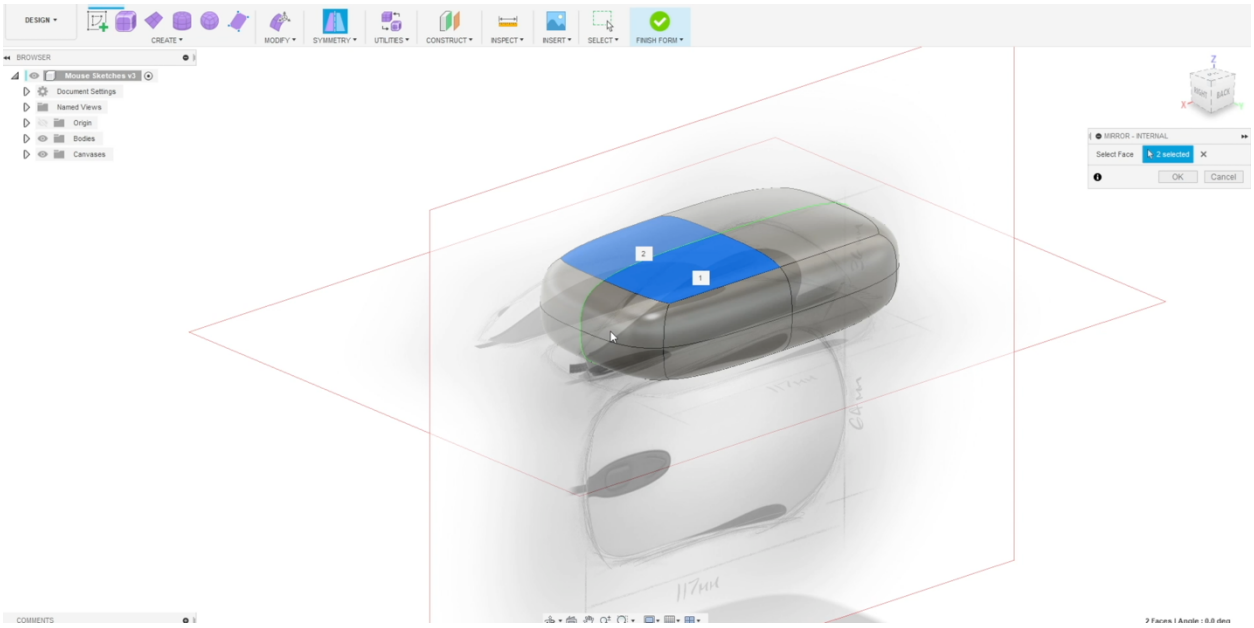
GENERATING THE INITIAL FORM

Modifying the initial form

To end up with the correct final shape, the created box must be shaped and moulded to match that of the mouse. The canvas can be used to make it easier to match the shape of the mouse to the shape of the model.

Add symmetry to the model.

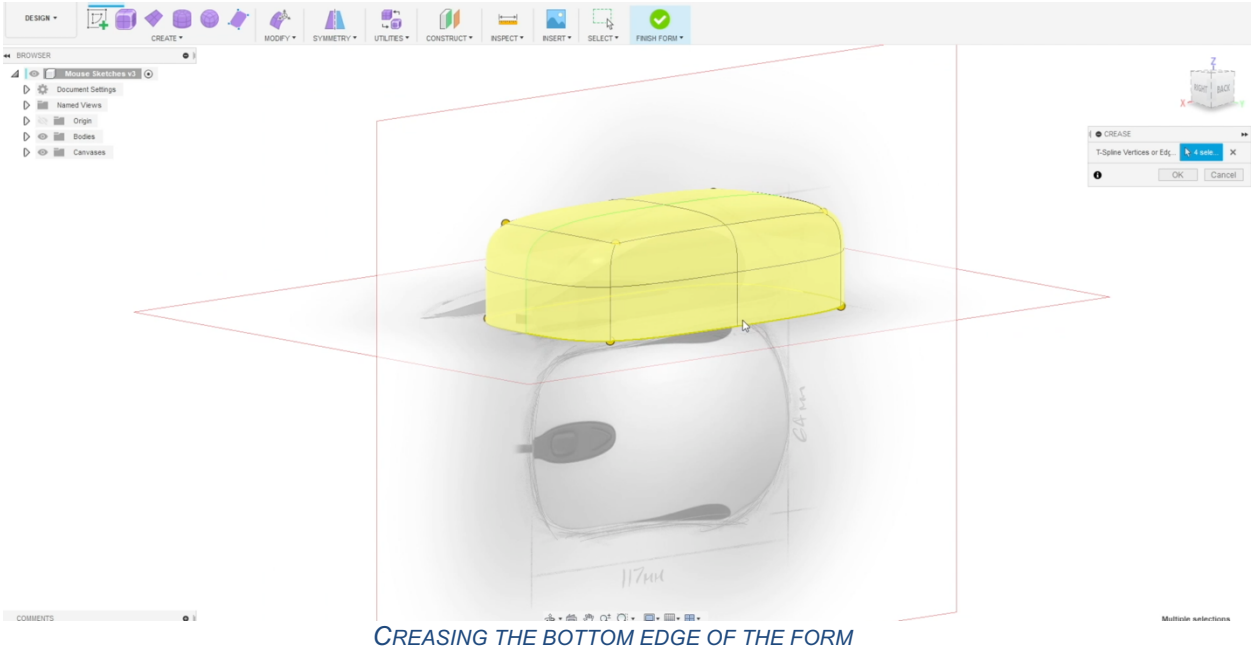
The mouse that needs to be modelled is symmetric down it’s length. Given how undefined modelling in the form workspace is, it would be almost impossible to match both sides if they were to be modeled separately. The symmetry tool is used to overcome this. Once symmetry is applied, any changes made to one side of the symmetry line are automatically applied to the other side. Select the “Mirror – Internal” tool from the symmetry menu. To apply the symmetry, select two faces that are directly opposite each other on the symmetry line you want to create. A green line should appear down the length of the model to indicate that symmetry has been applied.



SETTING UP SYMMETRY IN THE FORM

Crease the bottom edge of the mouse

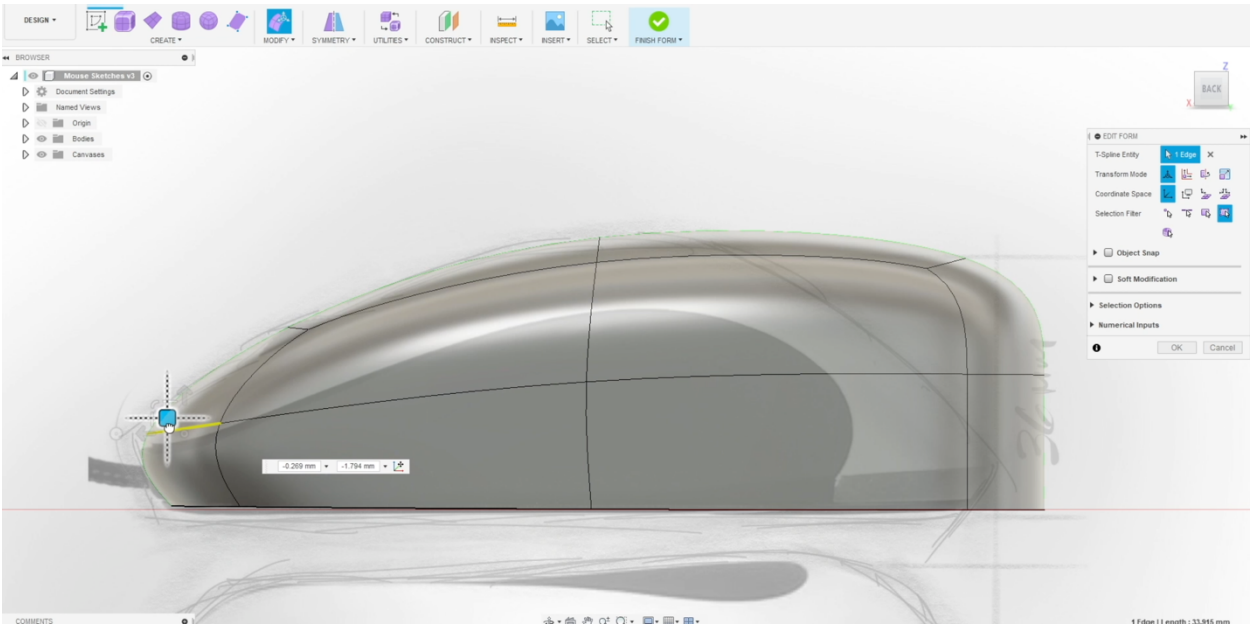
The form workspace is designed to work with smooth and curved surfaces. To break the curves you must use the crease tool. This sharpens the edges that you select to give a crease. The bottom of our mouse requires a sharper edge for where it sits on the table. Select the “Crease” tool from the modify menu and select the edges around the bottom of the box. You should see the symmetry apply the crease to both sides of the box. Ensure to select all the edges around the bottom.



CREASING THE BOTTOM EDGE OF THE FORM

Modify the outline shape of the box to match the mouse

We want to adjust the shape of the box so that it has the same outline as the mouse does from all sides. The “Edit Form” tool allows us to do this. The edit form tool can be used to select either a point, and edge or a face on a form body and once selected, they can be manipulated however is needed. Start by looking directly at the side of the mouse model. Use the edit form tool to individually select edges or faces to move each one to match where it should be on the corresponding mouse canvas. I personally prefer to move the edges as I find that it gives more control however you may have a personal preference as to how you like to do it. I would not recommend moving points however as this often results in an unwanted shape.



FITTING THE FORM TO THE CONCEPT SKETCH

Mould the mouse into the correct shape from the side, top and front views.

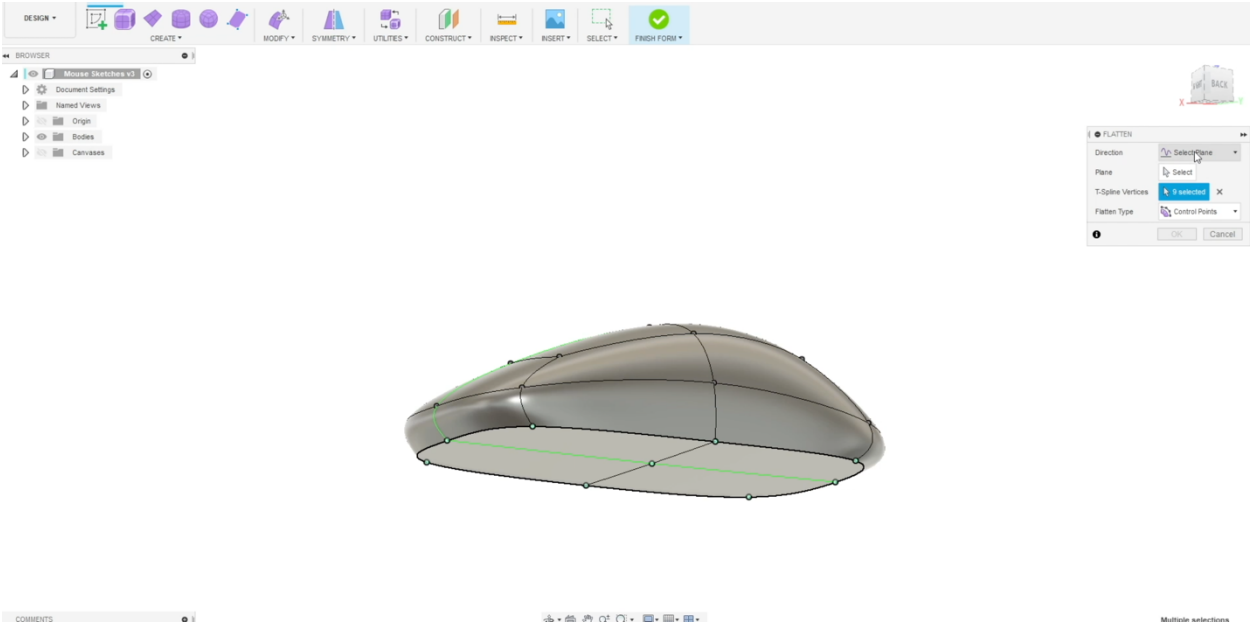
There is no set way to mould the box. You will find that moving edges will adjust the ones you have previously moved. You can select multiple edges or faces by holding shift when selecting them. You will need to return to each edge several times to get the shape right. Having done the side, move to the top and repeat the process. Be aware that edges or faces may be hidden from view when looking down from the top view so you may have to orbit around the model to select the right edges. It is easier to select all edges on each corner so that they remain together. There is no canvas for the front view but it can be adjusted to make it fit the shape of a mouse more closely.

Top Tip:

- When using the edit form tool, the arrows allow you to move the selection in only one axis, the squares allow you to move the selection in the given plane (2 axis). Both can be useful to adjust the model how you wish.
- You can always hide the canvas to get a better look at your model and make sure that it looks correct.
- If you wish to have more control over the box you can use the “Insert Edge” tool in the modify menu. The insert edge tool will allow you to add edges to the box that you can then manipulate and control.

Flatten the bottom surface of the mouse

The mouse has to rest on a table, but by modifying the shape, the bottom surface may no longer be completely flat and would wobble on the table if moulded. To make it flat, we use the “Flatten” tool, which can be found in the modify menu. Firstly, choose “Select Plane” as the direction. This allows us to match the bottom surface to a plane, which in this case we will imagine is our table. Select each of the T-Spline Verticie points on the bottom surface of the mouse (don’t forget the one in the middle!) and select the plane as the XY plane that we originally drew the box on. You should see the bottom surface move to match the plane (even if the movement is very small).



FLATTENING THE BOTTOM OF THE FORM

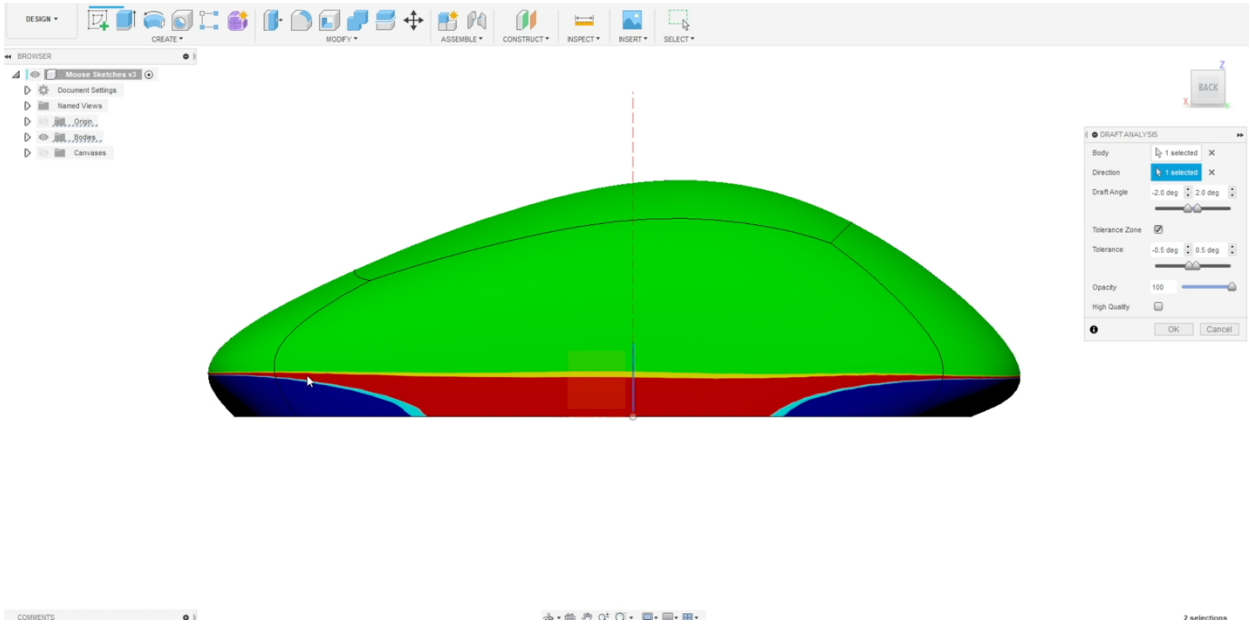
Analysing the mouse

Given that the mouse is to be used as a pattern for a mould, it is sensible to check the draft angles at this point. When moulding a part, there must be a draft angle on each side to allow the part to be ejected from the mould. Without a draft angle the part would get stuck. It is recommended that a 2 degree draft angle is applied to all sides of a moulded part. An analysis

will also allow us to determine the split line between the top and bottom sections so that each can be moulded.

Apply a draft analysis

To apply a draft analysis, click “Finish Form” to return to the normal design workspace. Select “Draft Analysis” from within the inspect menu and select the mouse body. The direction of the analysis indicates the zero angle from which the chosen angles will be rated. Select the Z axis line on the orign as the direction. You may have to hide the mouse body to be able to click on the Z axis line. The pre-set angles should be correct but you can change these to be 2 degrees if they are not already.



CARRYING OUT A DRAFT ANALYSIS

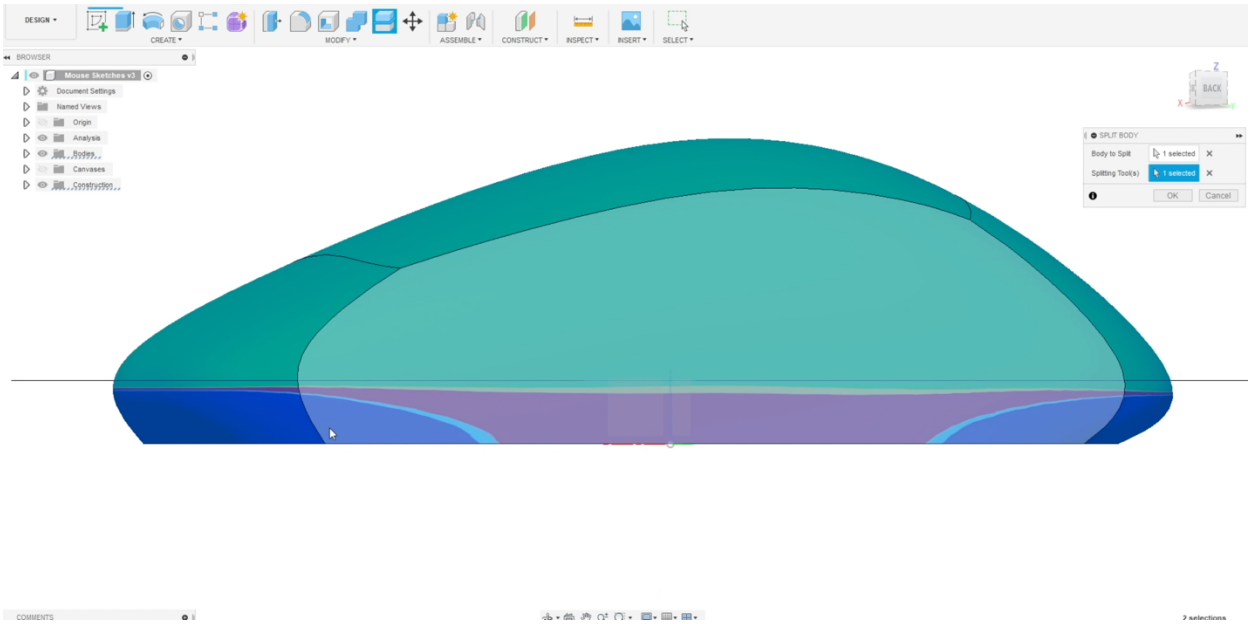
Using the draft analysis

The draft analysis will hopefully show a mostly green top body with blue on the bottom and red on the sides. Blue and Green indicate areas where the draft angle is outside the given constraints of less than/more than 2 degrees. The red areas indicate where the draft angle falls within these boundries. My analysis has a distinctive line around the model, however you may find that yours is different due to the fact that you may have moulded your mouse slightly differently. If the red climbs too far up the side of the mouse, you may need to go back in the timeline and edit the form of the mouse to make the sides slope inwards slightly more.

Split the mouse into two bodies

As we only want to mould the top section of the mouse, we need to split the mouse into its top and bottom sections. Create an”Offset Plane” from the construction menu and click on the XY plane underneath the mouse (this is the plane used to flatten the mouse and draw the box). Drag the plane upwards and looking from the side view, front view and back view, place the plane so that it sits as close to the red and blue areas as possible without meeting them. i.e, just above the coloured line. Press enter to confirm the plane. Select “Split Body” from the modify menu, choose the mouse body and the target body and the plane that was just created as the splitting tool. You can either hide the mouse to select the plane, or hold left click whilst over the

plane to bring up the list of selectable items behind the mouse. This is an easy way to select items even when they are obstructed by something else. Check the red splitting ring to ensure that all draft angles below 2 degrees are below the splitting plane. Click Ok and you will see a second body appear in the “Bodies” folder in the browser.



SPLITTING THE MOUSE INTO AN UPPER AND LOWER BODY

Complete a new draft analysis to check for errors

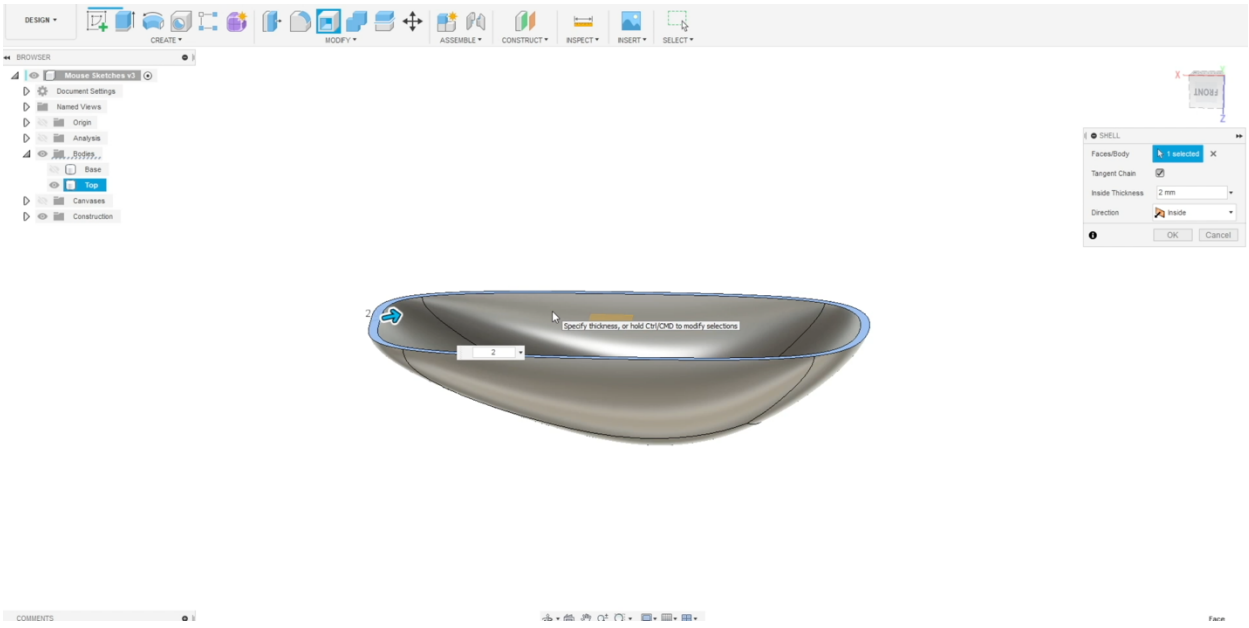
When splitting the body, the draft analysis only remains on the first body, you can rename each body to make it easier to find.. Hide the base of the mouse and apply a new draft analysis on the top of the mouse to double check that there are no draft angles that would cause the mould to have a costly fault.

Complete the model

Final touches are required to complete the model. It must be made hollow and a slot cut from the centre to allow for the mouse wheel.

Shell the top case

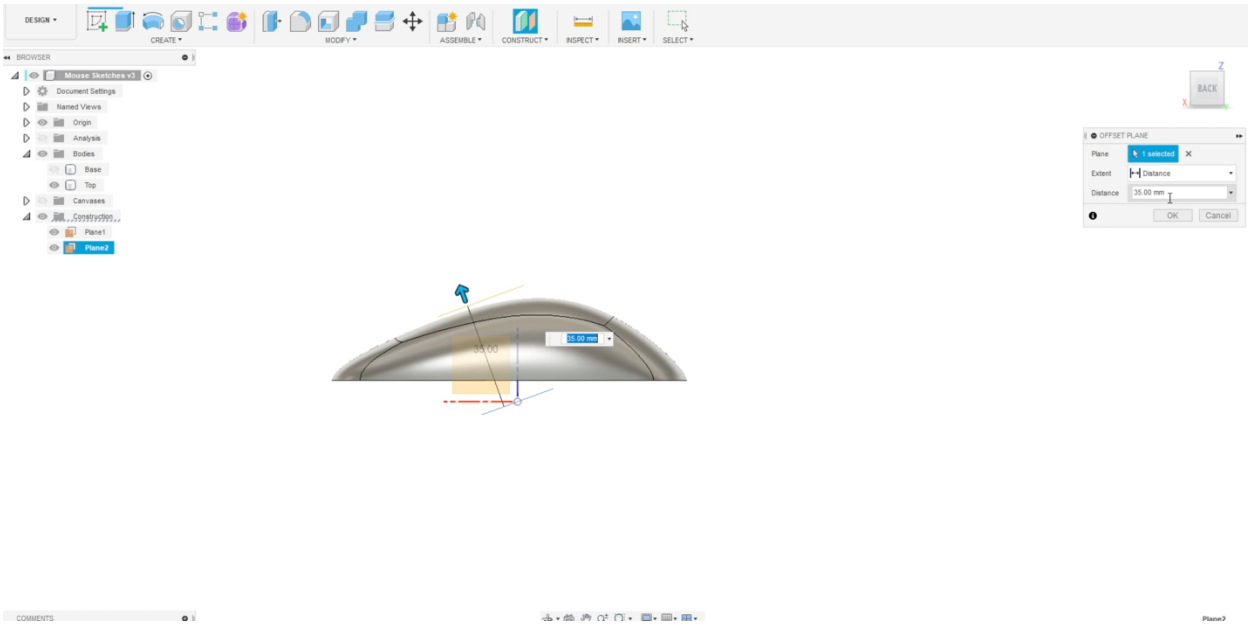
The part to be moulded is not a solid block of plastic but must be hollow to allow for the electronics to be installed post-moulding. Select the “Shell” Tool from the modify menu and select the bottom face of the mouse. Enter 2mm as the desired thickness. The tool will remove all material in the model leaving 2mm remaining on each edge. This also ensures that the part has a constant thickness of material throughout the part allowing for it to cool at the same rate when moulded and not cause internal stresses.



SHELLING THE UPPER BODY OF THE MOUSE

Set up a sketch plane for the mouse wheel slot.

To cut a slot, a sketch must be made on the surface of where the slot must go. However there is currently no plane on which to put the sketch that matches the front of the mouse so we must create one. First, create a “Plane at Angle” from the construction menu and select the axis around which we want to rotate the plane, in this case the Y Axis so that we can slant a plane at the same angle as the front of the mouse. Change the angle to match the mouse, it may be easier to look at the mouse directly from the side for this. For my mouse this angle is 20 degrees but yours may be slightly different. Whilst this plane is now at the right angle, it is in the wrong place, so we can create an “Offset Plane” from the construction menu and offset the plane at angle just created such that there is a plane at the correct angle, on the front of the mouse to sketch onto.



SETTING THE PLANE FOR THE MOUSE WHEEL HOLE

Create the slot

Create a new sketch on the surface of the offset plane just created. First, we must ensure that the slot is located in the centre of the mouse but when creating the sketch the origin of the sketch will be different to the origin of the mouse. To reference the mouse's origin, select the "Project" tool from within the create menu (Project is found at the bottom within the "project/intersect" sub-menu). Select the origin point for the mouse to project it onto the sketch plane we are using. Draw a line from the point straight along the length of the mouse and apply the horizontal/vertical constraint if it does not do so automatically. Ensure that this line is a construction line by selecting the construction button at the top of the sketch pallete. Select the "Slot" tool from the create menu. I am choosing to use a centre to centre slot but any of the first three options will work. For the centre slot, the length must be 15mm and the diamneter of the ends must be 6mm. Draw this towards the front of the mouse with the centre of the wheel matching roughly with the edge that is still visible on the mouse model. This can be moved later by editing the sketch if you feel like it could be better placed. Click "Finish Sketch" and select the "Extrude" tool from the create menu. Select the inside area of the sketch and drag it down through the model. The area will turn red to indicate that the tool is being used to cut. From the dropdown, change the cut from "Distance" to "All" to ensure that all visible parts of the model are cut. Click OK to cut the slot from the mouse.

Chamfer the slot

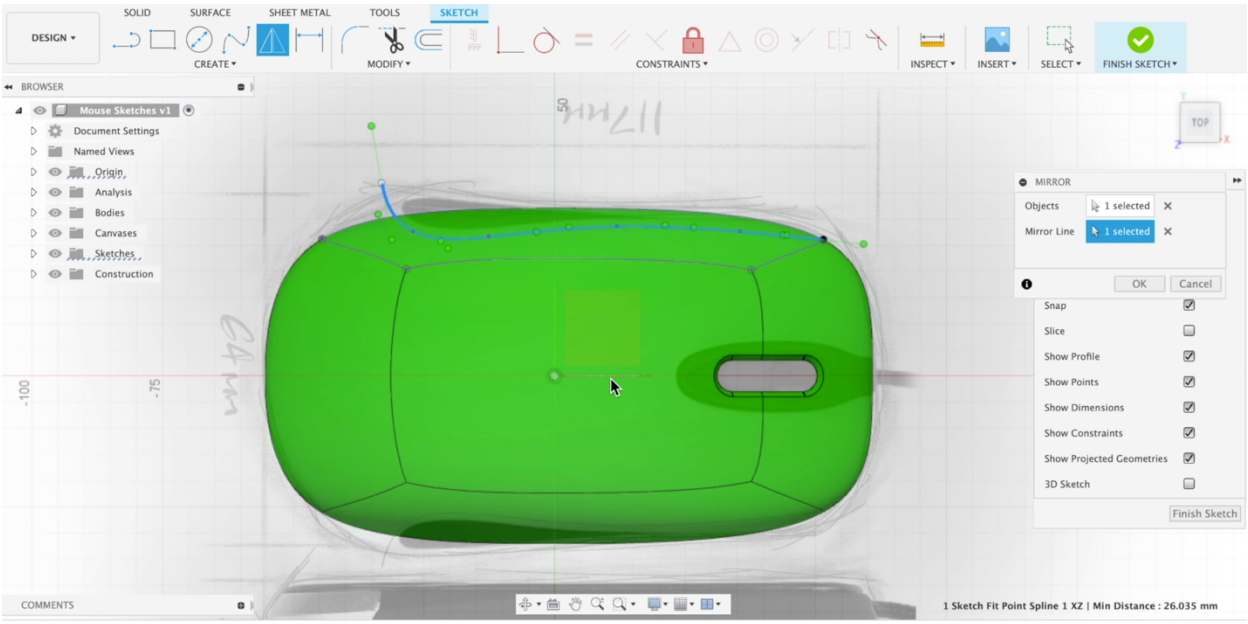
Turning on the draft analysis will reveal that the slot causes an angle and overhang that cannot be moulded. To overcome this, we must add a chamfer to either sides of the slot to make the angle wide enough to be moulded. Select "Chamfer" from the modify menu and select the edge around the slot on the inside of the mouse. Apply a chamfer of 1mm and click OK. The draft analysis should change to become blue and indicate that the angle is outside of the range selected and can therefore be moulded. However there will still be red where the chamfer has not reached. Apply a second chamfer on the outside edge of the slot, also to 1mm, to remove remove the problematic draft angles.

Rendering

We are now going to look at creating some visuals of our mouse design. To do that, we need to modify some of the faces of the mouse to make it more closely resemble our concept drawing then and apply some appropriate appearances.

Split the faces of the mouse

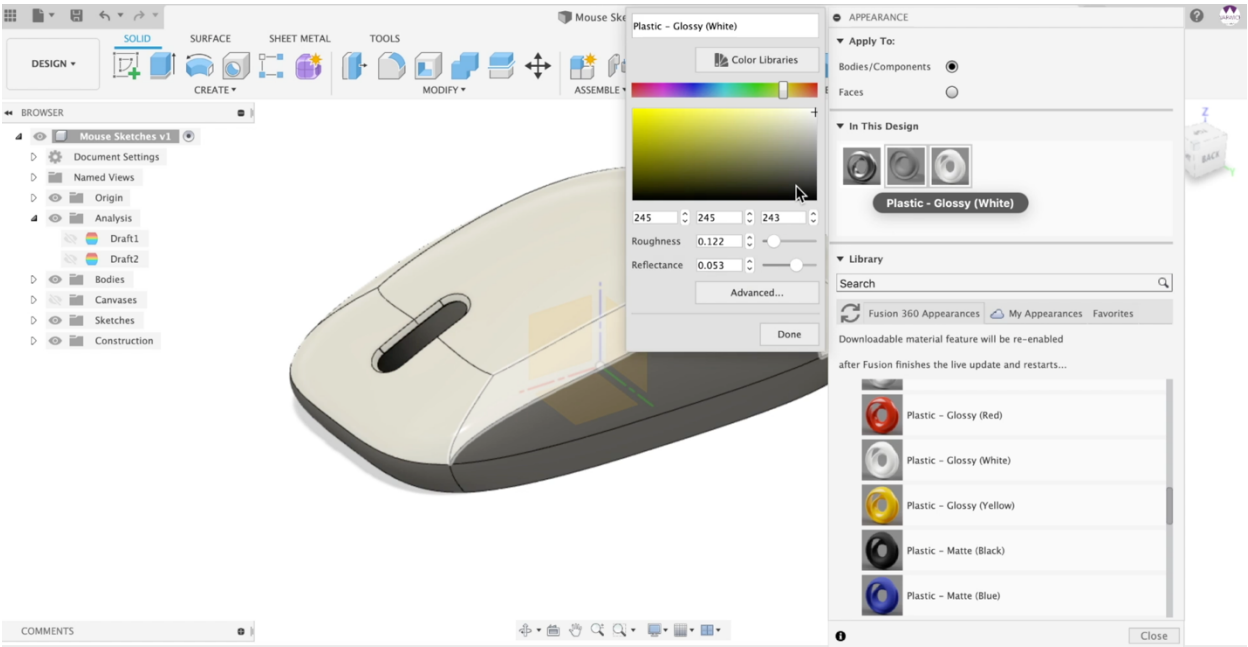
Select "Create Sketch" from the toolbar and select the top plane. The sketch tools will now appear. If your plan view canvas visibility is switched off, you can turn it back on to assist with this step. Using the canvas as a guide, draw a spline (located above "Create" in the sketch tools) roughly approximating the dark area on the side of the computer mouse drawing. It may take a couple of attempts to get this right. Once you are happy with your spline, select "Mirror" from above the 'Create" menu and mirror the spline you have just created over to the other side of the mouse, using the appropriate axis/plane from the origin. You can now finish the sketch. Finally select "Modify" and "Split Face' and then select the face at the side of the mouse and one of the splines you have created as the splitting tool. You will need to repeat this process for the other spline. The sketch visibility may have been toggled off after the first splitting operation, so it may need to be turned back on again to complete the second splitting operation.



SPLITTING THE FACES OF THE MOUSE

Apply appearances

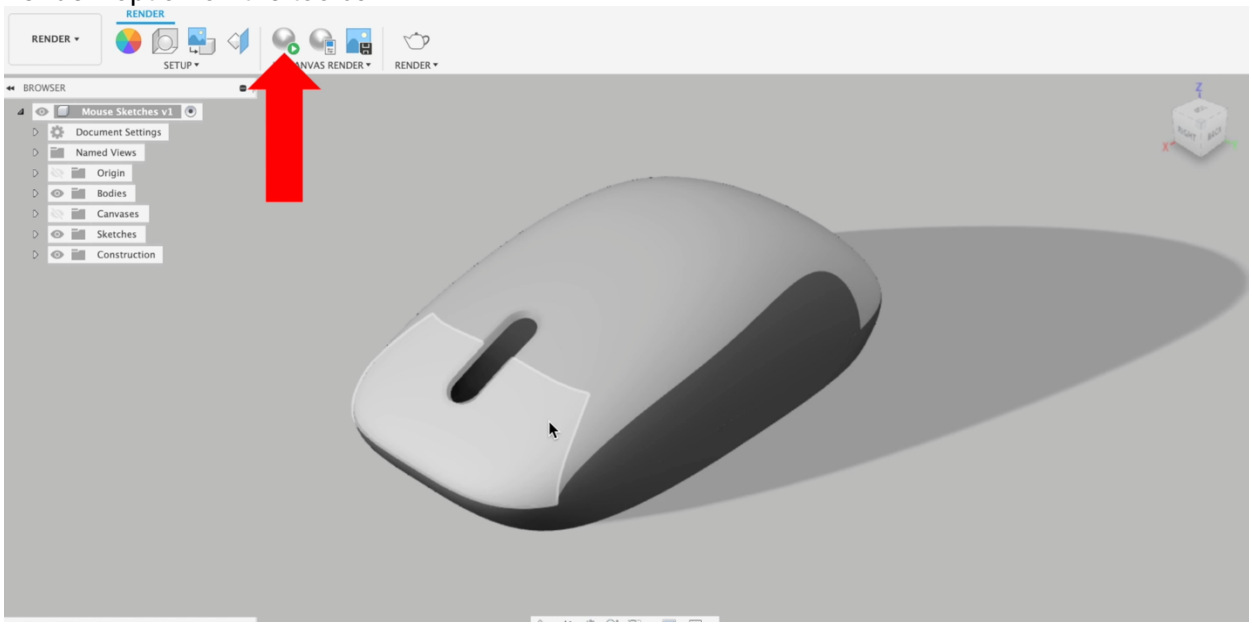
Now we have the split the faces of our mouse design, we can apply some appearances. You can quickly access the appearances menu by hitting the “A” key on your keyboard or selecting “Modify” and “Appearance”. Within the box that appears you can scoll and select a material appreance to drag and drop onto your mouse model. At the top of the box you will see the option to add the appearance to “Bodies/Components” or “Faces”. In this case we apply a lighter coloured material to the whole mouse body and then a darker grey plastic to the specific faces that we divided up in the previous step. You fine tune and edit colours and properties of materials by double clicking on the material in the “In This Design” box or right clicking on the material and selecting “Edit”.



APPLYING AND MODIFYING THE APPEARANCES

In-canvas render

Now we have applied materials and are happy with the appearance we can jump into the render workspace but selecting “Render’ in the workspace menu in the top left hand corner. There are lots of settings you can change when doing a render but to carry out a simple ray-tracing render you can click on the icon with the green arrow above “In-canvas Render” (as shown by the red arrow below”. This will begin a ray-tracing operation which may take a few minutes to complete. You can see the progress with the blue bar in the bottom right hand corner. Fusion 360 also has the capability to do a cloud render, which you can access with the teapot icon above the “Render” option on the toolbar.



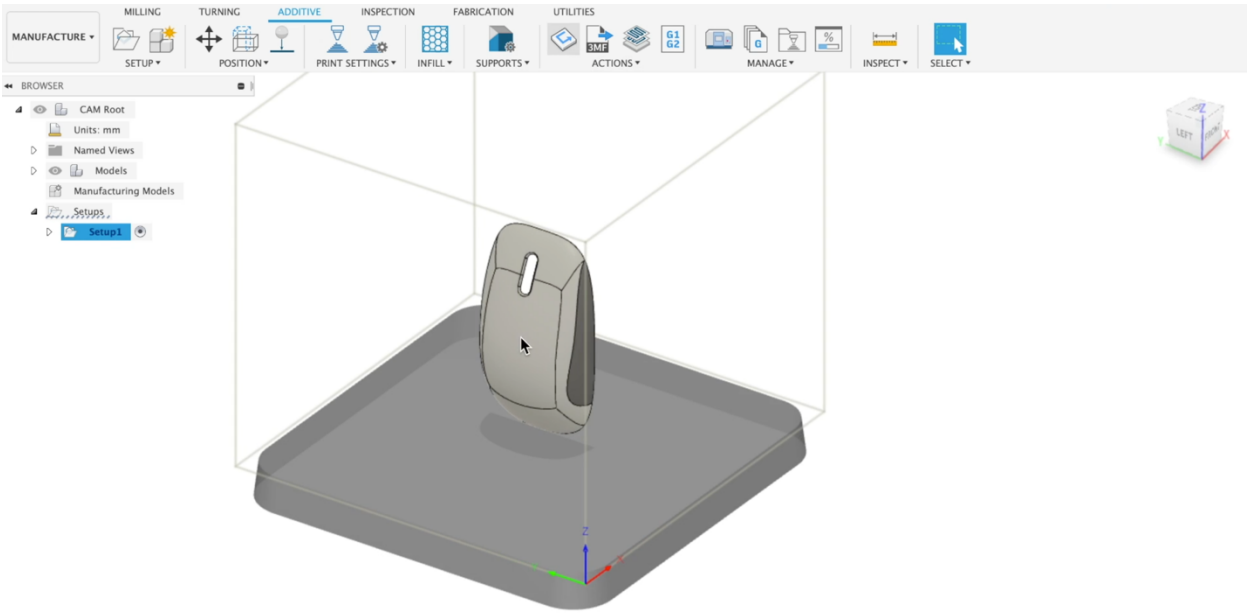
STARTING AN IN-CANVAS RENDER

Manufacturing

Next, we’re going to look at some of the various options for manufacturing our design. First up, we will look at the tools in Fusion 360 that would allow you to 3D print a prototype of your design.

3D printing setup (optional step)

We will briefly look at how you could go about 3D printing a prototype of the mouse. Click on the “Manufacturing” workspace. Select “Additive” from above the toolbar and select “Setup” from the “Setup” section on the toolbar. Set “Additive” as the operation type and select the top of the mouse as the model to be used. For the “Machine” option, select the machine that matches your 3D printer. Fusion 360 has a library of machines to choose from. Once you have selected a machine, click “OK”. You can work along the toolbar changing the orientation of the part and selecting appropriate print settings.



SETTING UP AND ADDITIVE MANUFACTURING JOB

To complete generation of the 3D printing job, select “Generate” in the “Action” section of the toolbar. You can preview/simulate the 3D printing process by selecting “Simulate Additive Toolpath” again in the “Action” section of the toolbar. To complete the export of the 3D printing job, select “Post Process” and the appropriate post processor for your machine.

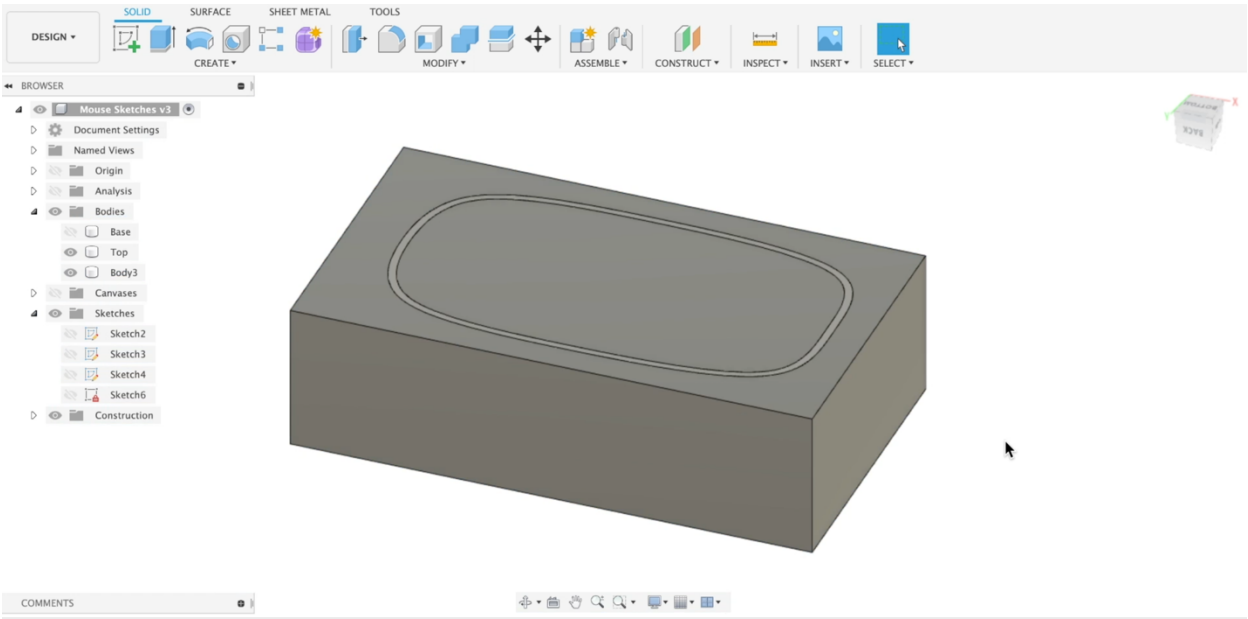
THE 3D PRINTED PROTOTYPE

If you wish to take a look at a more in-depth demo of the Additive Manufacturing workflow in Fusion 360, please see the instructional demo:

Fusion 360 and 3D Printing—Tips and Tricks for a Successful Workflow - MFG463366

Making the injection mould

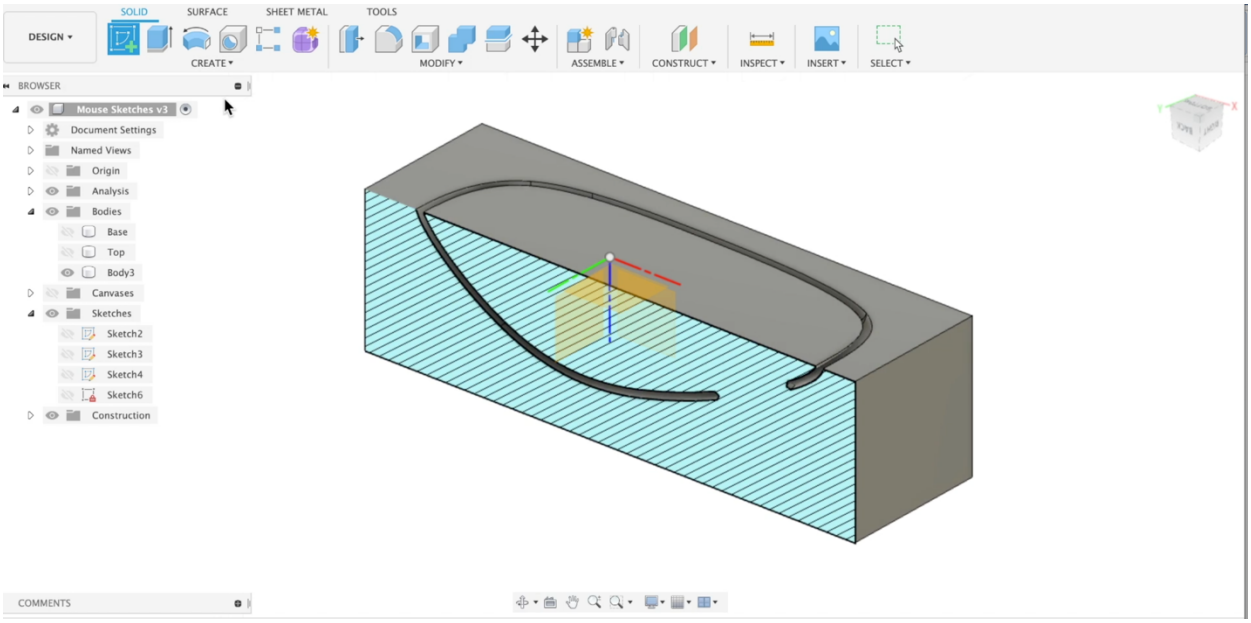
We began this process knowing we would want to injection mould the top casing of our mouse design, so we will look at the steps needed to design a simplified representation of a mould set and CNC machine it. Rotate the mouse casing and create a new sketch on the bottom face. Using a centre point rectangle, create a rectangle of the size required for your mould. Finish the sketch and then use the “Extrude” tool to extrude the box above the top of your mouse case. It will probably be about 40-45 mm in height. Make sure to select “New Body” in the extrude dialogue box. You should now have something similar to the below image. If you don’t select “New Body” you may end cutting away the mouse.



THE INJECTION MOULD WITH THE MOUSE UPPER BODY

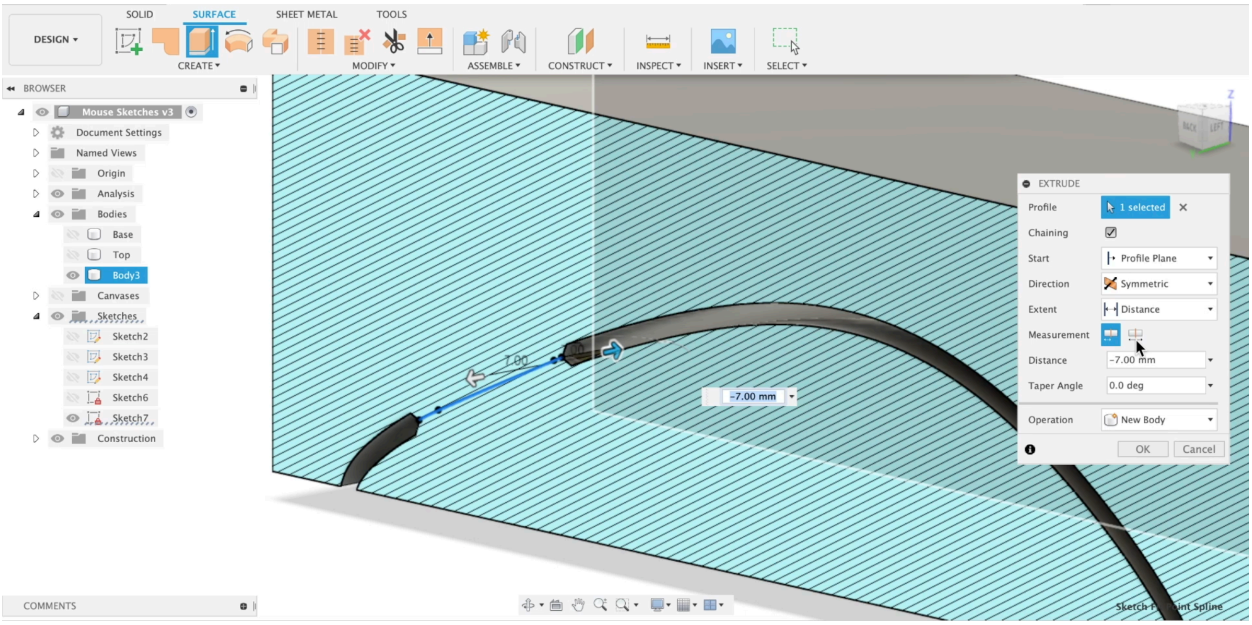
Splitting the mould halves

Now we need to split the mould into two pieces. It is useful now to turn off the visibility of the mouse in the browser. If you click on “Inspect” and “Section Analysis”, then on the side of the mould or one of the planes, you can then drag the section and carry out an analysis of the mould you should see something like the below image. The two parts of the mould are connected because of the hole left for the middle mouse button in the casing. We will now create a new sketch through the middle of the mould. In that sketch we now want to draw a spline/line that represents the splitting line of the two halves. Where you draw the line is not particularly important for this simplified mould.



A SECTION VIEW OF THE JOINED MOULD HALVES

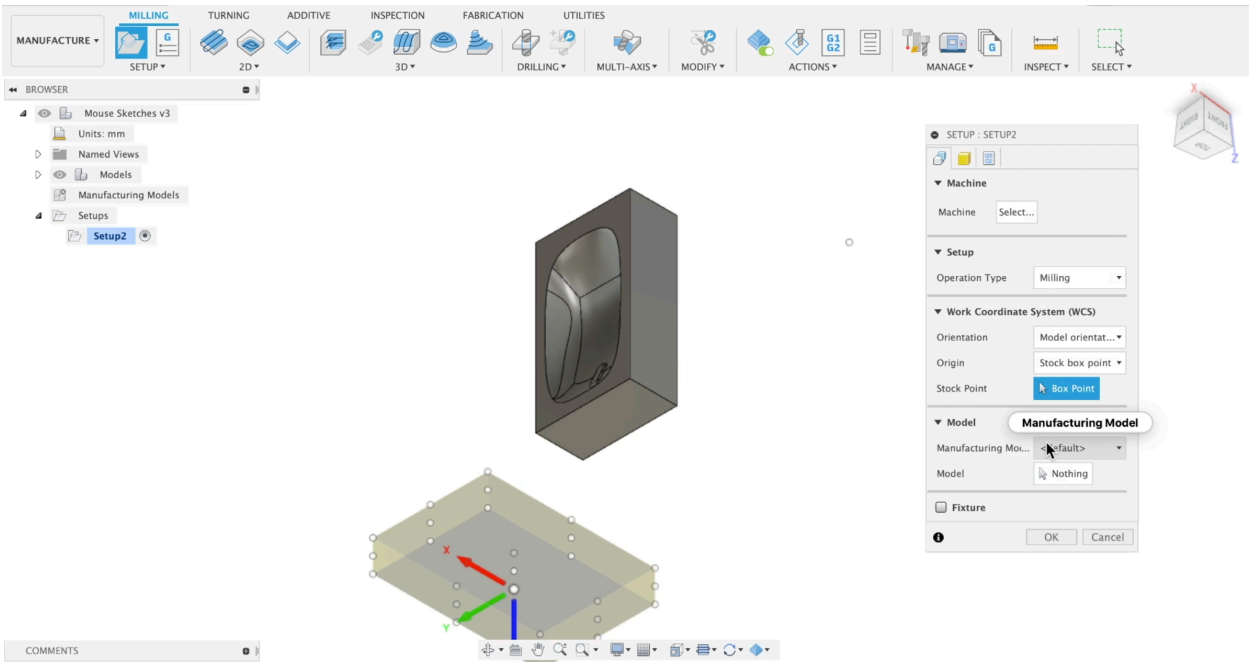
You should now change to the surface workspace by clicking “Surface” above the toolbar. Using the spline/line you just created, you should now select “Extrude” above the “Create” label in the toolbar. Set the “Direction” to “Symmetric” and drag the surface just beyond the extent of the join between the two halves of the model. Click “OK” and change back to the “Solid” tools. Do not turn off the section analysis yet. To complete splitting of the mould halves, select “Modify” and “Split Body”. Select the mould tool and the new surface you created as the “Splitting Tool”. Clicking okay should now split the mould in two and give you two bodies corresponding to the male and female tools in the browser on the left hand side.



SPLITTING THE TWO HALVES OF THE MOULD

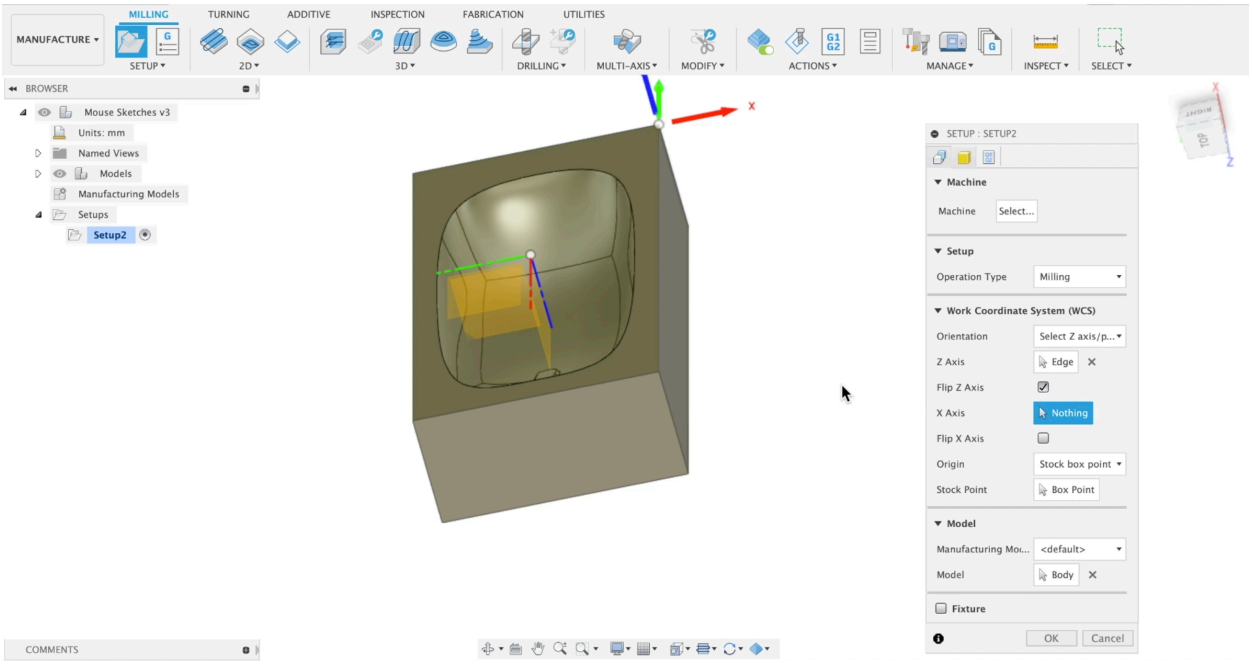
Setting up for CNC Machining

We will now look at setting up a CNC job for manufacturing the female part of the mould set. Select “Manufacture” from the workspace selection option in the top left hand corner. Once in the workspace, make sure that ‘Milling” is selected from above the toolbar. As with a lot of the workflows in Fusion 360, it is best to work left to right along the toolbar. First we need to create a new setup. Click on the “Setup” icon above the word “Setup” on the toolbar. In the setup box that appears set the “Manufacturing Model” to be the be the mould by clicking on it.



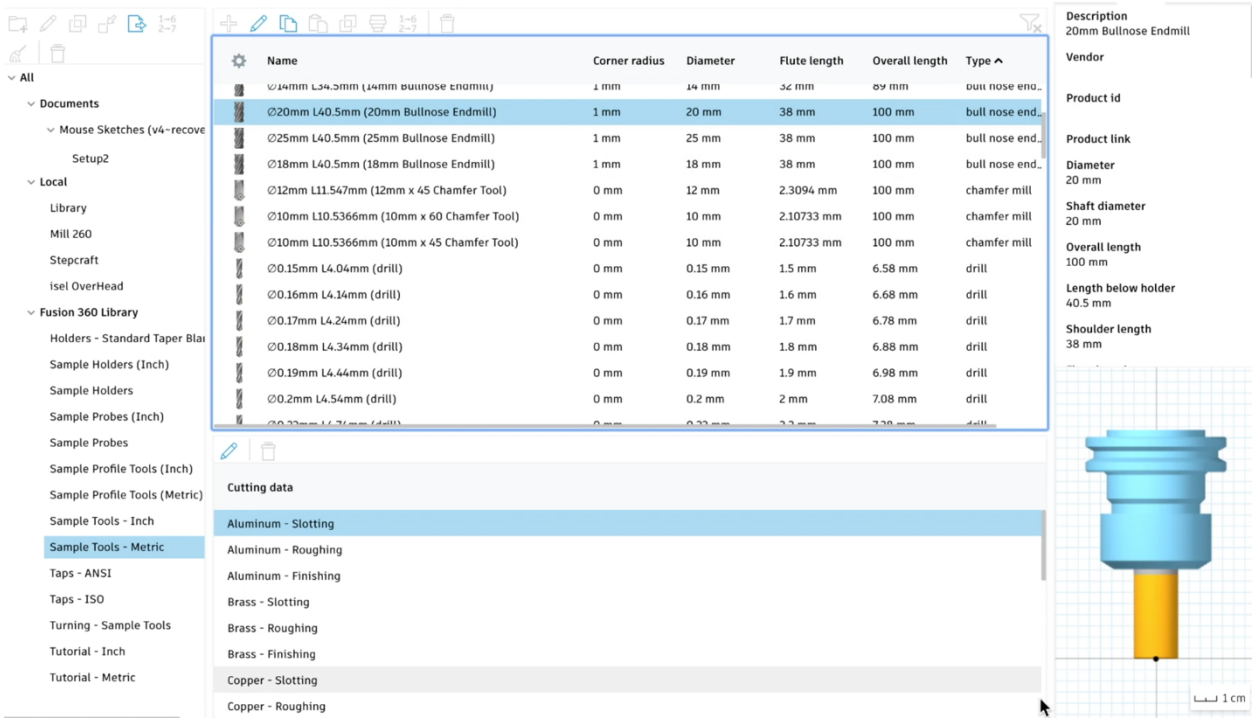
SETTING UP THE CNC JOB

We will now set up our work coordinate system. This setting will tell Fusion the orientation you want set up the machining block and set the coordinates used in the CNC machine. These settings will vary based on your CNC machine and how it is set up. In this case we will set Z pointing upwards from bottom to top of the mould and then X and Y in the plane of the mould surface as per the image below. We also need to specify the point of the block that we will call 0,0,0 for the machining operation. Again, this will vary based on your CNC setup but in this case we will just specify the top corner of the mould tool.



DEFINING THE WORK COORDINATE SYSTEM

Finally, in the second tab of the “Setup” box, specify that the stock mode is “Fixed size box”. By doing this, we are essentially telling Fusion that the block we are maching the mould from is already the correct size and we won’t need to remove any material from the outer sufaces of the block and we are just going to remove the material that represents the shape of the computer mouse.



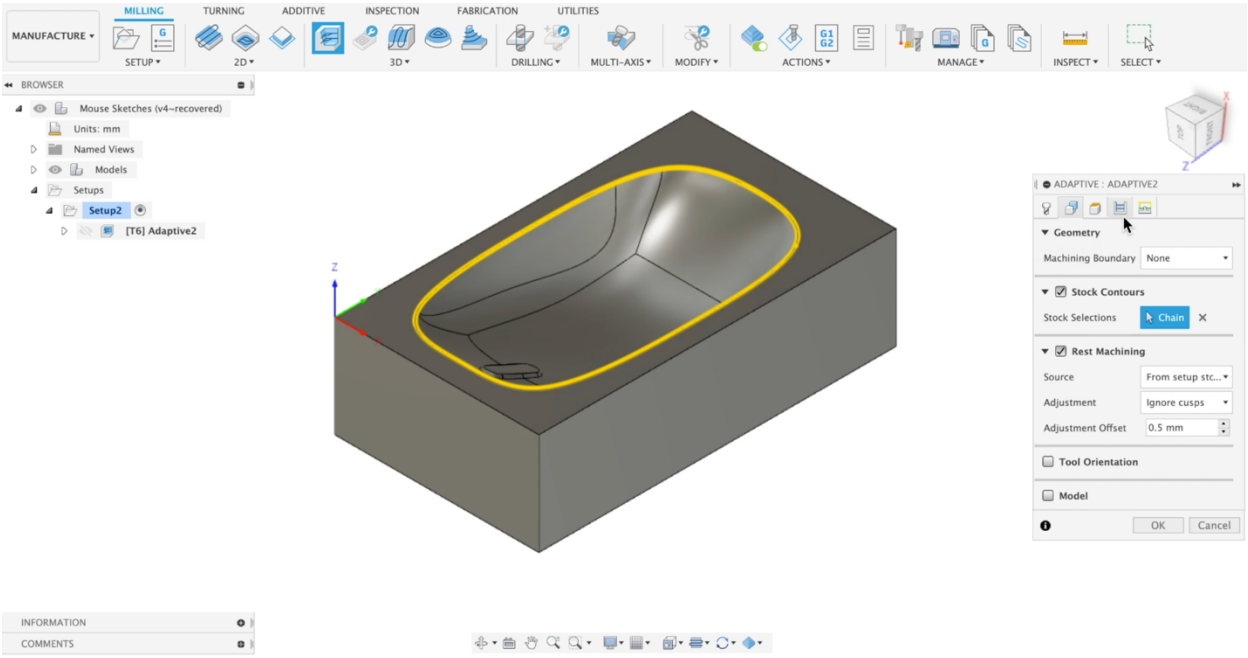
SELECTING A TOOL IN THE TOOL LIBRARY

Toolpath and Tool Selection

For our first toolpath, we are going to remove the bulk of the stock material. To do this we will select a fairly large/coarse tool. Select “Adaptive Clearing” (the first toolpath option above “3D”). In the box that appears we will need to select the tool we want to use. Click next to “Tool” to select a tool. In the box that appears, click “Sample Tools – Metric” and arrange the list by type. For our first operation we will select the 20 mm Bull Nose Endmill with an overal length of 100 mm.

Toolpath Generation

Don't worry about the other options in the "Tool" tab, they will be specific for the tool, material and CNC machine you are using. Next, select the second "Geometry" tab and select the outer profile of the top of the mouse as per the image below. You can work through the remaining tabs in this menu, setting working heights for machining and details of the passes you wish to make. When you are happy with the settings, click "OK" and Fusion will start generating a toolpath.

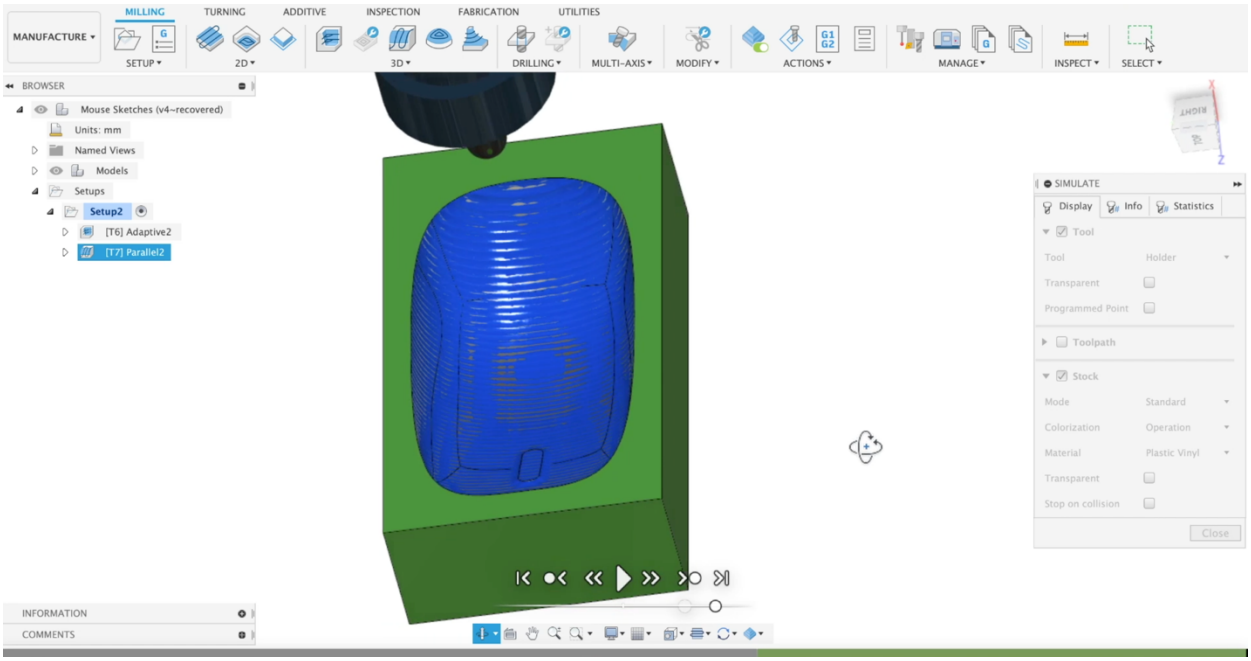


SELECTING THE STOCK CONTOURS TO MACHINE

You can see this from the percentage value next to the operation details under setup in the browser on the left hand side. Next we will start to look at a finishing operation. Click "Parallel" above the "3D" option again. Follow the same process as above but this time selecting a 12 mm ball endmill. In order to control the finish achieved with this step you can control the 'stepover' of the machining operations in the 'Passes" tab. Simply, a smaller stepover value will lead to a better surface finish. Again, click "OK' to generate a toolpath.

CNC Simulation

Once you have a roughing and finishing toolpath created, click on "Setup" in the browser and then "Simulate" under the "Action" section in the toolpath. By doing this, we are going to visualise the toolpaths we have created and what the effect will be on the stock. In the box that appears you may need to select "Stock" to make the stock visible. Clicking the play arrow at the bottom of the screen will start the toolpath simulation, simulating them in the order we wish them to be done.



SIMULATING THE TOOLPATHS

From there you can now go and edit the toolpaths if you wish to change settings like the tool or stepover. In order to then prepare these CNC toolpaths for your CNC, you can right click on “Setup” again in the browser and select “Post Process”.

Other Fusion 360 Resources at AU2020

If you wish to have a go at a more in-depth hands-on demo of CNC/CAM toolpath generation in Fusion 360, please see the hands-on demo:

The Secret That Doesn't Need Keeping: CAM Is Easy with Fusion 360 - MFG463381-L

If you wish to have a go at simulation in Fusion 360, please see the hands-on demo:

Stressing Out: Simulation Workspace in Fusion 360 - MFG463370-L