



Rubber Band Powered Car Fusion 360 Tutorial



This tutorial was created by Emma Prescott during an Undergraduate Summer internship at WMG

Overview

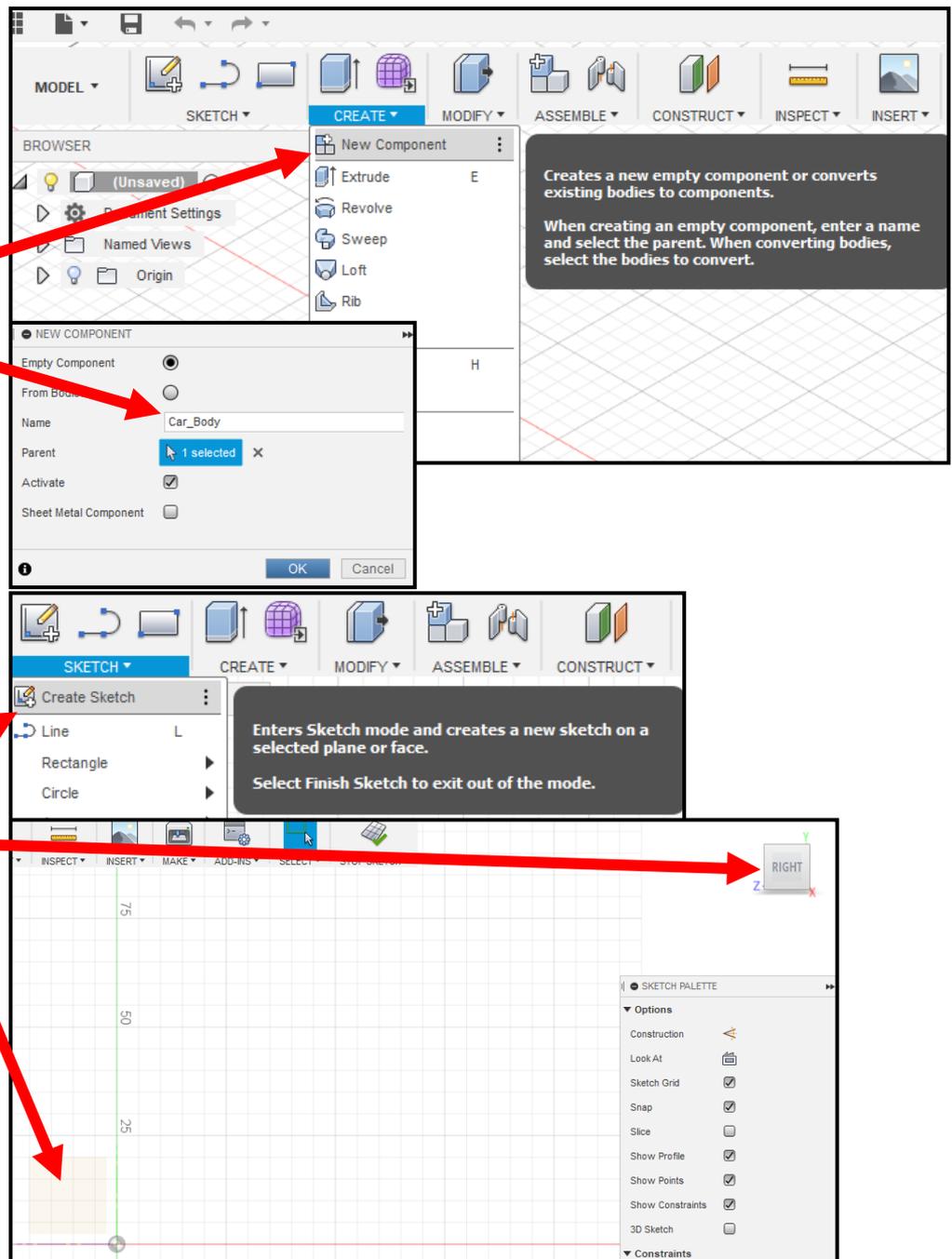
This activity involves creating a 2D template for a rubber band powered car. This design can be used to create a real life model that can be printed out using a laser cutter. If a laser cutter isn't available a sheet of paper can be printed out to use as a template for cutting out the parts by hand. Recommend

The material used to create the car in this tutorial was 3mm plywood suitable for a laser cutter to use. Plywood was chosen as it is light, cheap and strong.

Creating the Body

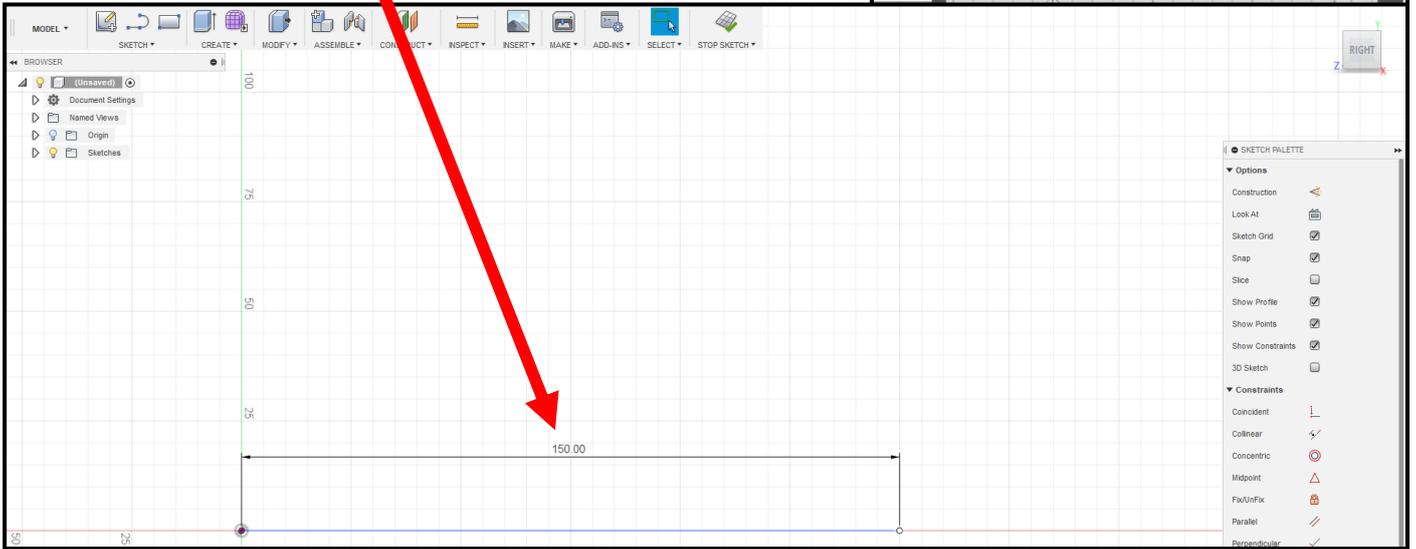
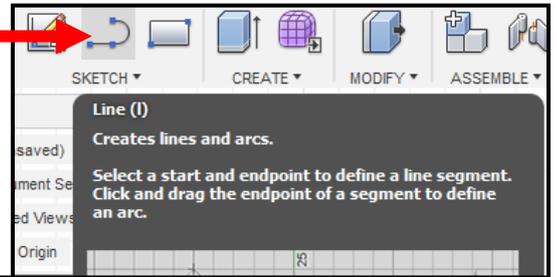
1. Begin by creating a new component and naming it "Car_Body". Confirm by clicking on 'OK'.

2. Now create a new sketch and select the right plane to begin sketching on it.



3. Select the **line tool** from the 'Sketch' tab and select the origin as the beginning of the line. Make sure the lightbulb is lit up next to Origin to make it visible.

Set the line length to **150 mm**, this will be the

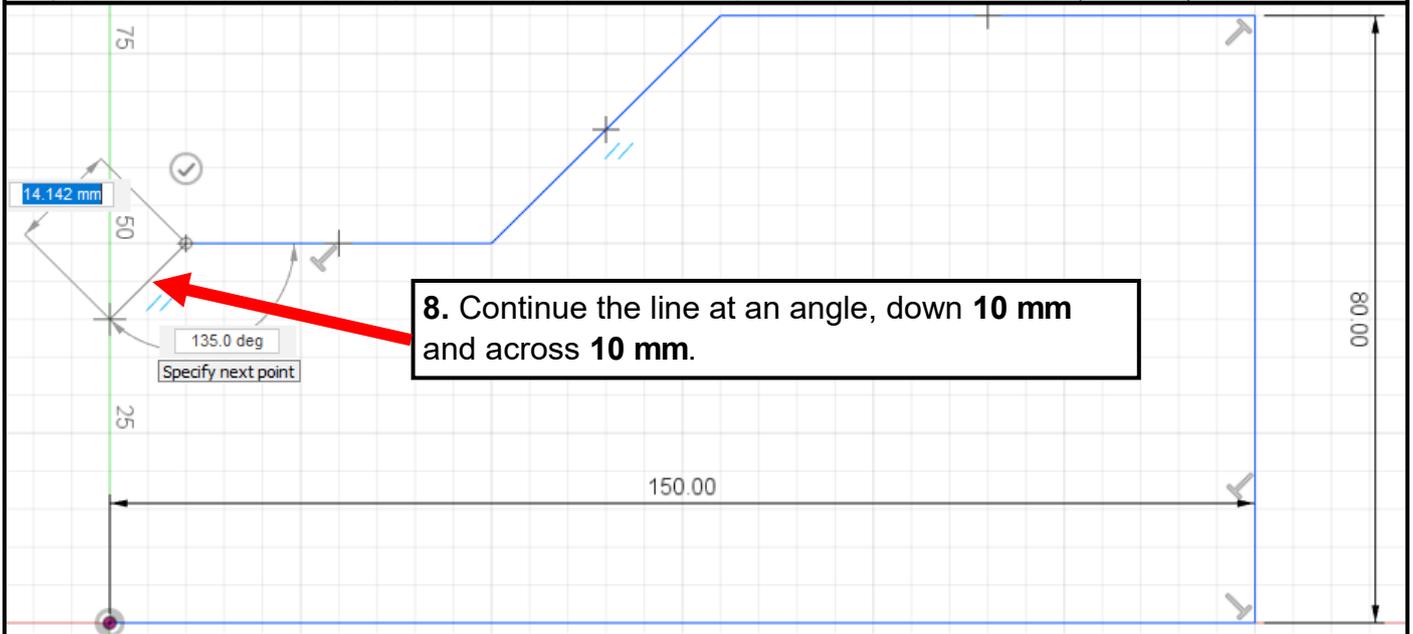
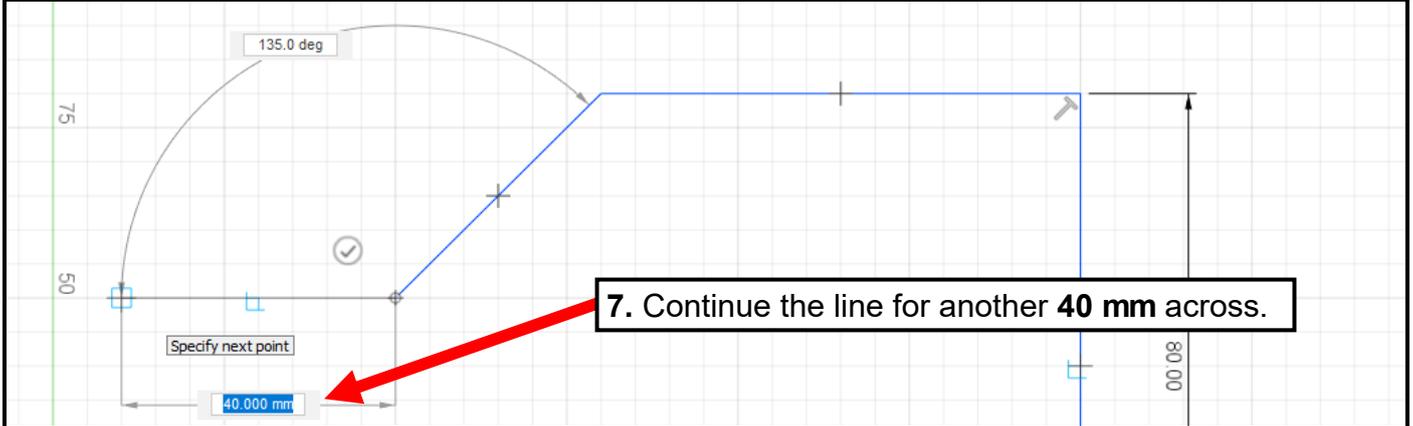
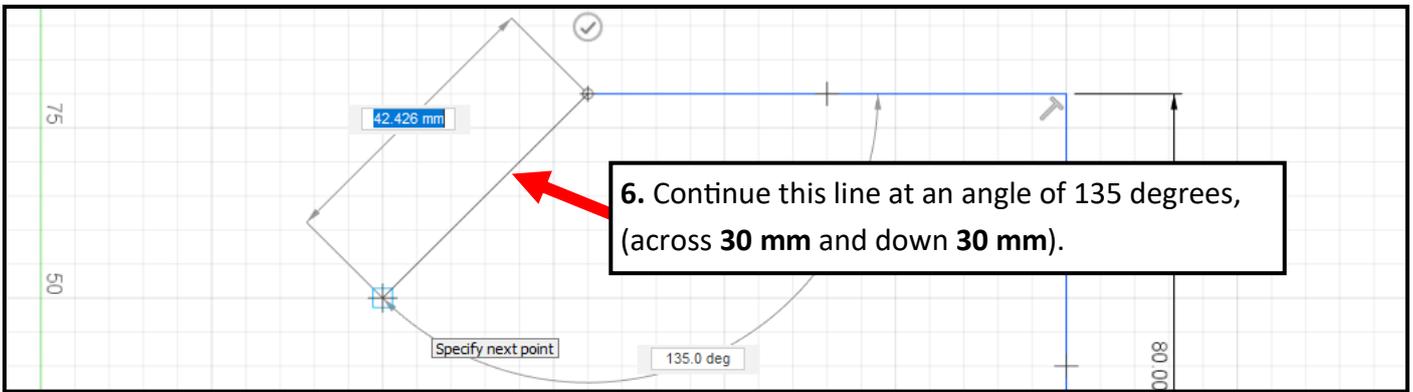


4. At **90 degrees** from the end of the last line, draw another line and set it to **80 mm**. This is will be the height of the car.



5. From the end of the last line draw a line **70 mm** long as shown.



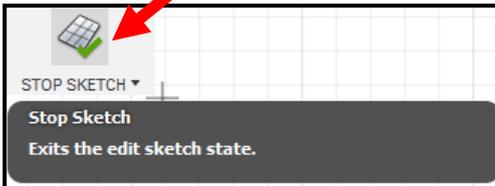
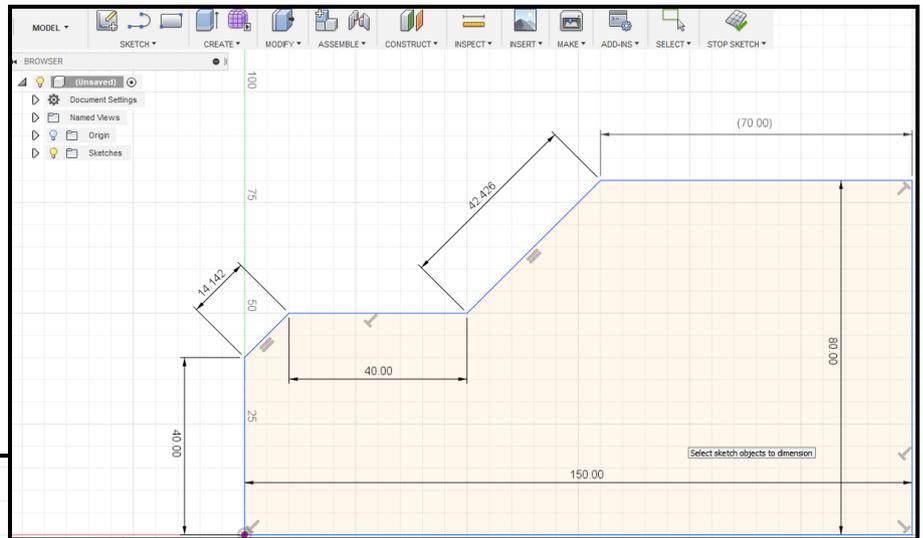


9. Now finish the line by continuing it downwards to the starting point. This should now create a closed face, finishing the body outline.



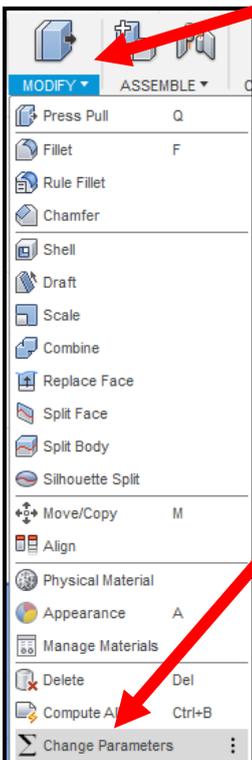
10. Go to the sketch tab and select sketch dimension, alternatively press 'd' on the keyboard. Select each line of the body to add a dimension. This constrains each line and fixes it in place.

11. Now the sketch is finished click on 'Stop Sketch'.

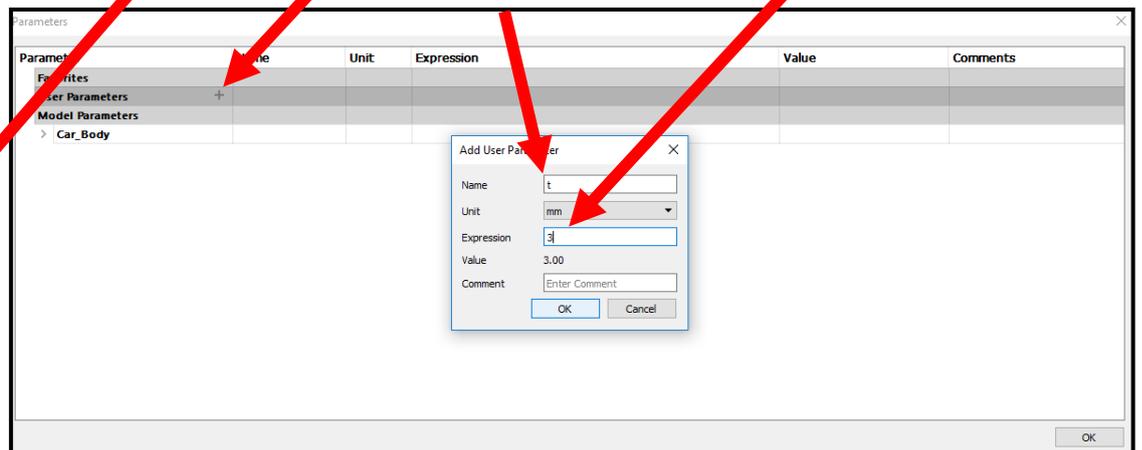


12. Parameters can be added to put set rules in place for your design. Since this design uses the same material thickness for each part we can define this in the parameters section.

To do this go to 'Modify' then at the bottom select 'Change Parameters'.



13. Add a new user parameter by selecting the '+' sign. Name this parameter 't', make sure the unit is set to 'mm' and make the expression value '3'. This represents the thickness of the material used. If the material you use is thicker or thinner you can adjust the thickness of all parts that use this parameter by changing the value.

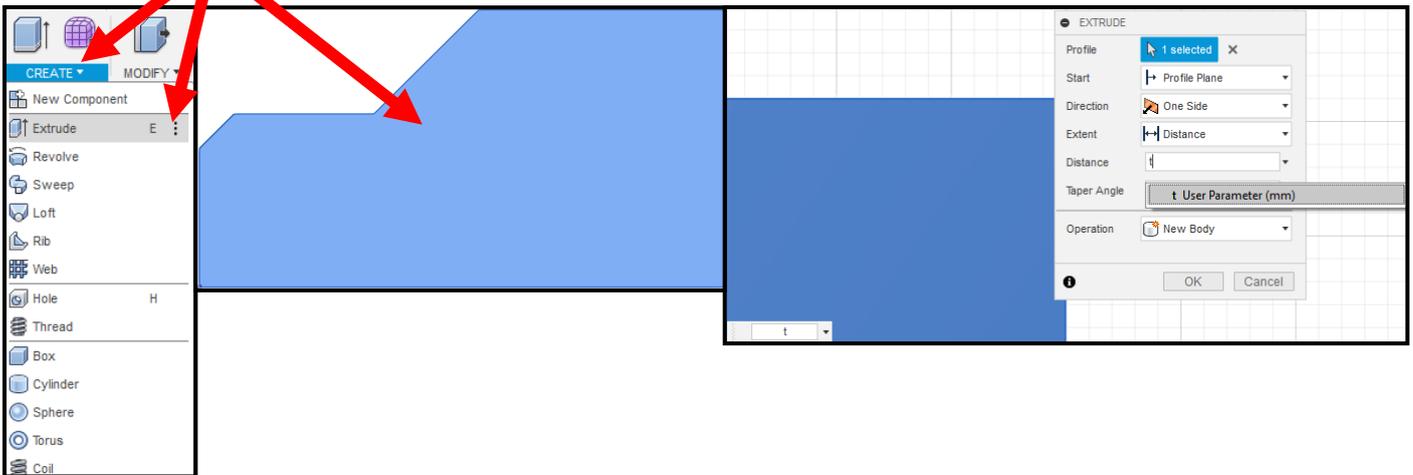


14. As before add another parameter, this will represent the width of the car. Name this parameter 'w' and set it to 70 mm.

Parameter	Name	Unit	Expression	Value
Favorites				
+	User Parameters			
☆	User Parameter	t	3 mm	3.00
☆	User Parameter	w	70 mm	70.00

15. Under the 'Create' tab select 'Extrude' and select the face of the body (alternatively you can select the face and press 'e' on the keyboard to extrude).

16. If you type 't' into the Distance box it will use the parameter that you created for the length of the extrude.

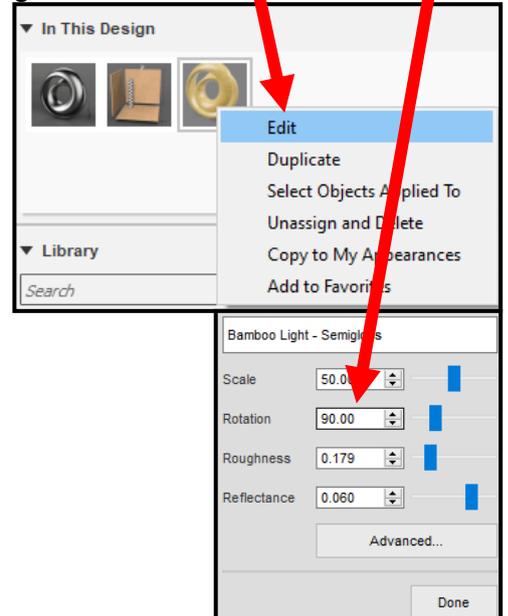
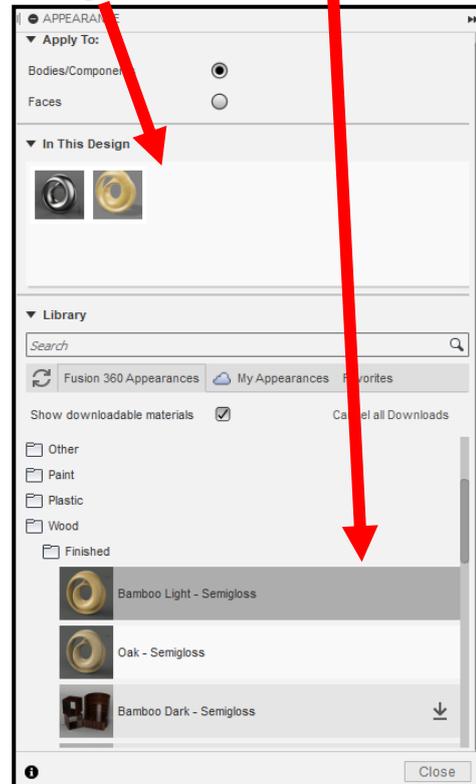
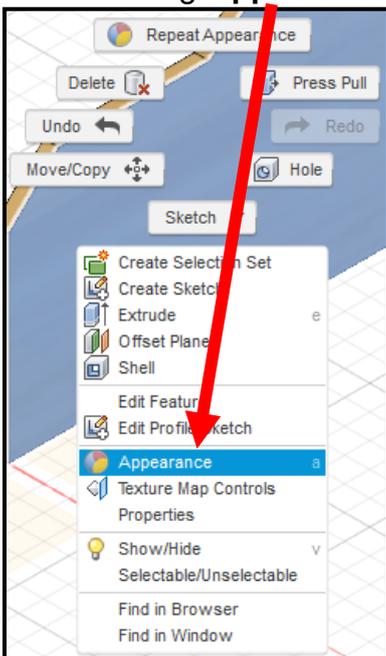


17. To give the object a more realistic look you can change the appearance by right clicking on the object and selecting 'appearance'.

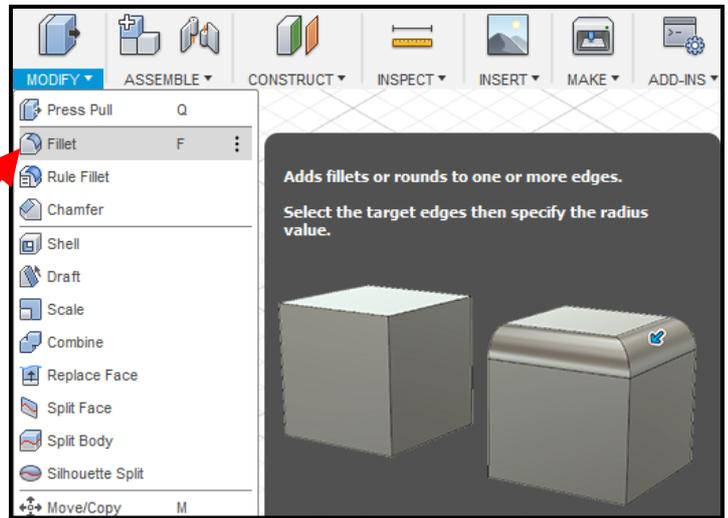
18. Under the library tab there are folders containing different materials. Select the wood folder, then select the finished folder and drag the 'Bamboo Light - Semigloss' into the 'In This Design' section.

19. Now drag and drop the new material onto the body to give it the wood finish.

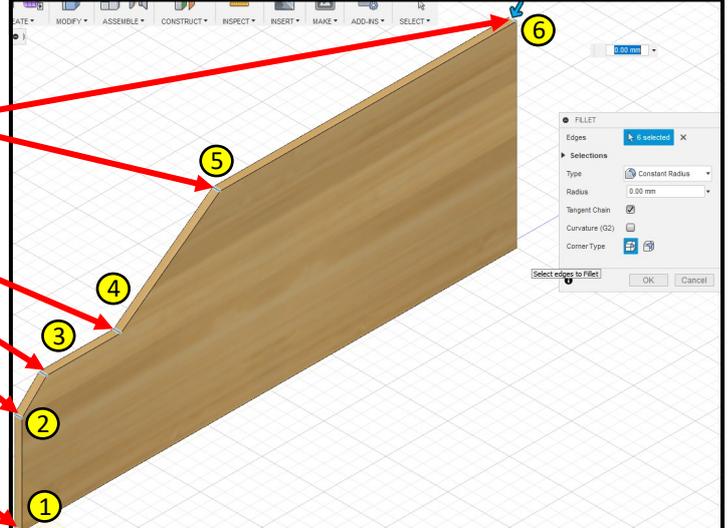
Right click on the new material and select 'edit' and change the rotation to 90 degrees. This will change the direction of the wood grain.



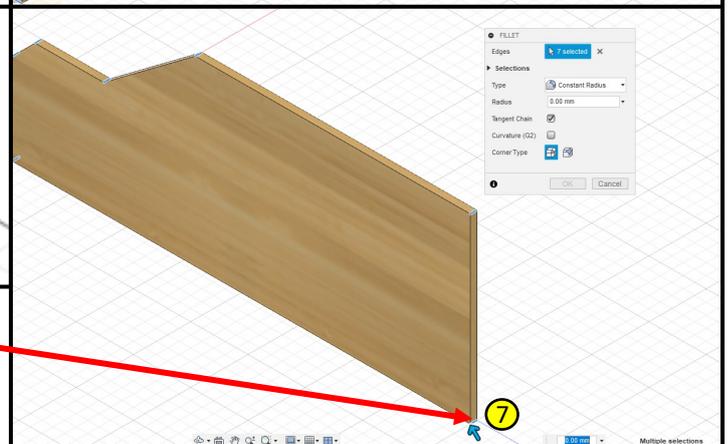
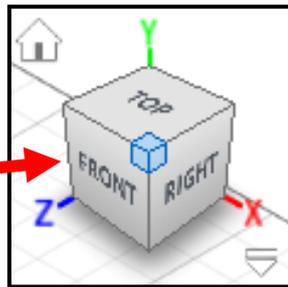
20. Currently the edges of the body are sharp, so to round them the fillet operation needs to be used. This can be done by selecting 'Fillet' under the 'Modify' tab.



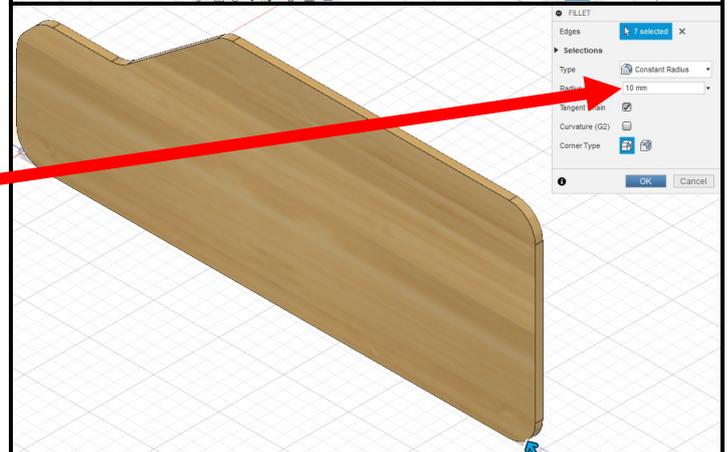
21. Use the scroll wheel to zoom in and select the desired edges. Hold down **shift** (⇧) on the keyboard and select each edge. There are seven in total.



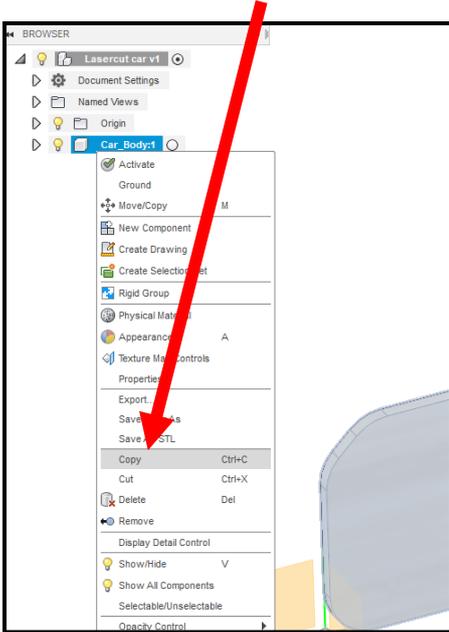
22. Use the box to change the view so that the last edge can be selected.



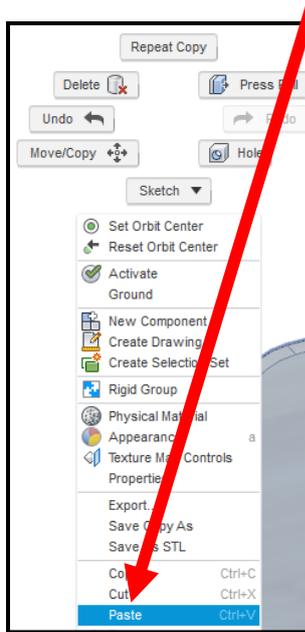
23. Now that all the desired edges have been selected, a suitable radius value can be chosen. For this activity the radius of the fillet should be set to **10 mm**.



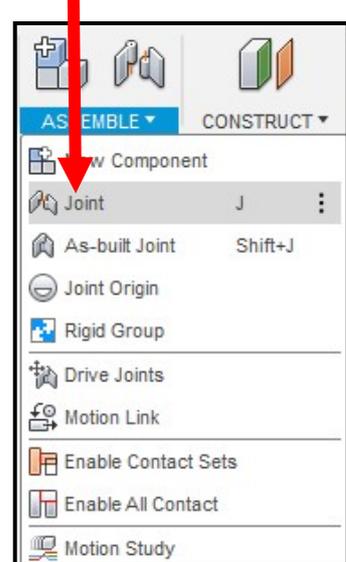
24. Right click on the object from the menu on the left. Select 'Copy'.



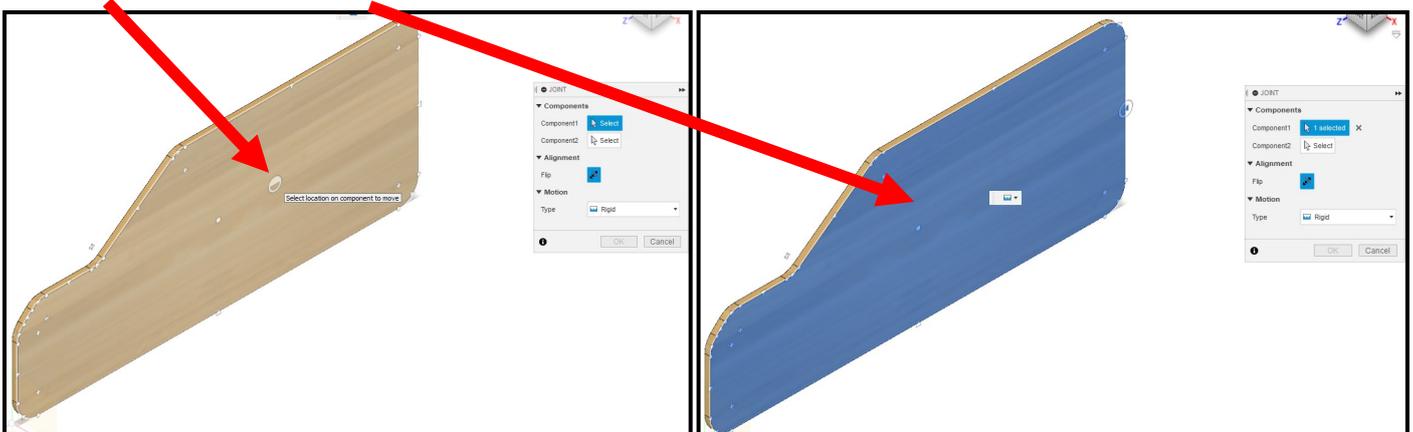
25. Right click anywhere on the blank white space next to the object and select



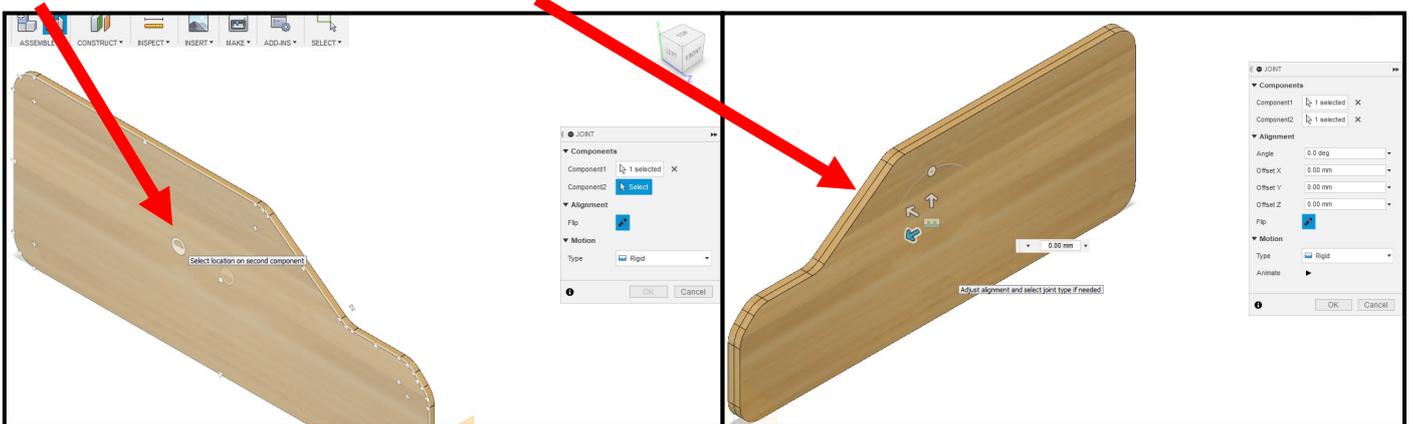
26. Under the 'Assemble' tab select 'joint'.



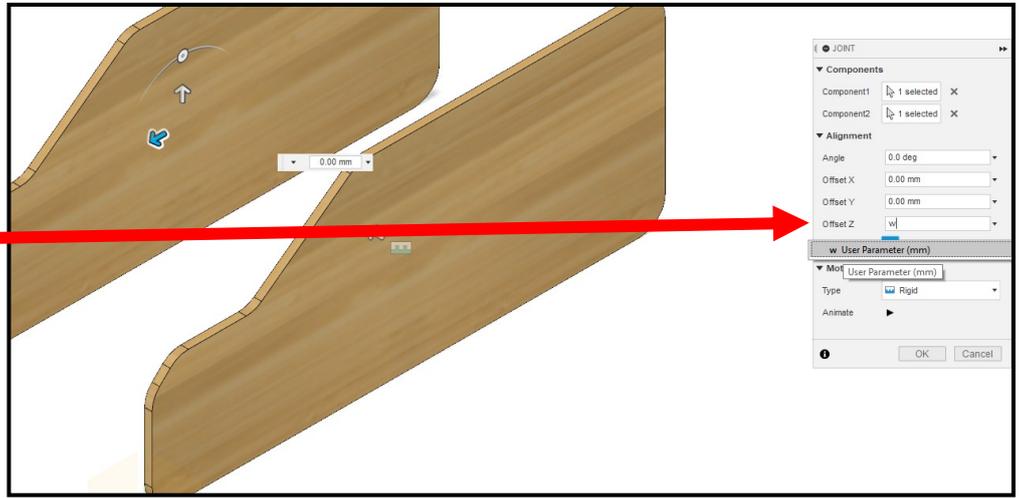
27. Right now there are two objects directly on top of each other. Select the face as shown and click on it until it turns blue. This will select the first object.



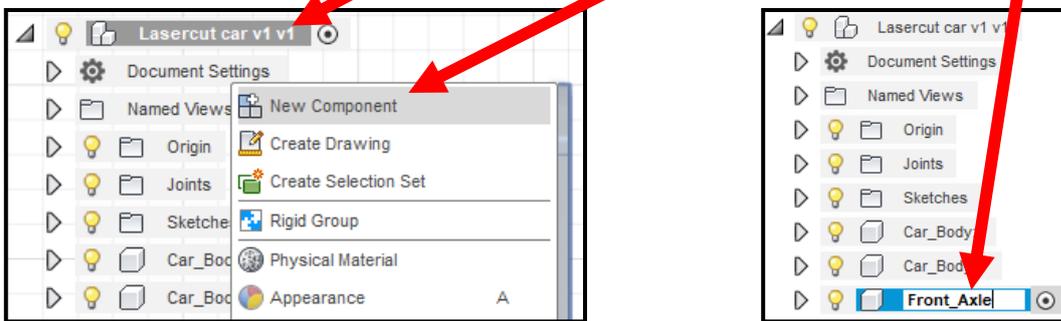
28. Now change the view using the box in the top right corner so the other side of the object is visible. Repeat the previous actions carried out in step 27. Once both objects have been selected they should both appear side by side.



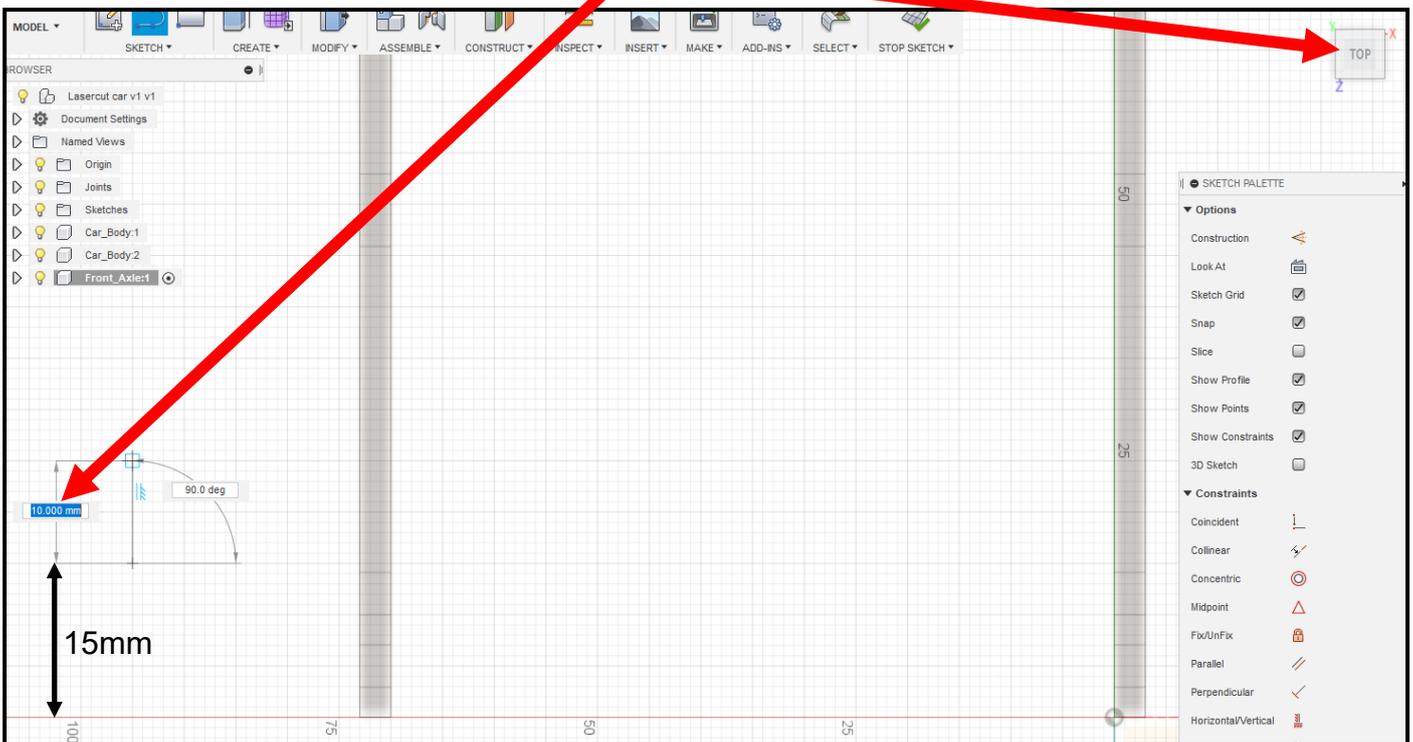
29. In the menu additional options should now appear that allow you to change the alignment of the two parts. Change the 'Offset Z' value to 'w', which is the parameter that was set as the width of the car. This sets the two parts a distance of 70mm apart.



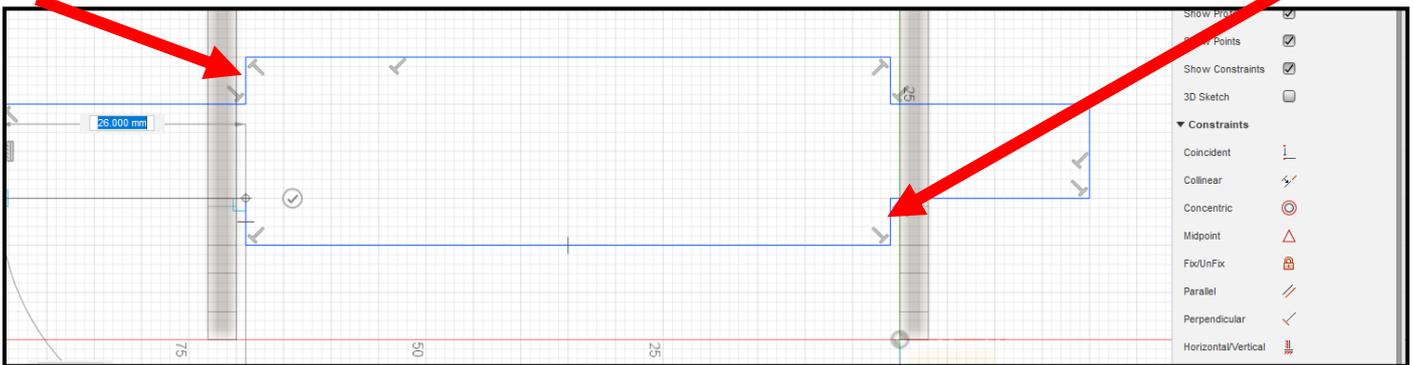
30. Now right click on the document name and select 'new component' and rename it 'Front_Axle'.



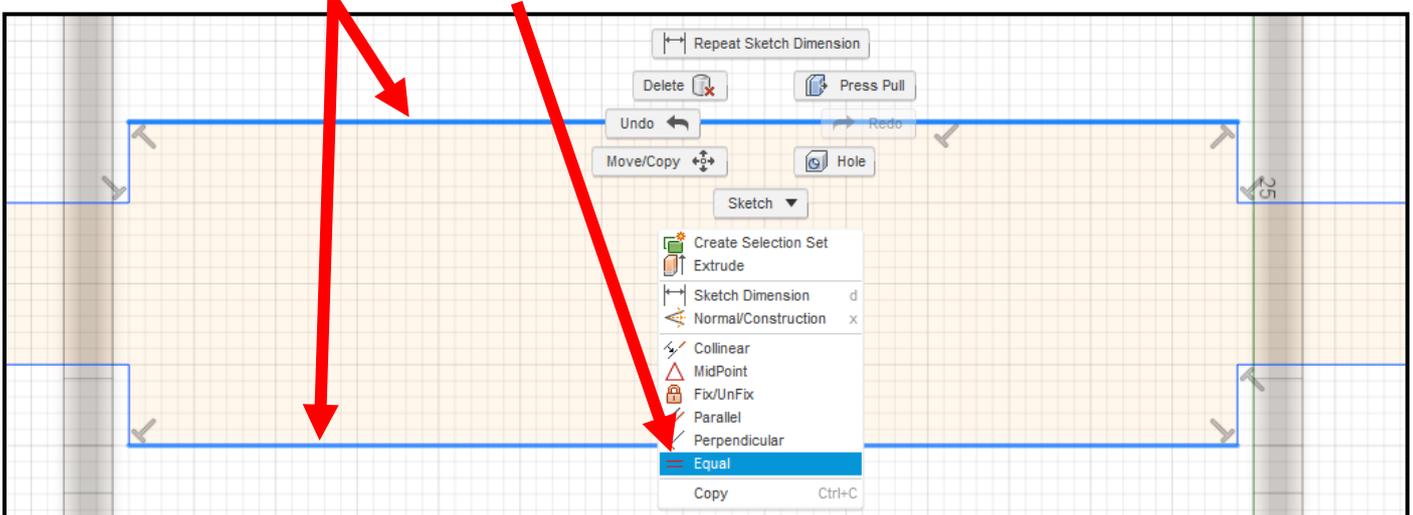
31. Start a sketch for this new component and select the line tool. On the top plane begin a vertical line 10mm long to the left of the bodies. The starting point should be 15mm from the bottom.



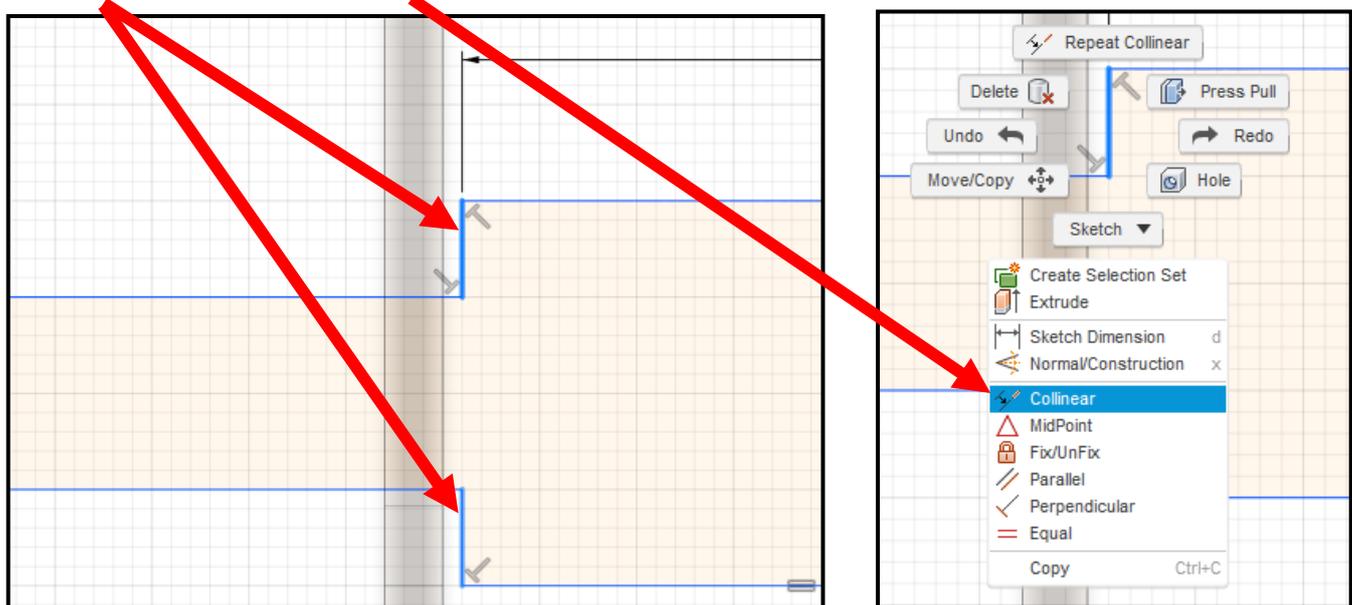
32. Continue using the **line tool** to draw the following shape. It does not need to be exact as dimensions for each line can be added later. When drawing lines the line tool will want to snap onto the edges of the other component in grey, do **NOT** do this. Make sure to **draw the lines next to and not on top** of the component in grey as this can cause complications.



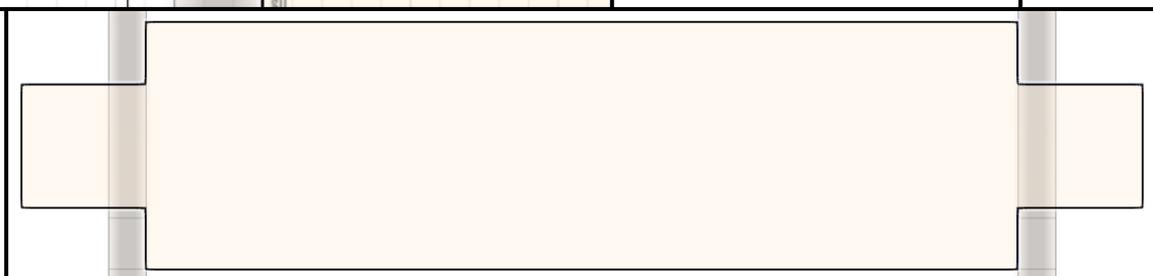
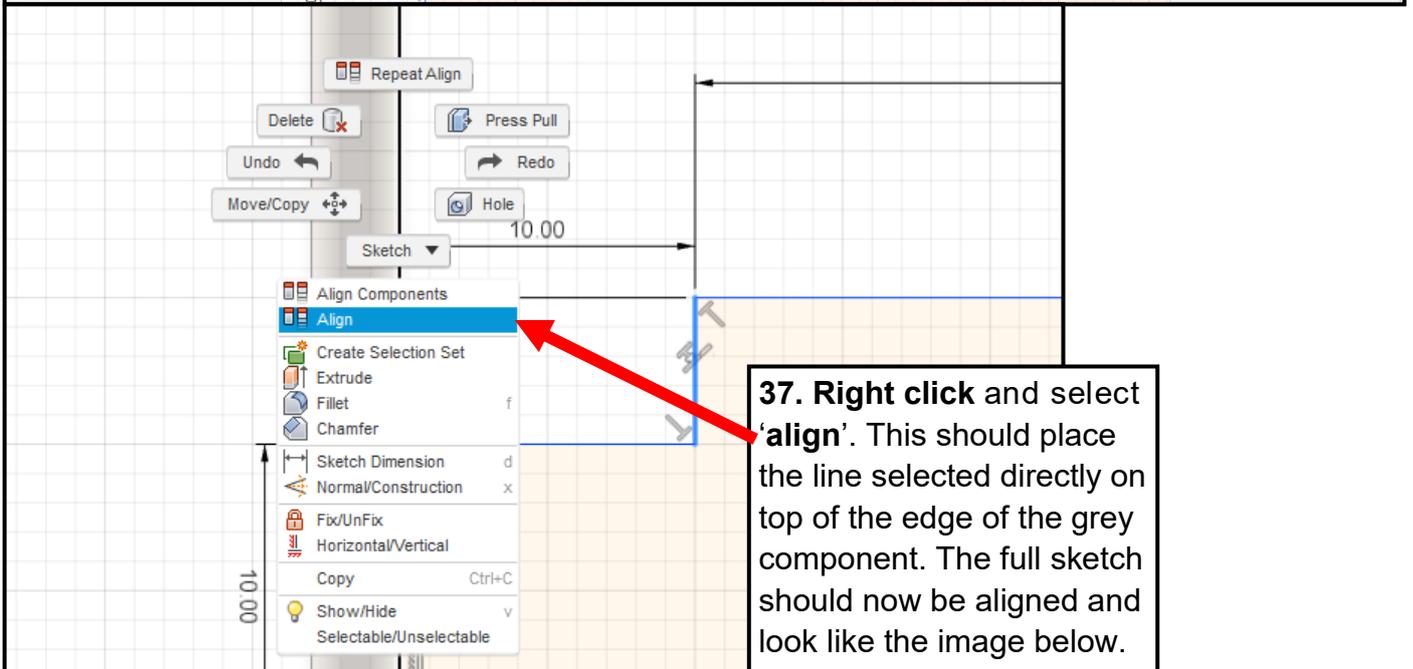
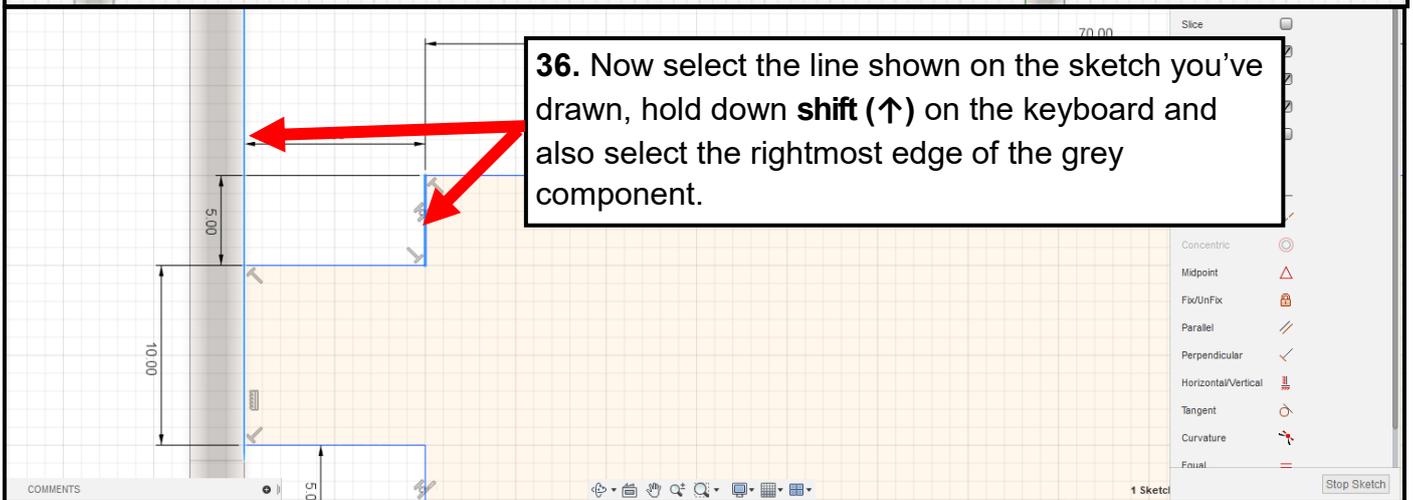
33. Hold down **shift (↑)** on the keyboard and select the two lines shown. **Right click** on one of the lines and select '**Equal**'. This will make both of these lines the same length. Changing the length of one will automatically change the length of the other.



34. Hold down **shift (↑)** on the keyboard and select the two lines shown. **Right click** on one of the lines and select '**Collinear**' so that these two lines are vertically in line with one another.



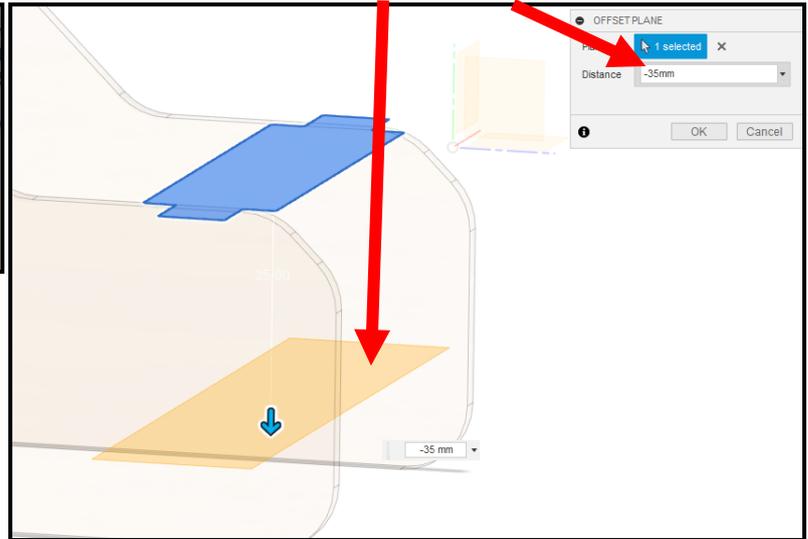
35. Using the **sketch dimension** tool located in the **sketch tab**, add the following dimension labels to the sketch you have created.



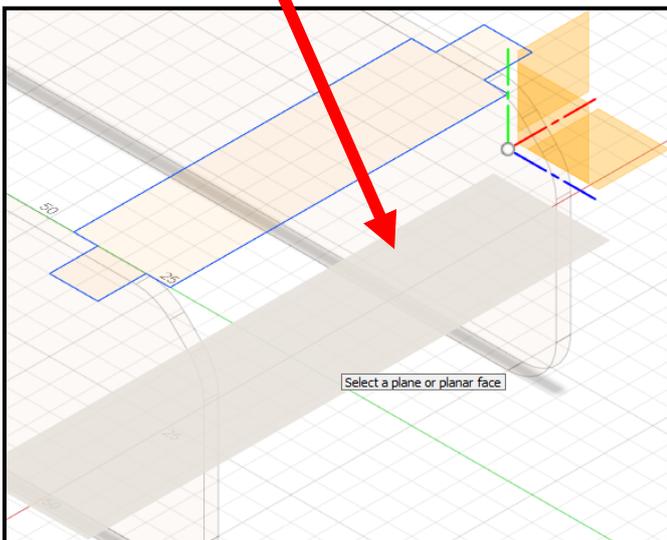
38. After finishing the sketch, go to 'Construct' and select 'Offset Plane'.



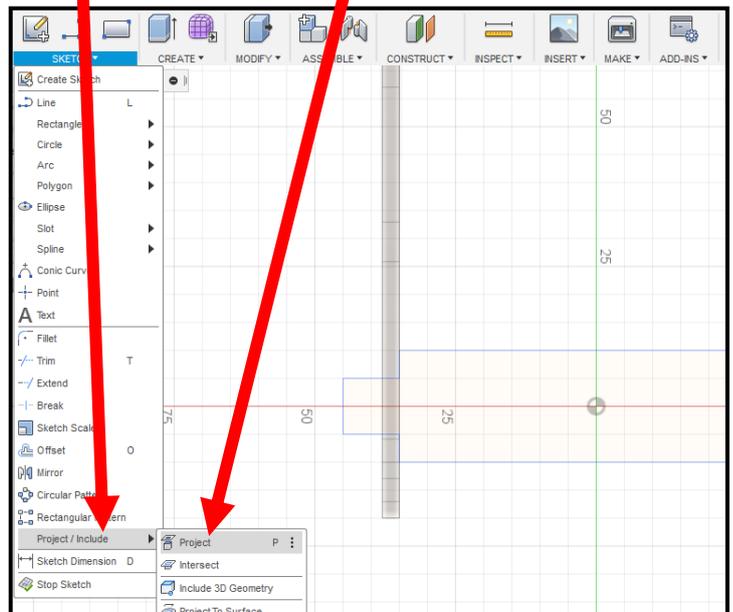
39. Now select the face of the shape in blue and make the offset plane distance as -35mm. This will create a new plane 35mm below the selected face.



40. Start a new sketch and select this plane to sketch on.



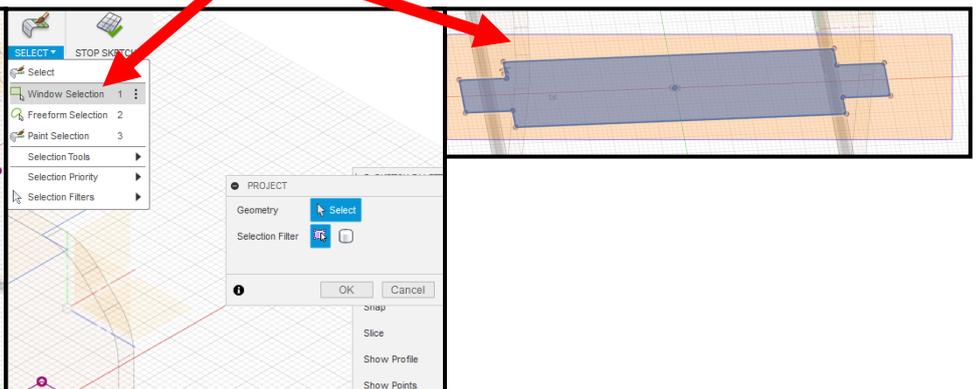
41. Under the sketch tab hover over 'Project/ include' and select 'Project'.



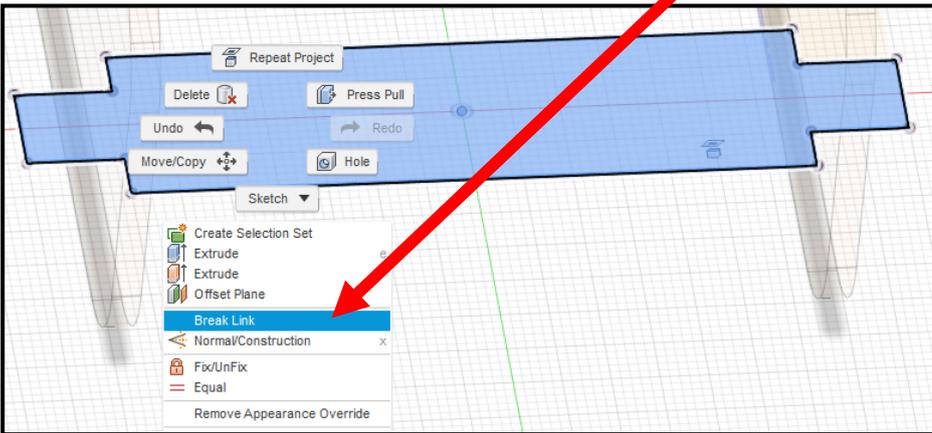
42. Left click on each line of the sketch created earlier. This should make the same shape appear in pink on the new plane as shown below.



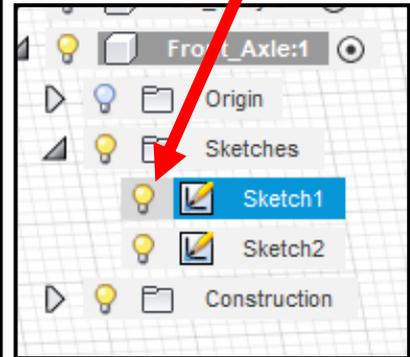
43. Under the 'Select' tab click on 'Window Selection', hold down left click and draw a box around this whole pink shape to select all of it.



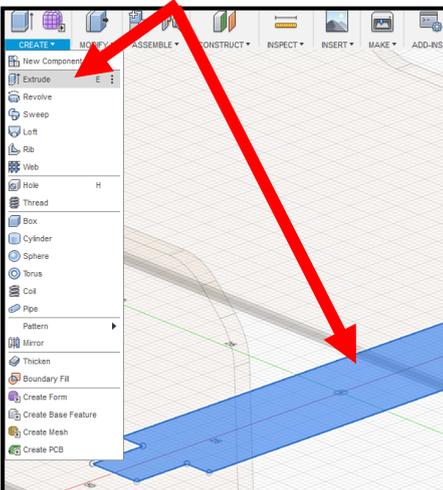
44. Right click on this selected shape and select 'Break Link'. This will mean that this projected shape will now become a separate sketch object.



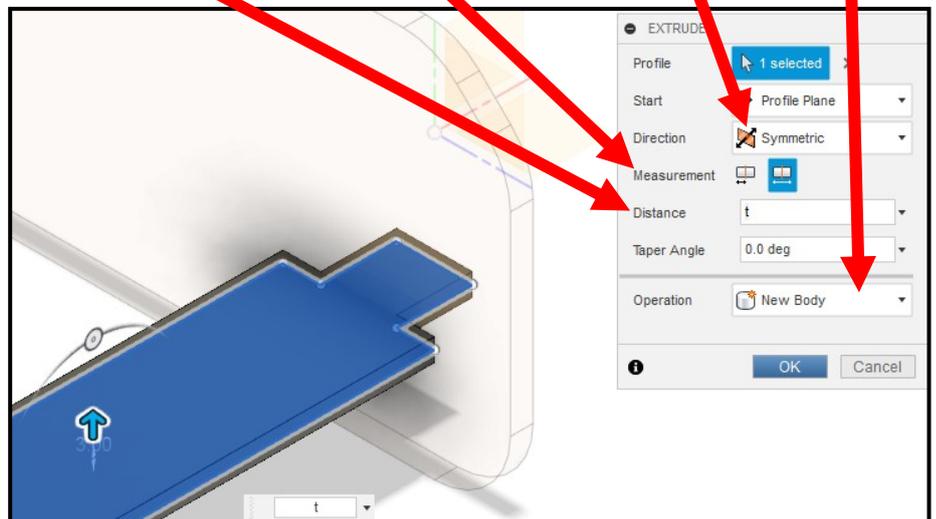
45. Now there should be two sketches located in the menu. Click on the lightbulb next to the first sketch so that it turns grey to hide it.



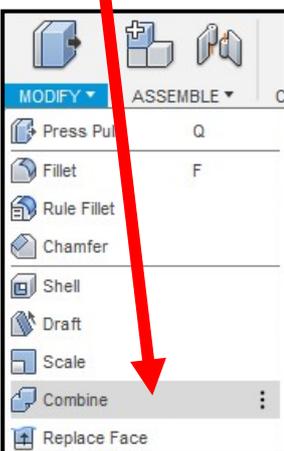
46. Under the create tab select 'Extrude' and select the face of the sketch so that it turns blue.



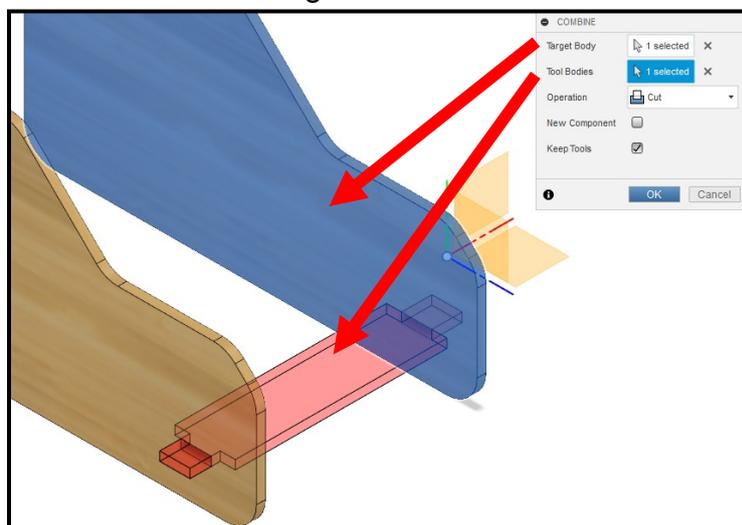
47. In the menu box set the Direction to 'Symmetric', the Measurement to 'Whole Length' and the Distance to the parameter 't'. The Operation should be set as 'New Body'.



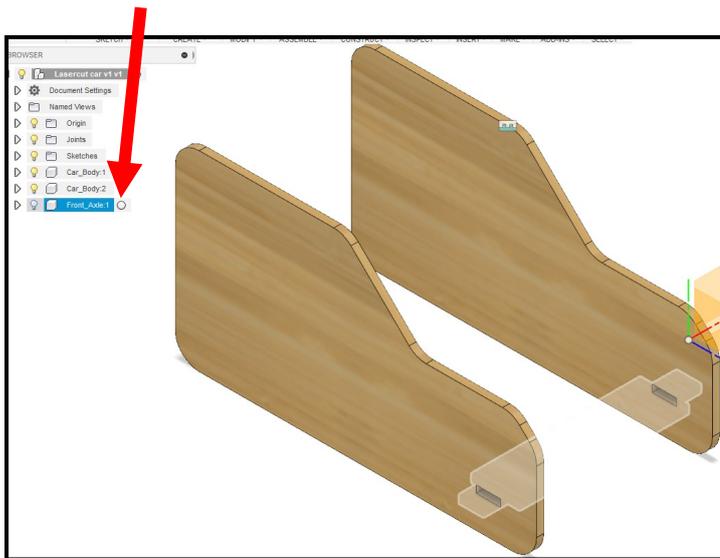
48. Under the modify tab select 'Combine'.



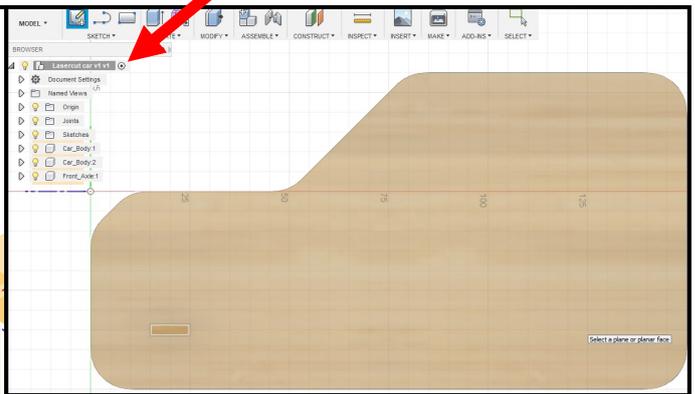
49. Click on the rightmost Car_Body (the original one) to make it the Target Body. Then click on the Front_Axle to make it the 'Tool Body'. Make sure that the 'Keep Tools' box is ticked. Click 'ok', this should then cut both car bodies using the new front axle.



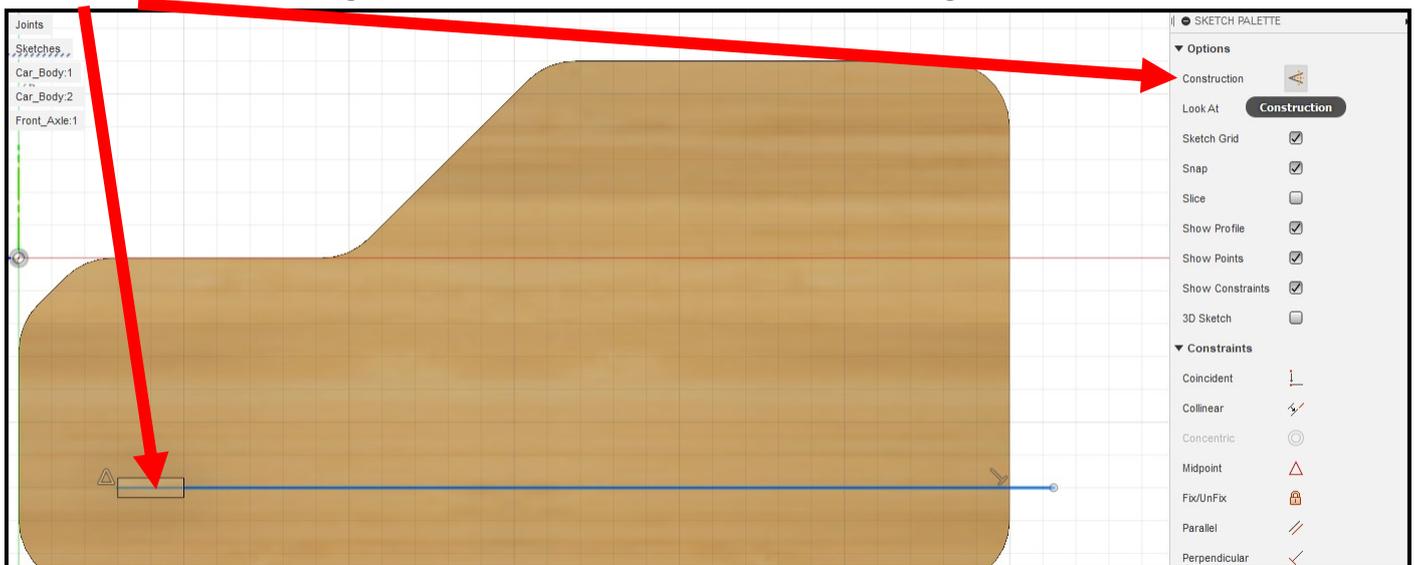
50. Uncheck the circle next to **Front_Axle** in the menu to hide it and confirm that it has cut through both car bodies.



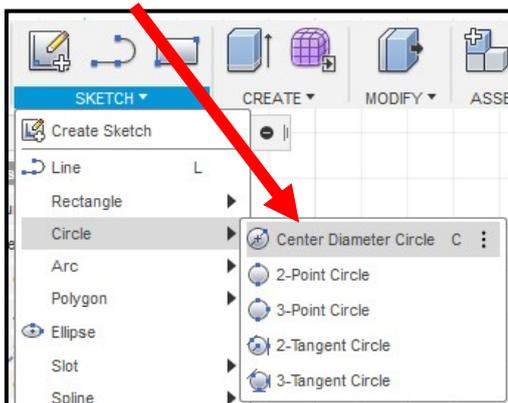
51. Click on the circle next to the name of the project at the top of the menu, and select a **right view** from the box in the top right corner.



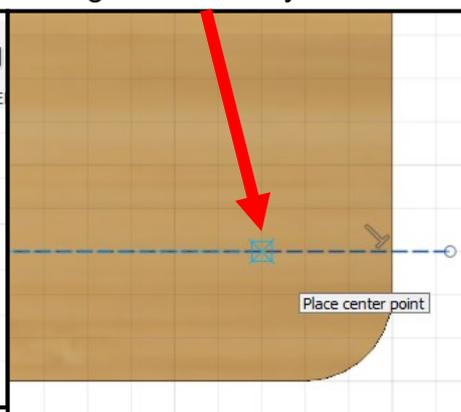
52. Start a new sketch on the face of the car body and draw a **horizontal line starting from the middle of the cut out rectangle** as shown. In the **options menu** to the right, click on **'Construction'** to change this to a construction line, this will change from a solid to a dotted line.



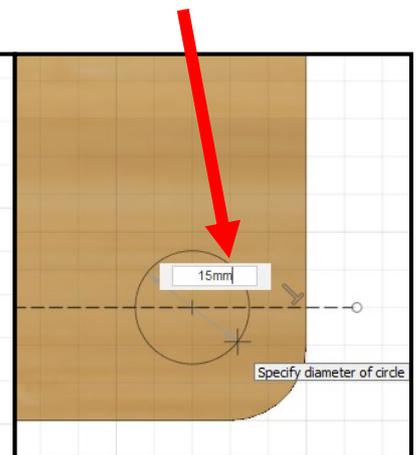
53. Under the sketch tab go to **'Circle'** and select **'Center Diameter Circle'**.



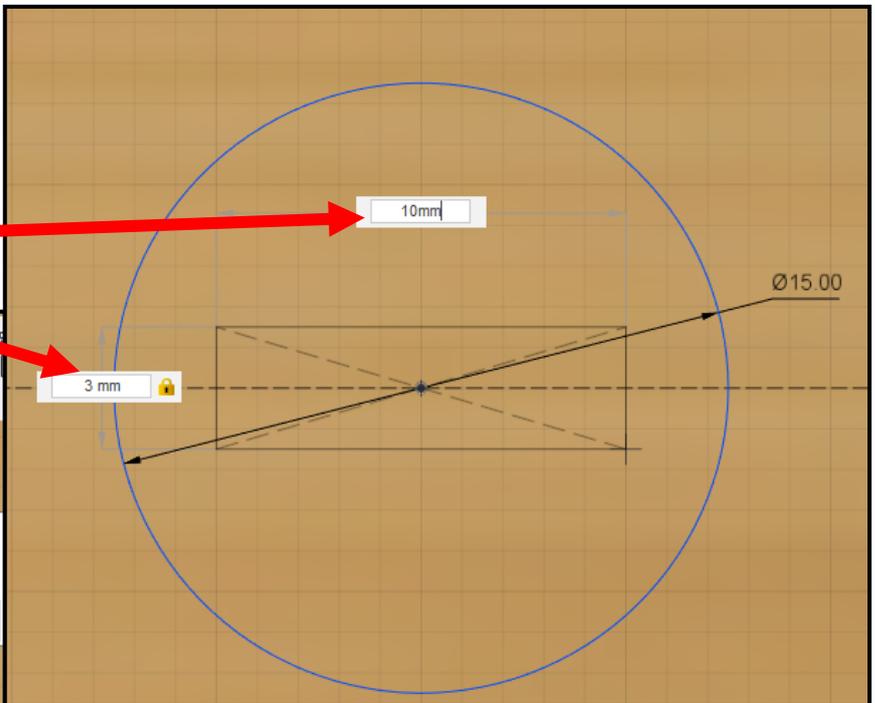
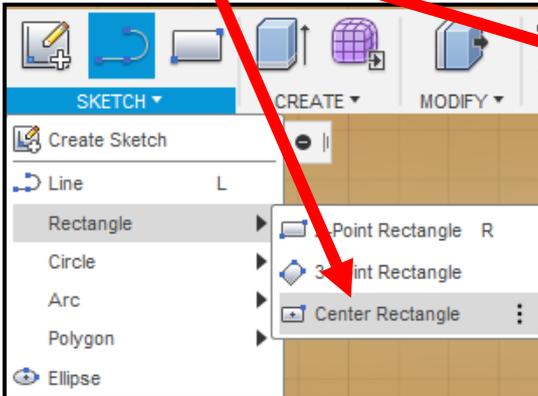
54. Begin the circle on the construction line **15mm** from the right of the body.



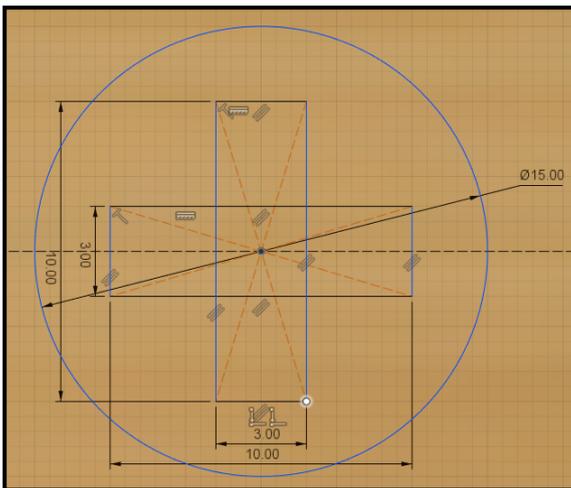
55. Set the **diameter** of the circle as **15mm**.



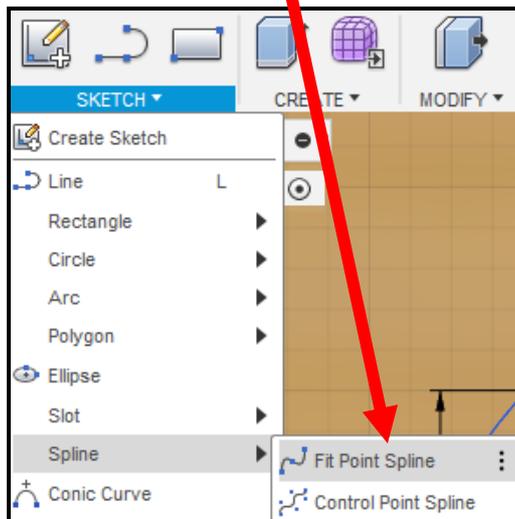
56. Under the **sketch tab** go to 'Rectangle' and select 'Center Rectangle'. Begin this rectangle from the centre of the circle. Set the height to **3mm** and the length to **10mm**.



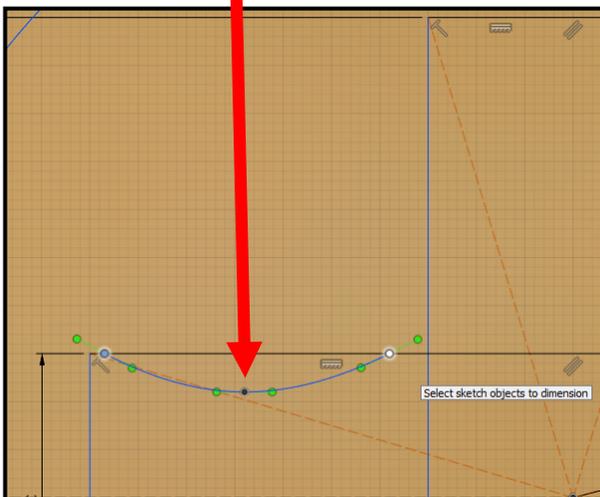
57. Draw the same rectangle from the centre, but this time vertically. This will create a cross as shown below.



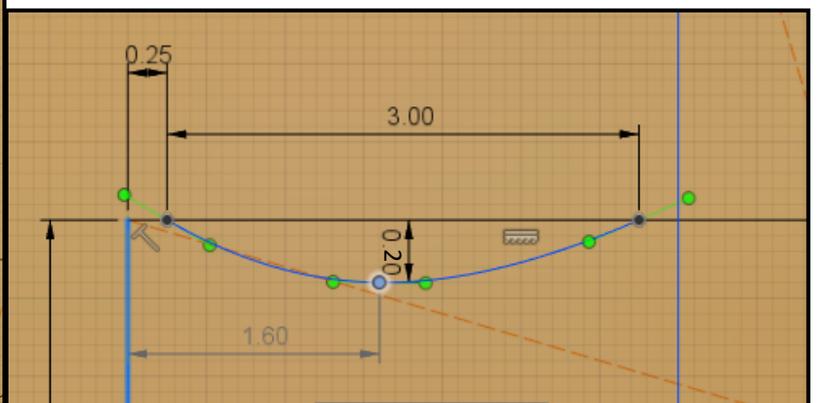
58. Under the **sketch tab** go to 'Spline' and select 'Fit Point Spline'.



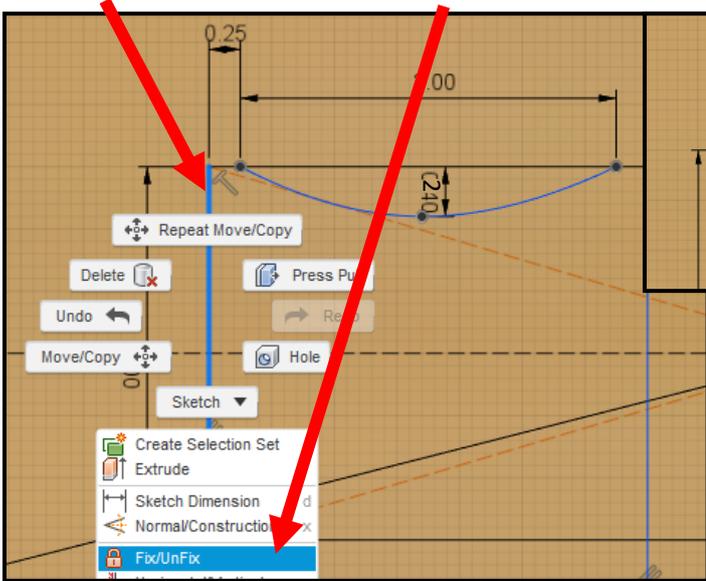
59. Begin drawing the 3 point spline on the line shown below.



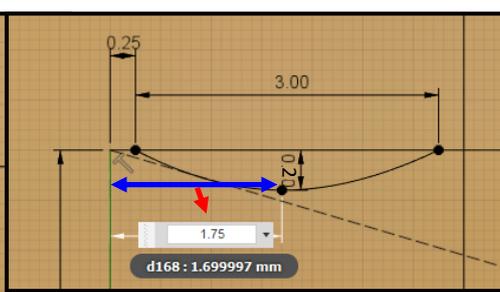
60. Add the **dimensions** (select 'D' on the keyboard) shown below for the spline. Make the first point **0.25mm** from the leftmost edge, the height of the middle point **0.2mm** and the distance between the first and third points **3mm**.



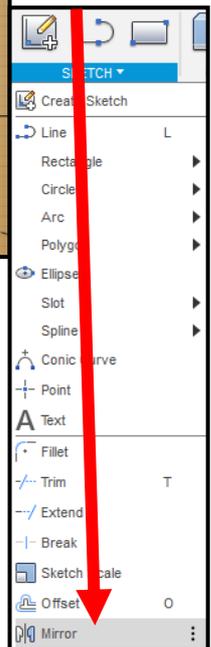
61. Right click on the line to the left shown below and select 'Fix/



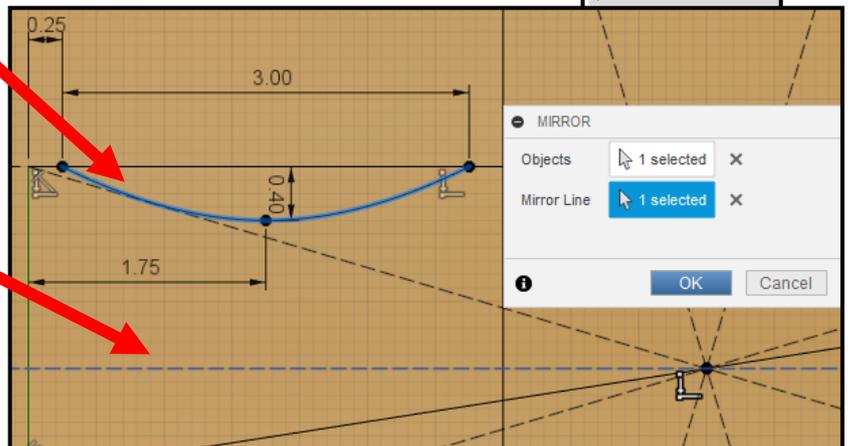
62. Set the distance from this line to the middle point of the spline to 1.75mm.



63. Under the sketch tab select 'mirror'.

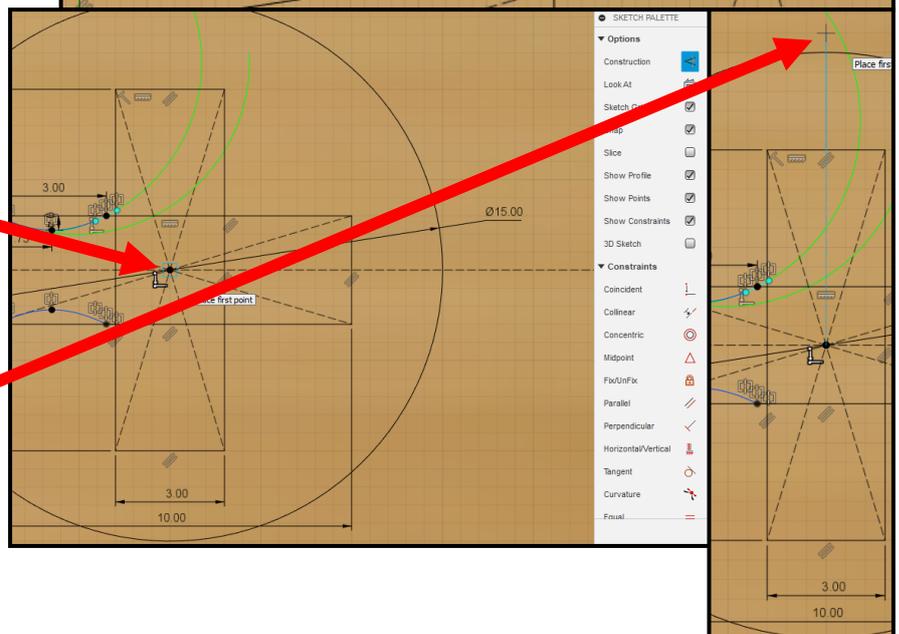


64. Select the spline as the object and the construction line shown as the mirror line.

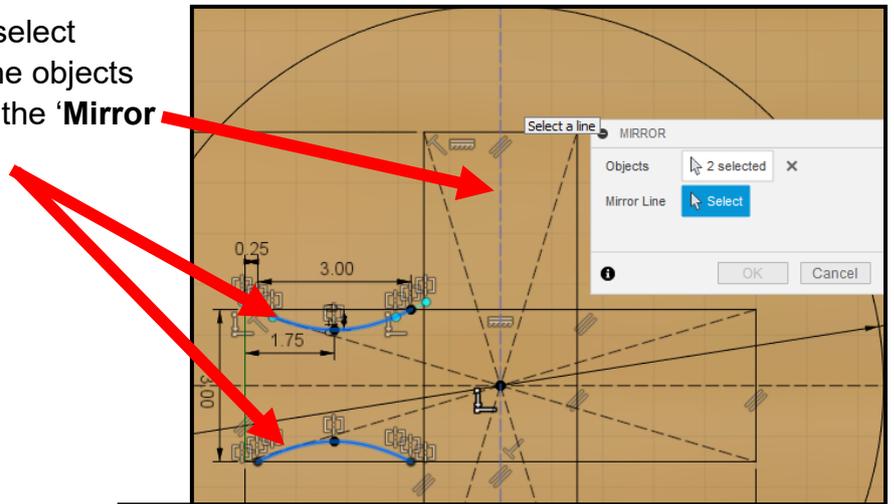


65. Under the sketch tab select the line tool and at the top of the sketch palette make sure 'construction' is selected.

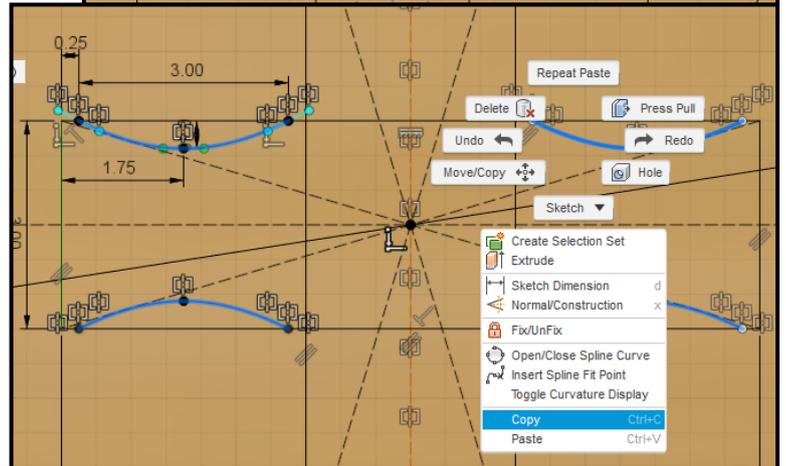
Hover over the centre point and drag the mouse upwards, creating a reference line. Click just above the circle to begin a vertical construction line downwards through the centre and end it just below the bottom of the circle.



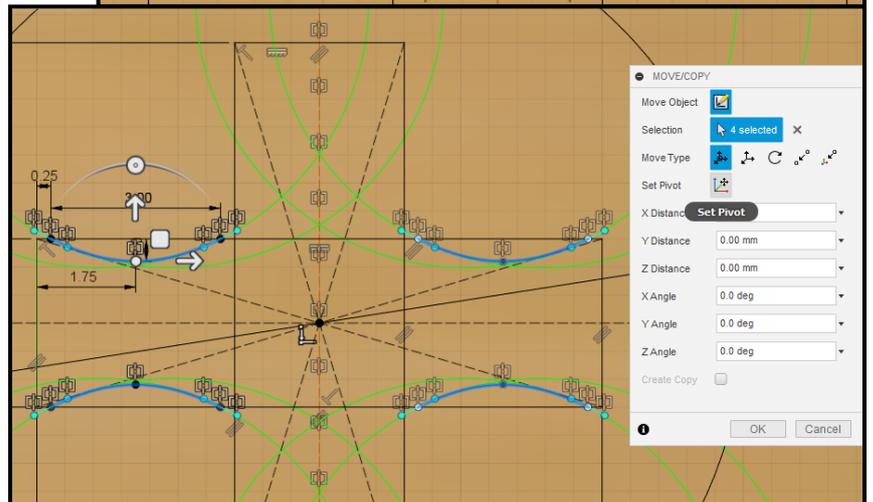
66. As before under the **sketch tab** select **'Mirror'**. Select the two splines as the objects and the vertical construction line as the **'Mirror Line'**. Press **'OK'** to finish.



67. Hold down **shift** on the keyboard and select the **4 splines**. Right click and select **'Copy'**. Then **hold down Ctrl** and press **'V'** on the keyboard to paste.

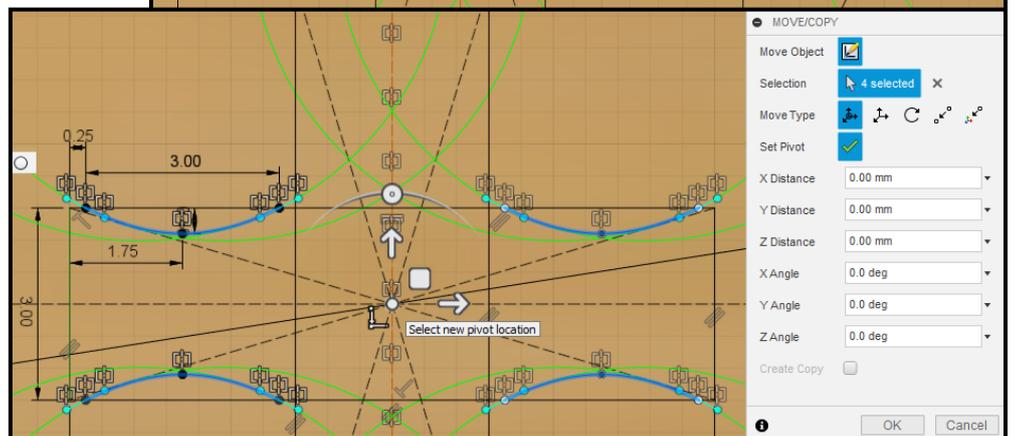


68. The **'Move/Copy'** tab will now be open. Select the symbol next to **'Set Pivot'**.



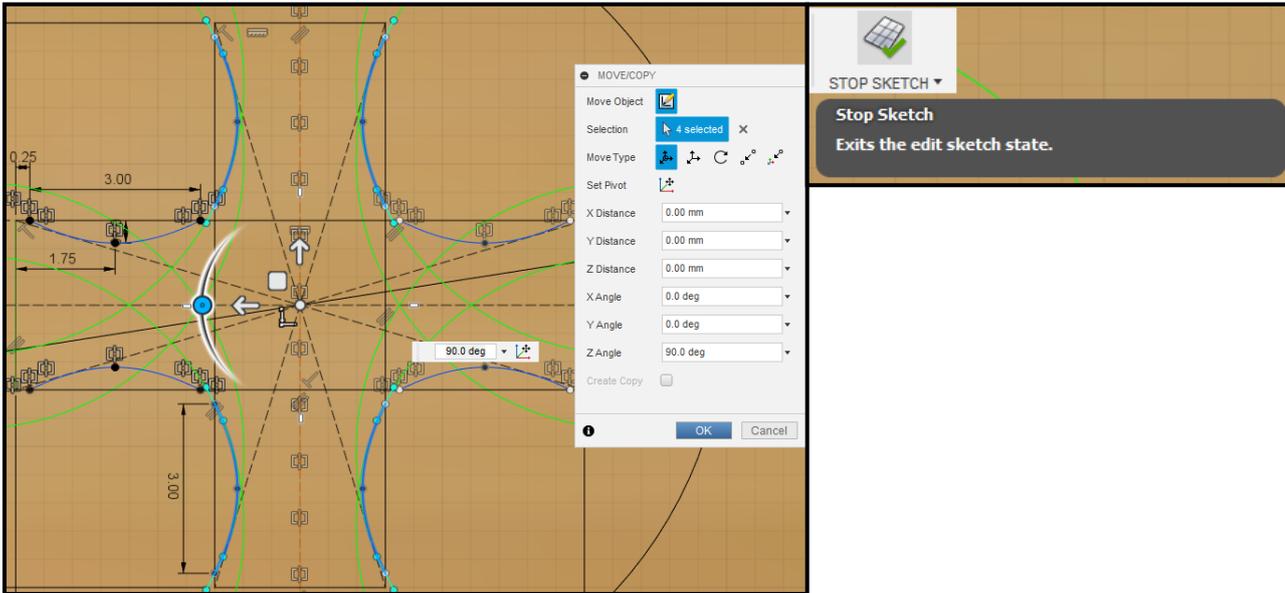
69. Click the **centre** of the pivot and **drag** it so that it **snaps to** the centre of the sketch.

Click on the **green tick** to confirm the position of the pivot.



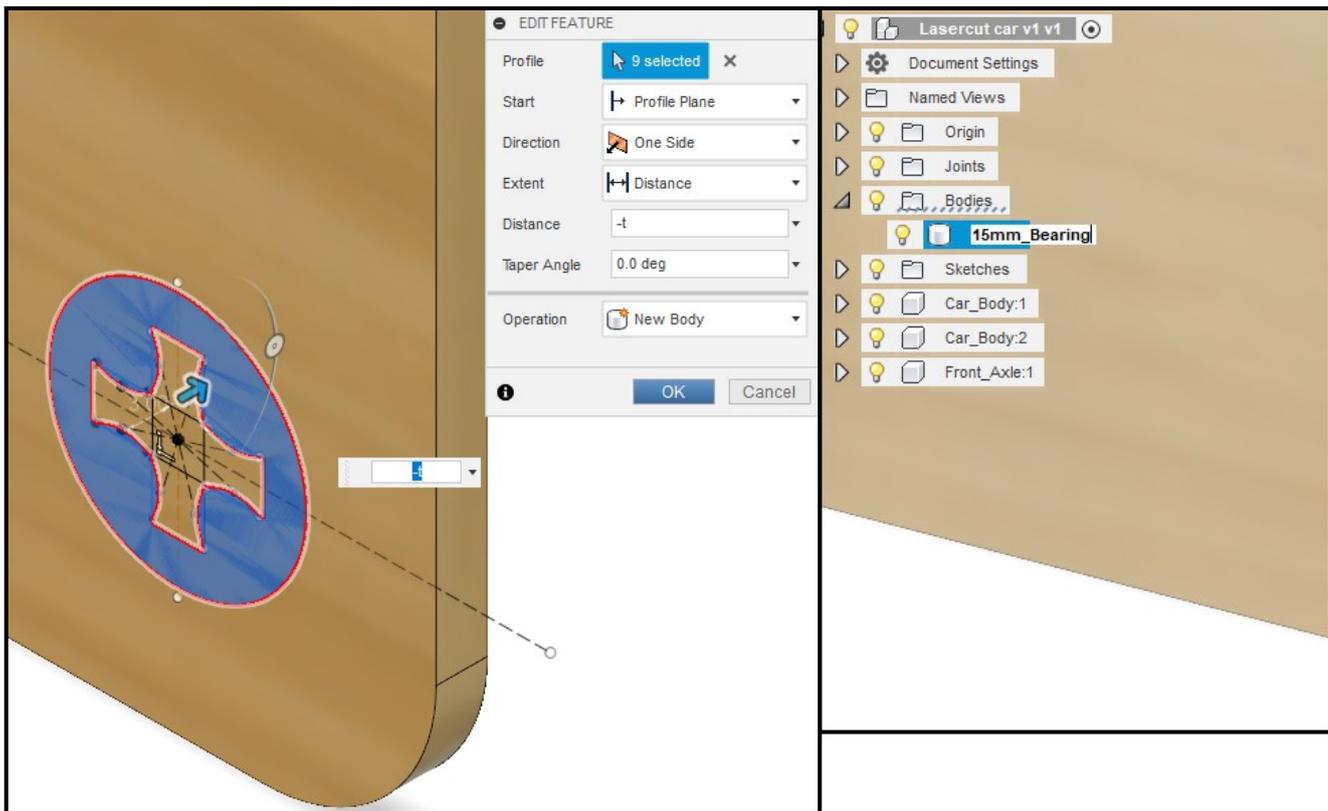
70. Set the 'Z Angle' to 90 degrees. This will rotate the copied splines. To finish click 'OK'.

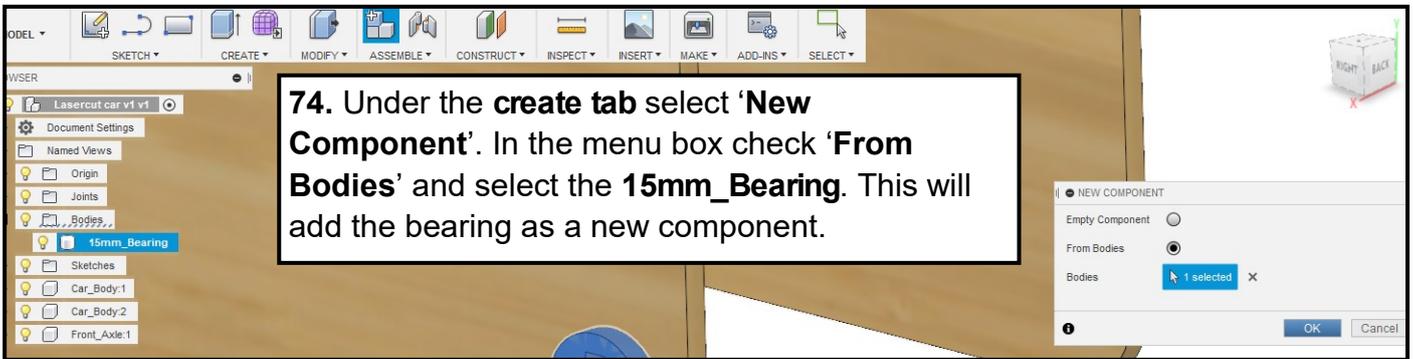
71. To complete the sketch select 'Stop Sketch'.



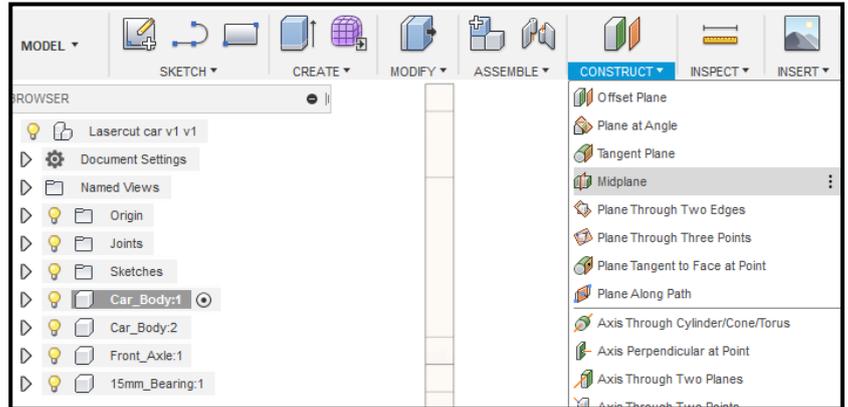
72. Select the face shown in blue below and press 'E' on the keyboard to extrude. The **Direction** should be set to 'One Side', the **Distance** to '-t', and the operation should be 'New Body'.

73. You can find this component under bodies in the menu. Re-name it **15mm_Bearing**.

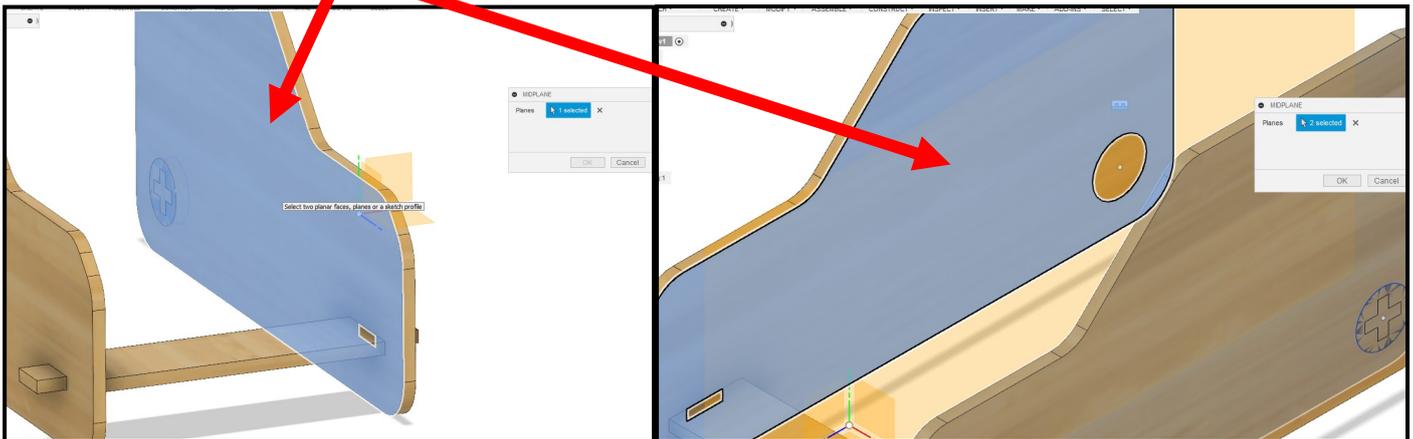




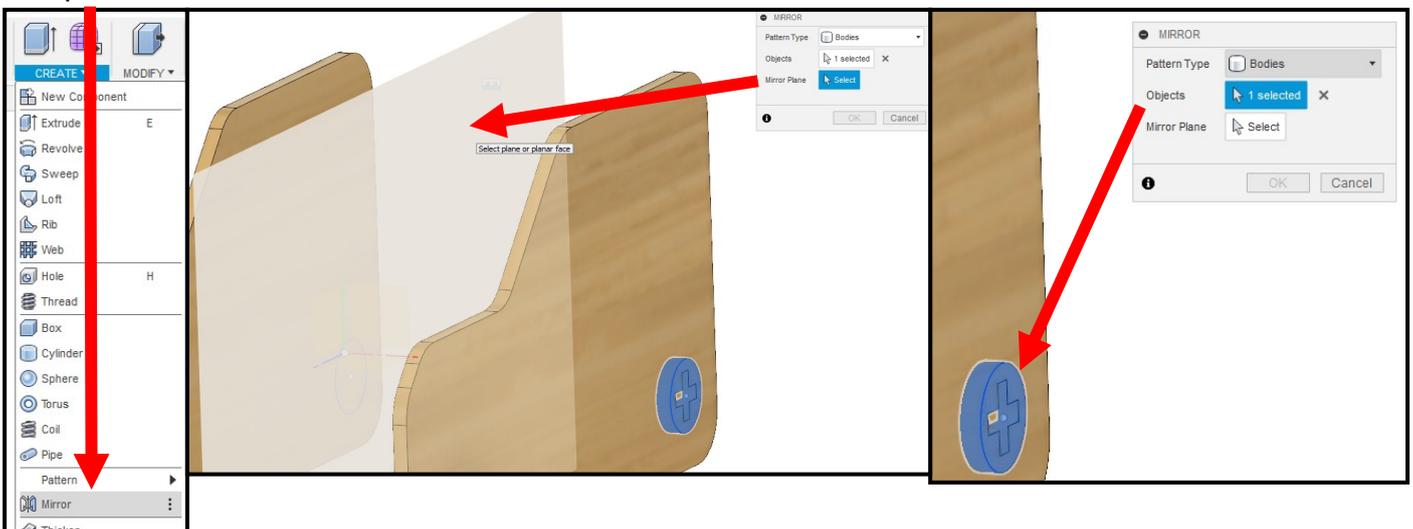
75. Click on the circle next to Car_Body to make it the active component. Now under the **Construct** tab select 'Midplane'.



76. Select the inner faces of each car body to create a plane midway between them as shown below.

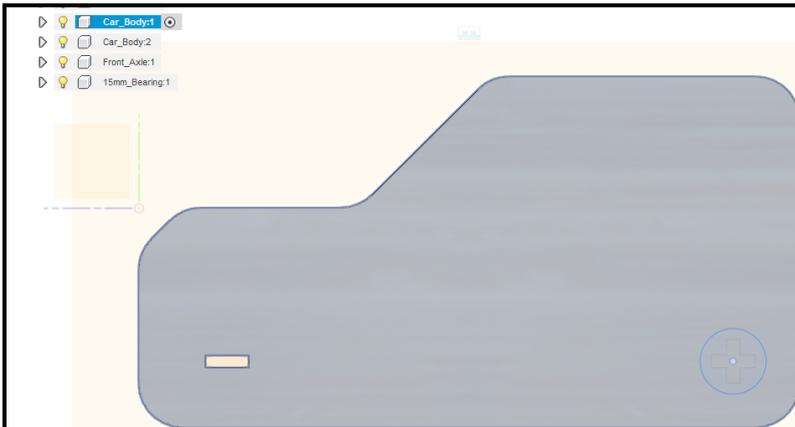


77. Under the create tab select 'Mirror'. Select the bearing as the **Object** and the newly created midplane as the 'Mirror Plane'.

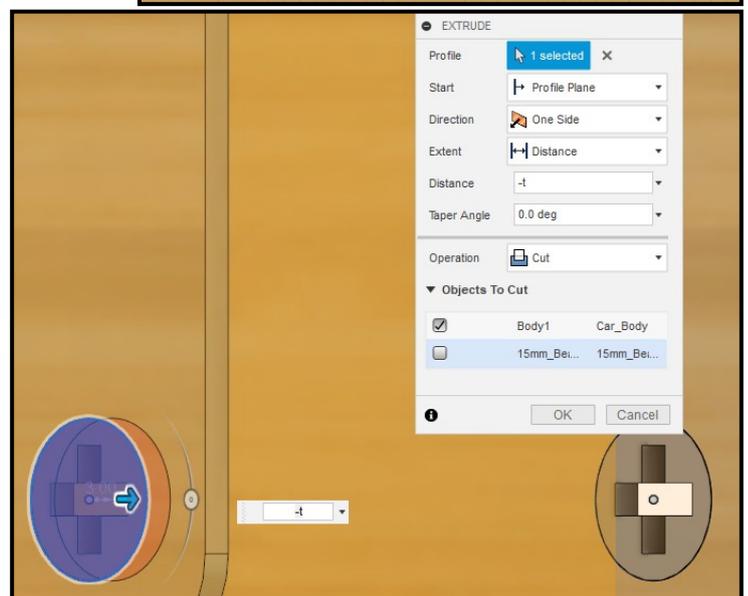


78. Activate the Car_Body component and start a **new sketch** on the side face of the car body.

79. Begin drawing another '**Center Diameter Circle**' starting from the centre point of the bearing and make the diameter the same size of **15mm**.

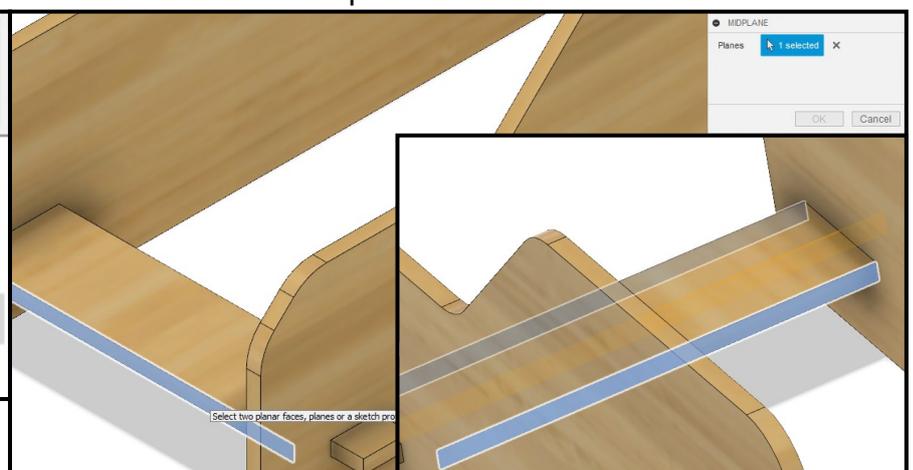


80. Click on this newly drawn circle and type '**E**' on the keyboard to extrude. Make the **distance** '-t' and the **operation** should be '**Cut**'. Under objects to cut **tick** the box for 'Car_Body' and **uncheck** the box for the '15mm_Bearing'. Click '**ok**' to finish.

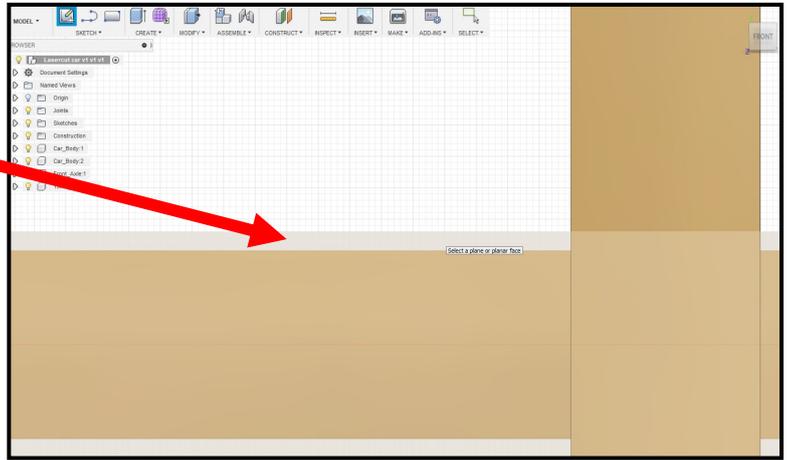


81. Under the '**Construct**' tab select '**Midplane**'.

82. Select the faces shown on the front axle and click '**ok**' on the menu to create the plane in between both faces.

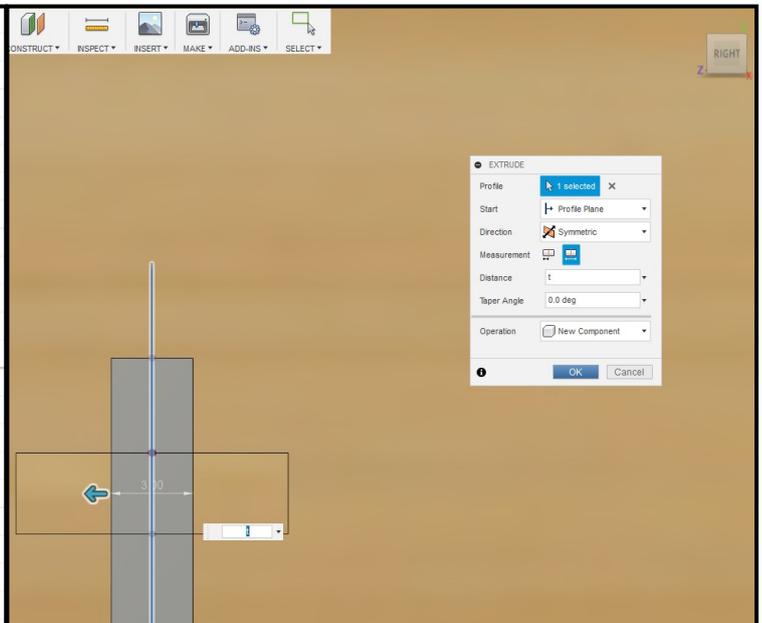
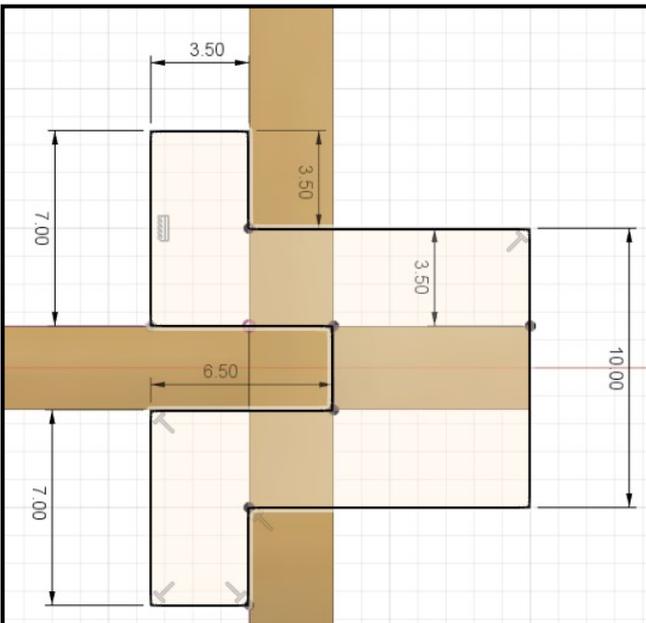


83. Start a **new sketch** from the **Front View**, and select the plane that was just created as the plane you want to sketch on.

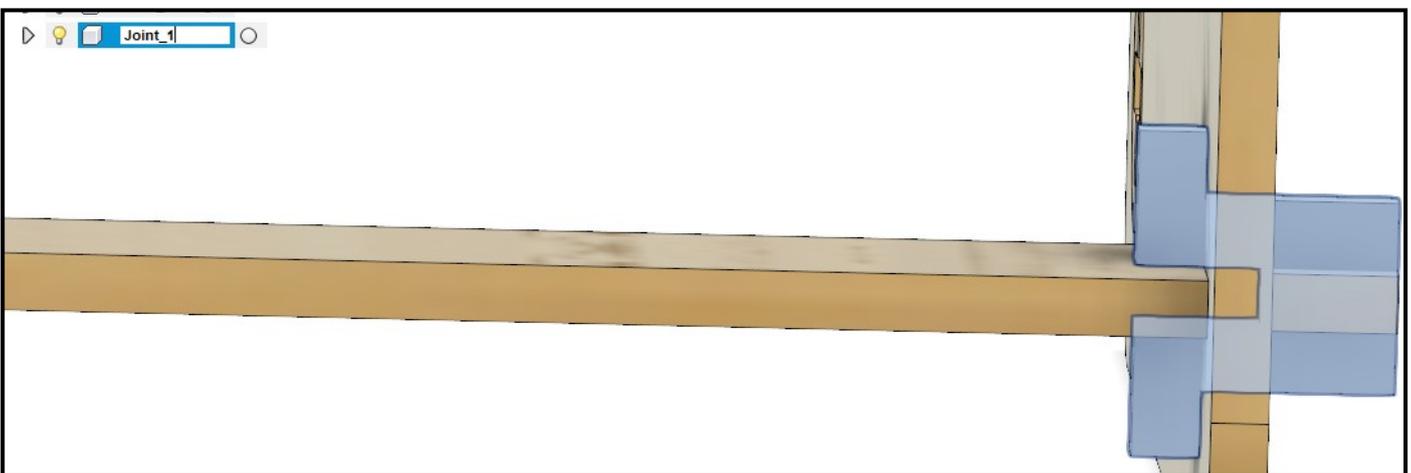


84. Using the **line tool** draw the shape below and add the correct **dimensions** and **finish** the sketch.

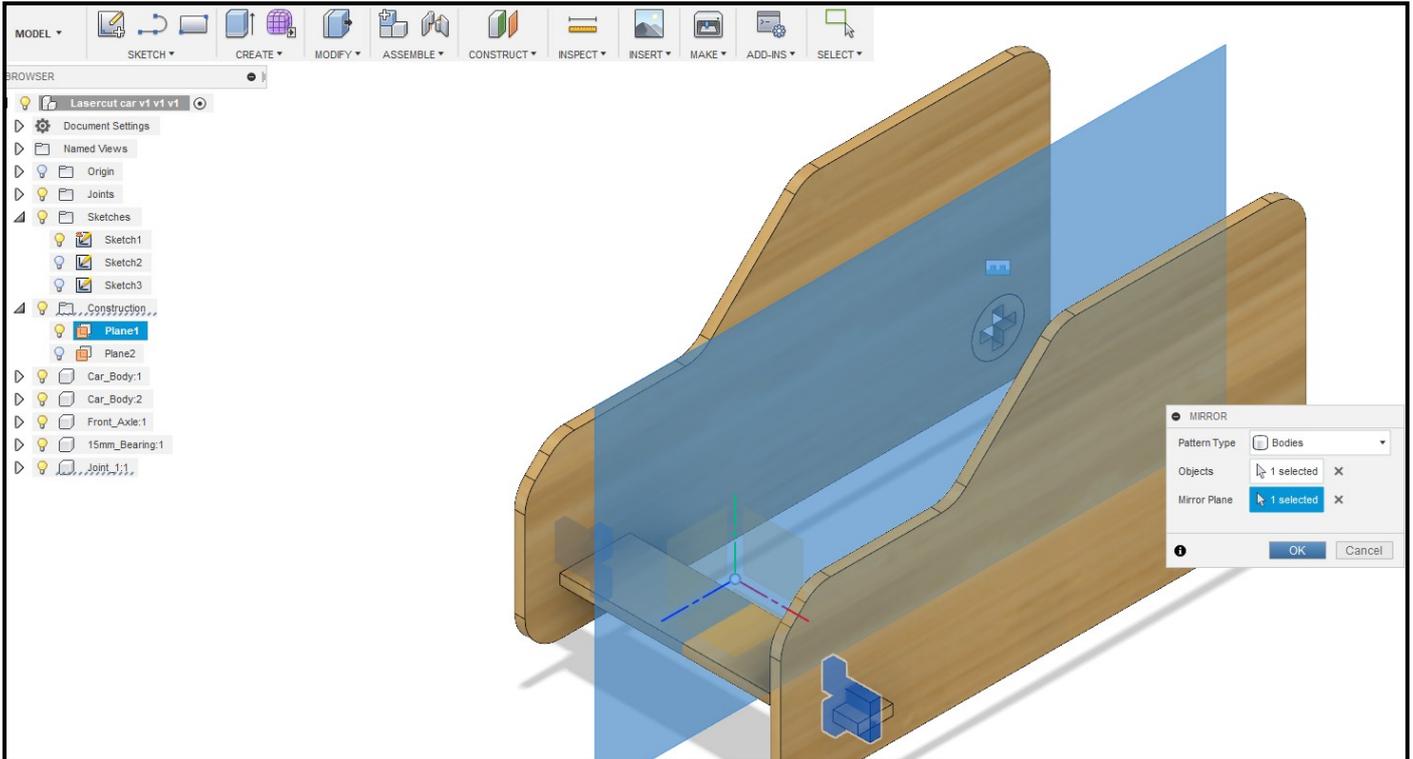
85. Now from a **Right View** begin an **extrude** (type 'E' on the keyboard). Select the sketch that was just drawn as the extrude **Profile**. For the **Direction** select '**Symmetric**', for **Measurement** select '**Whole Length**', make the **Distance** 't' and the **Operation** '**New Component**'.



86. Rename this component '**Joint_1**'.

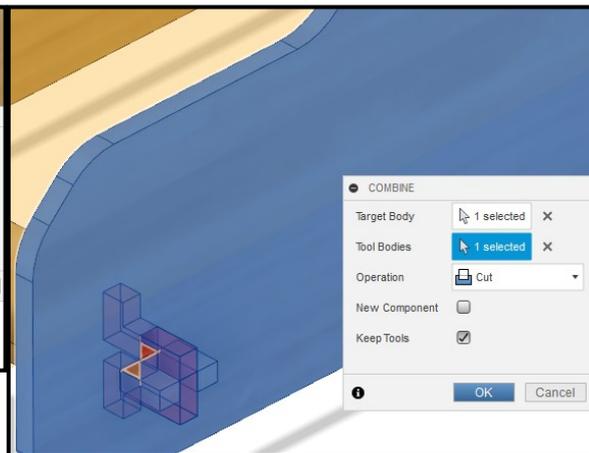
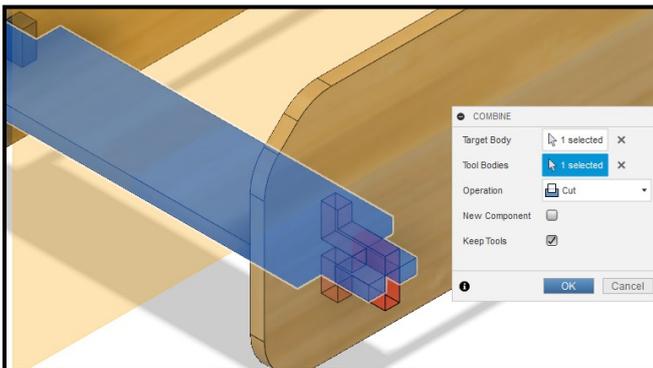


87. Under the **create tab** select '**Mirror**'. Make sure the **pattern type** is set to '**Bodies**' and select the joint part that you just created as the '**Object**', and under the **construction tab** on the left click the **lightbulb** next to **Plane1** to view it, now select this plane as the '**Mirror Plane**'.

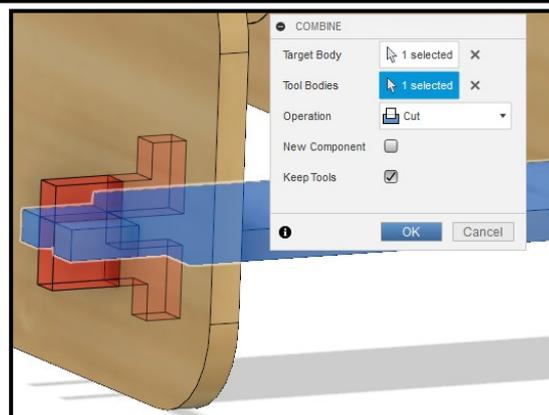


88. Under the **modify tab** select '**Combine**'. Select the **front axle** as the **target body** and the new **joint** as the **tool body**. Make the operation a '**Cut**' and tick the box next to '**Keep Tools**'. Click '**OK**' to finish.

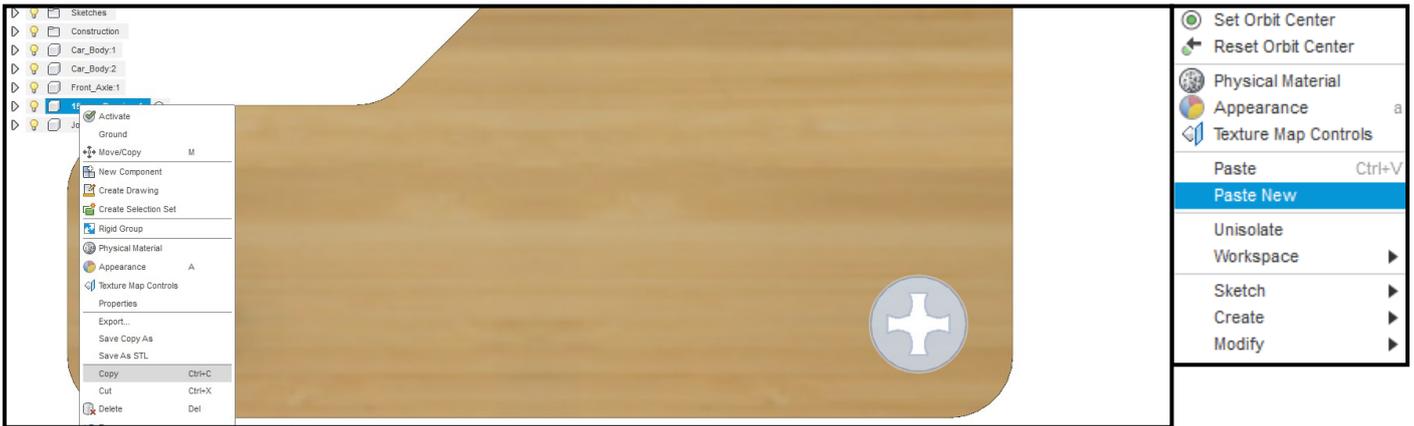
89. Start another combine and this time select the **car body** as the **target body** and the **joint** as the **tool body**. Again make the operation a '**Cut**' and check '**Keep Tools**'.



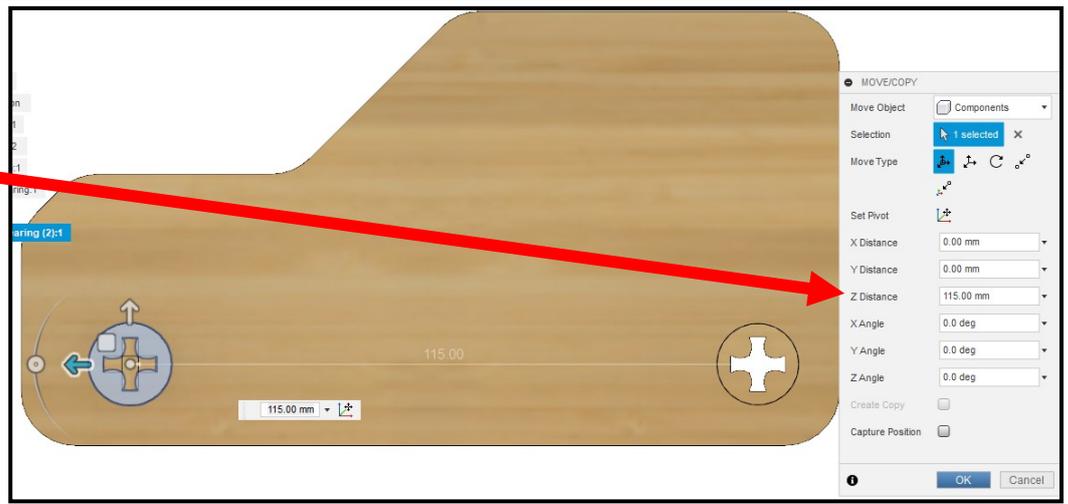
90. Repeat step **88** but this time on the other side. Selecting the **front axle** as the **target body** and the **mirrored joint** as the **tool body**.



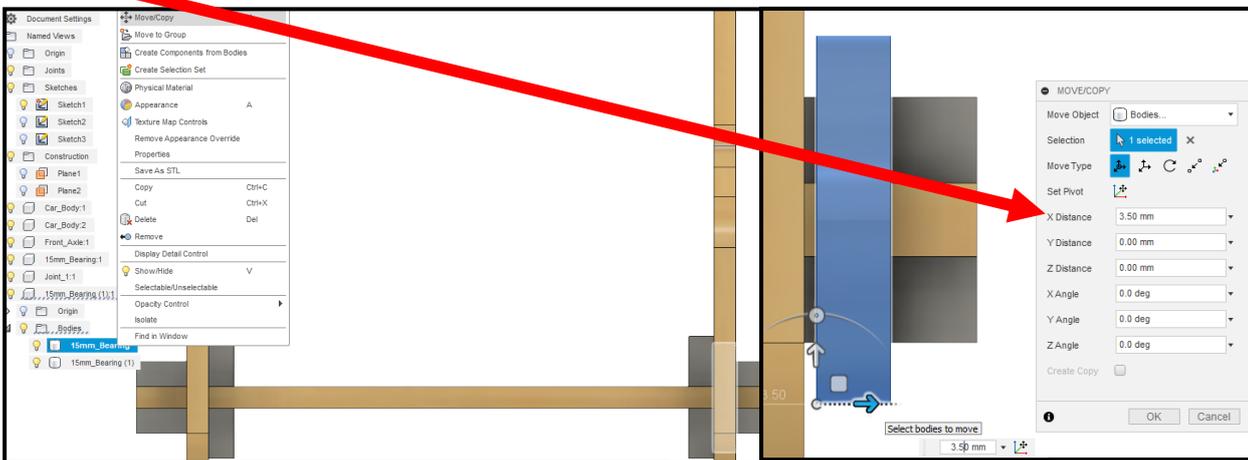
91. Right click on the bearing from the menu and select **'Copy'**. Then right click on the white area and select **'Paste New'**.



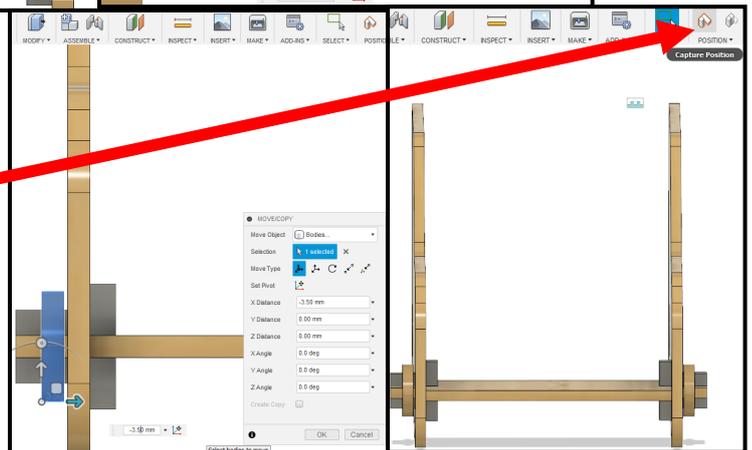
92. Set the **Z Distance** to **115mm**. This will move the bearing 115mm to the left.



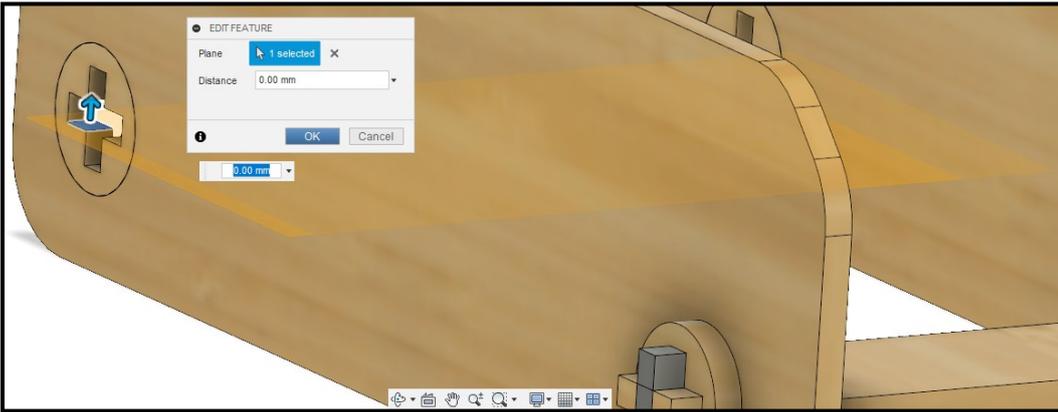
93. Right click on this component in the menu and select **'Move/Copy'**. Set the **'X Distance'** to **3.5mm**.



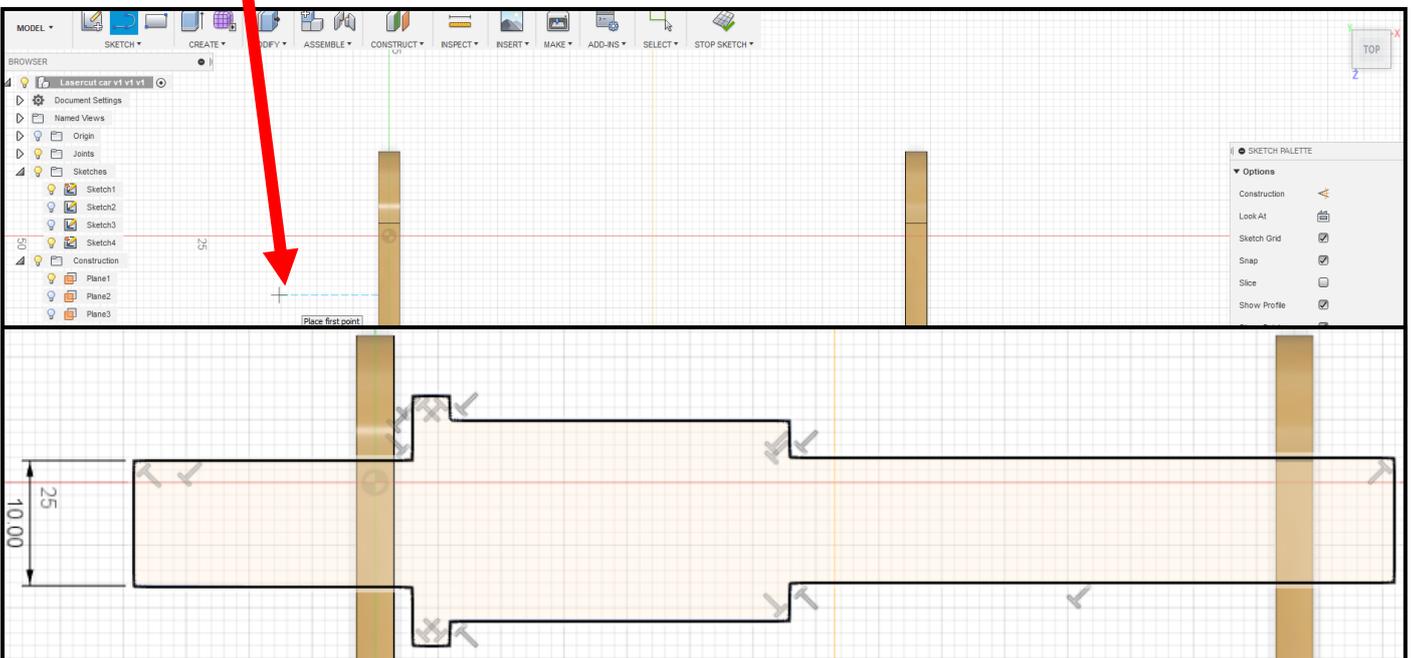
94. Repeat step 93 on the other side, this time setting the distance to **-3.5mm**.
Once the bearings are correctly positioned click on **'Capture Position'** to finish.



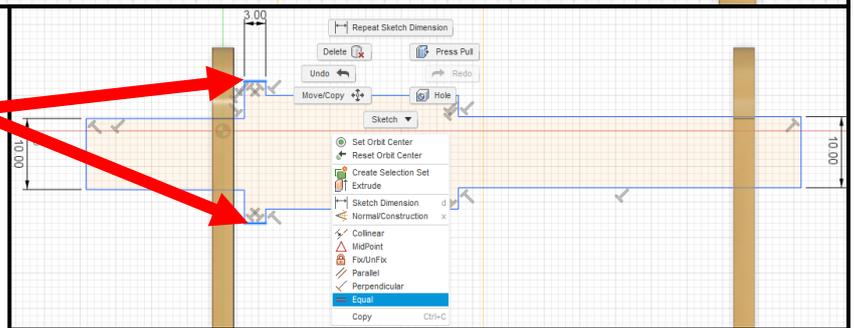
95. Under the construct tab select 'Offset Plane' and chose the flat face on the bearing shown below as the starting point. You may need to zoom in to select this.



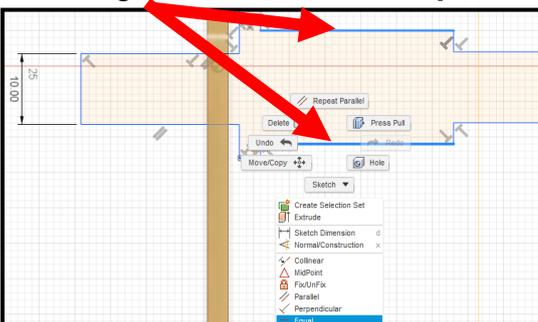
96. Start a new sketch on this plane and select the line tool, then draw the shape shown. At this stage it does not have to be exact.



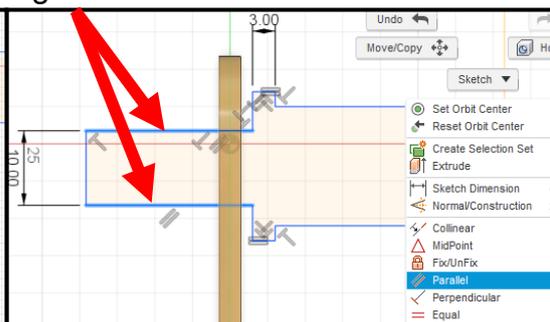
97. Hold down shift on the keyboard and select the two lines shown, right click and select 'Equal'.



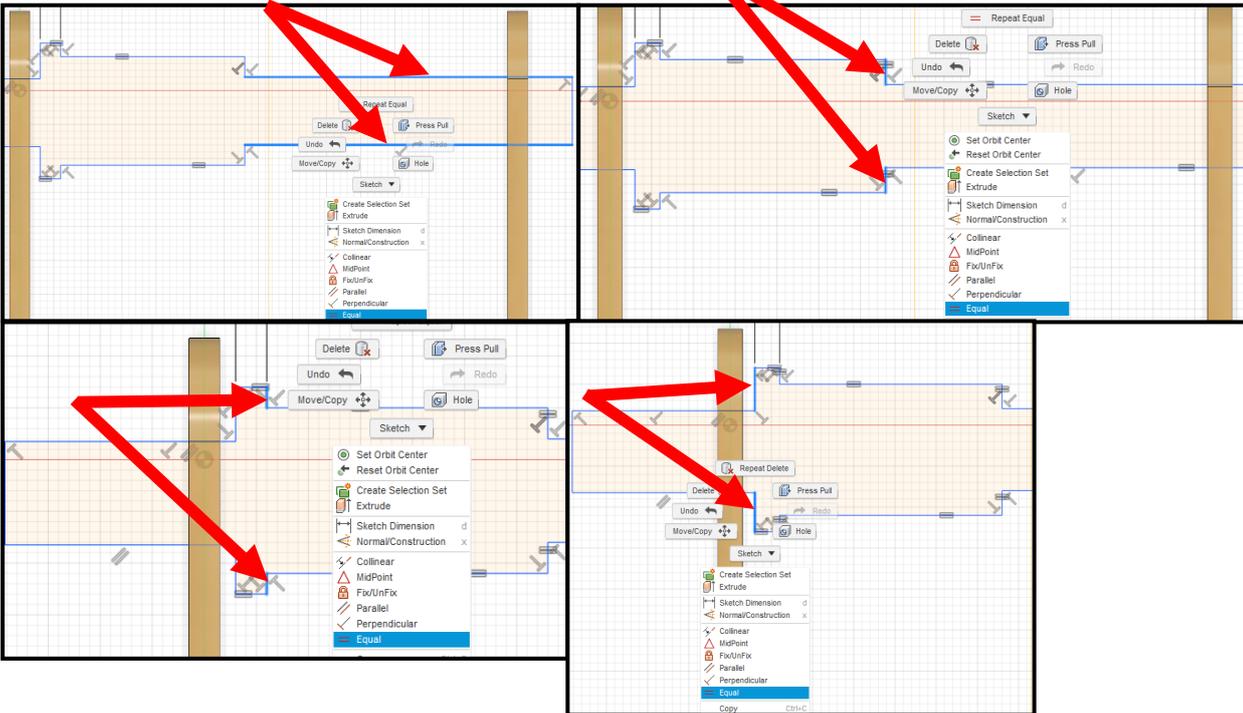
98. Hold down shift, select the two lines, right click and select 'Equal'.



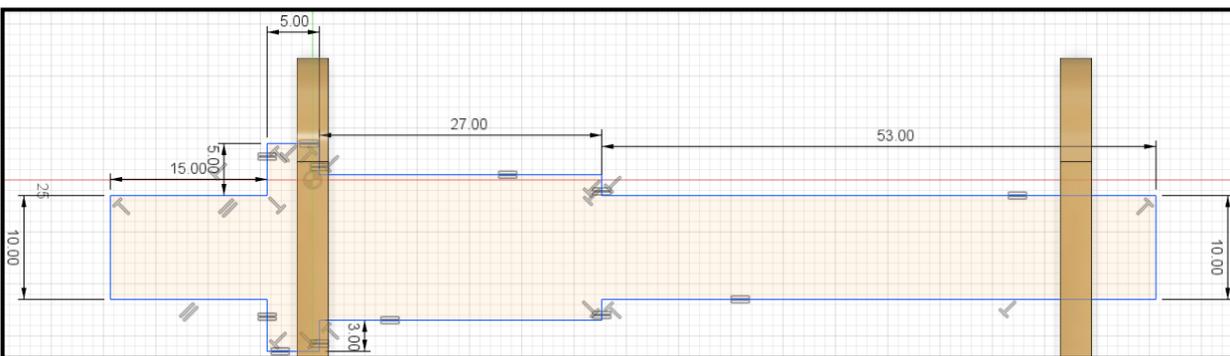
99. Hold down shift, select the two lines, right click and select 'Parallel'.



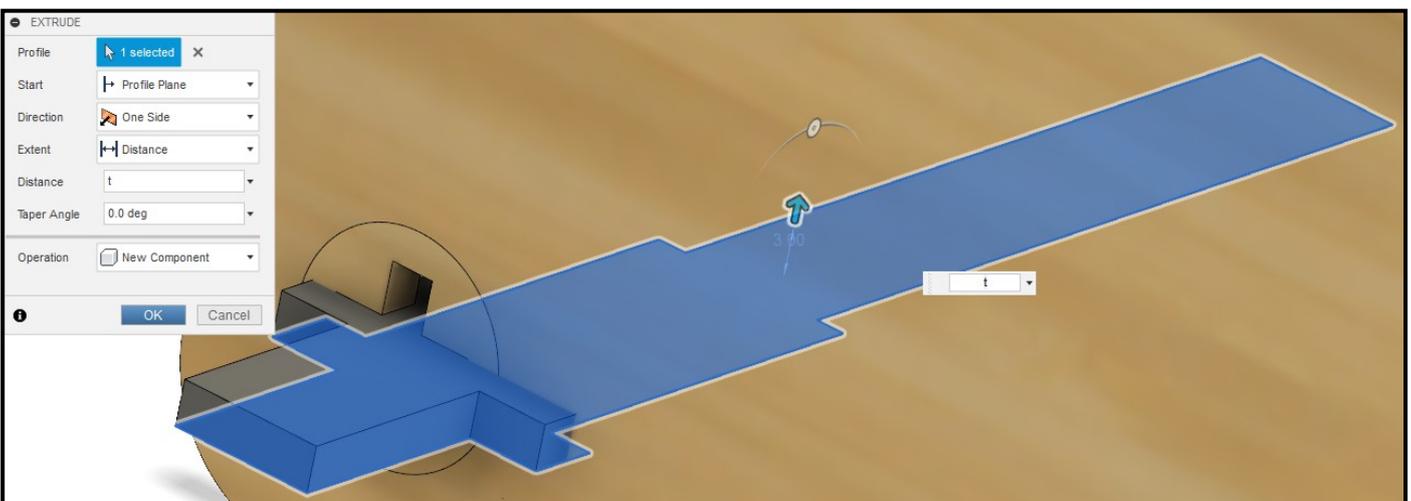
100. Hold down shift, select the two lines, right click and select **'Equal'**. Do this for each of the following 4 images.



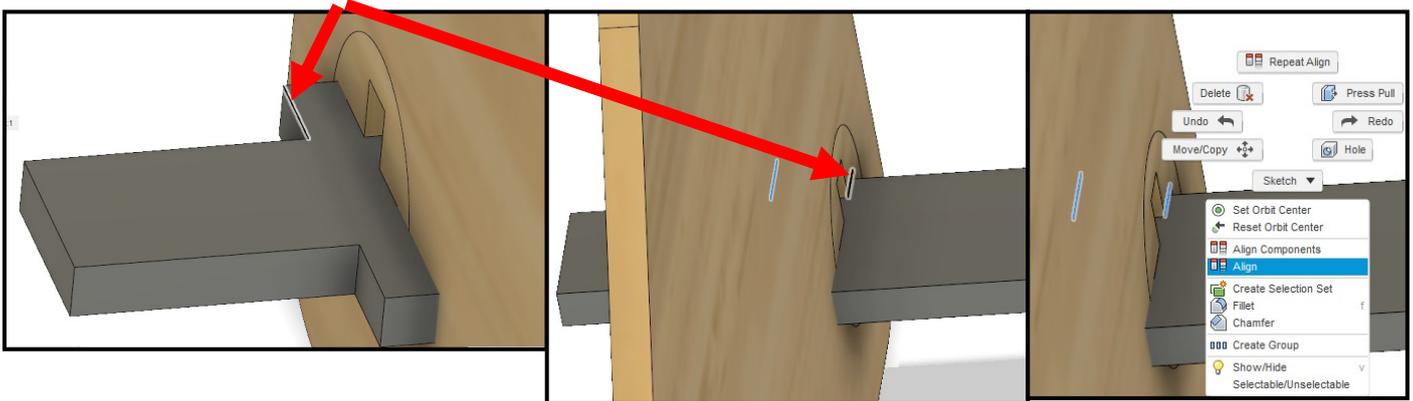
101. Add the following dimension labels for this shape.



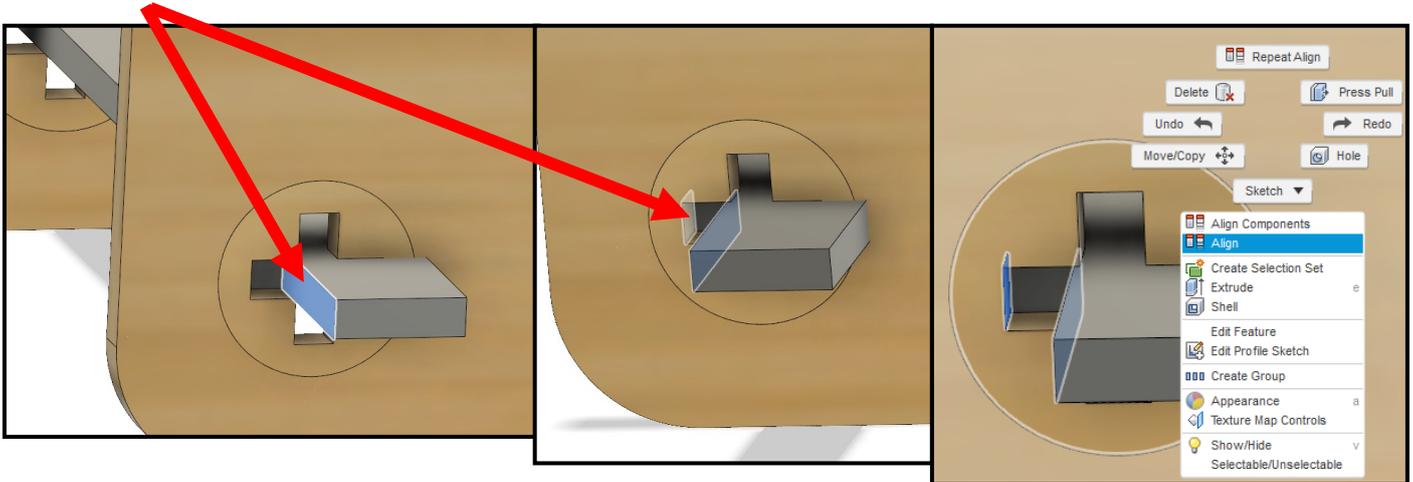
102. Finish the sketch and then **extrude** the shape. Set the distance as **'t'**, and the operation as **'New Component'**.



103. Hold down shift and select the two edges shown. The first on the new component and the second on the straight edge of the bearing. Again you may have to zoom in to select this edge that is beside the spline that was drawn earlier. Right click and select 'Align'.



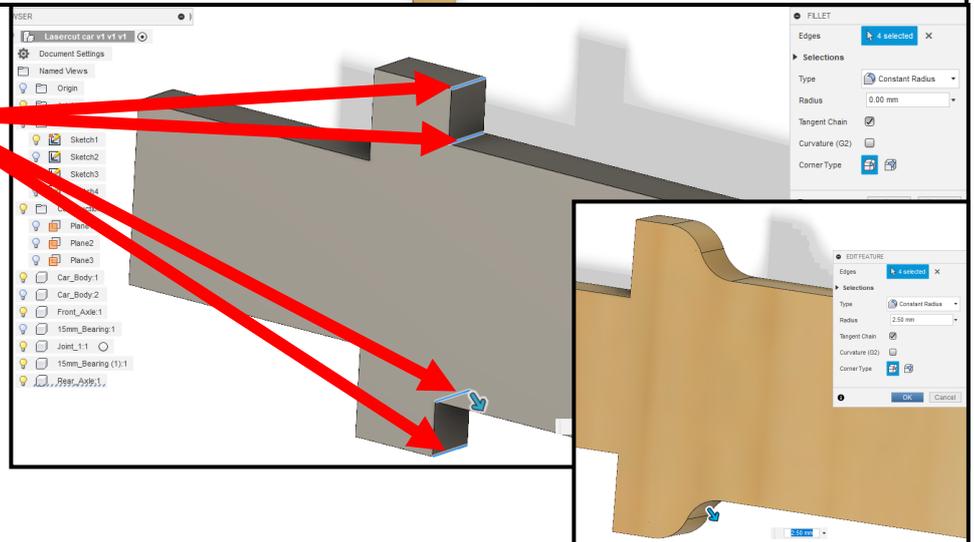
104. Now holding down shift, select the shown face of the new component, then select the face shown on the bearing. Right click and select 'Align'.



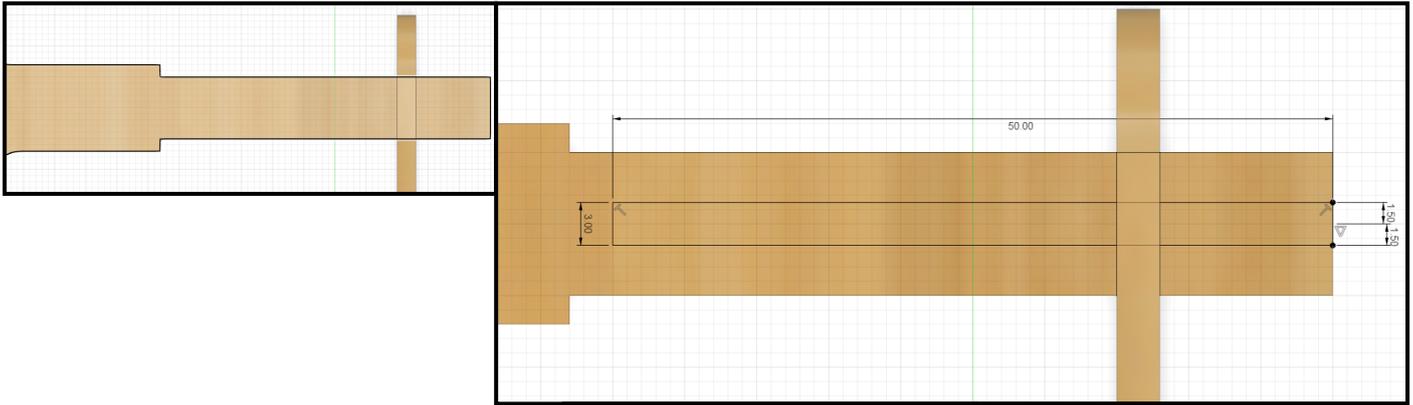
105. Re-name this new component 'Rear_Axle'.



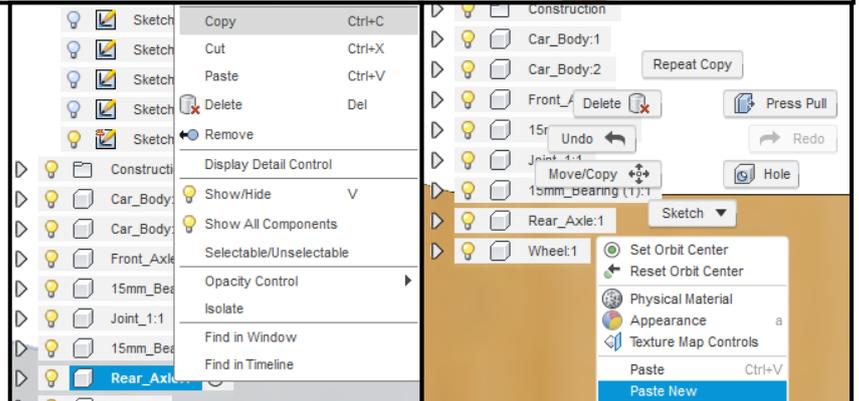
106. Under the **modify** tab select the 'Fillet' option. Then select the 4 edges shown and set the radius to 2.5mm.



107. Start a new sketch and chose the top face of the rear axle as the sketch plane. Using the line tool draw the rectangular shape shown in the centre of the axle and add the dimensions.

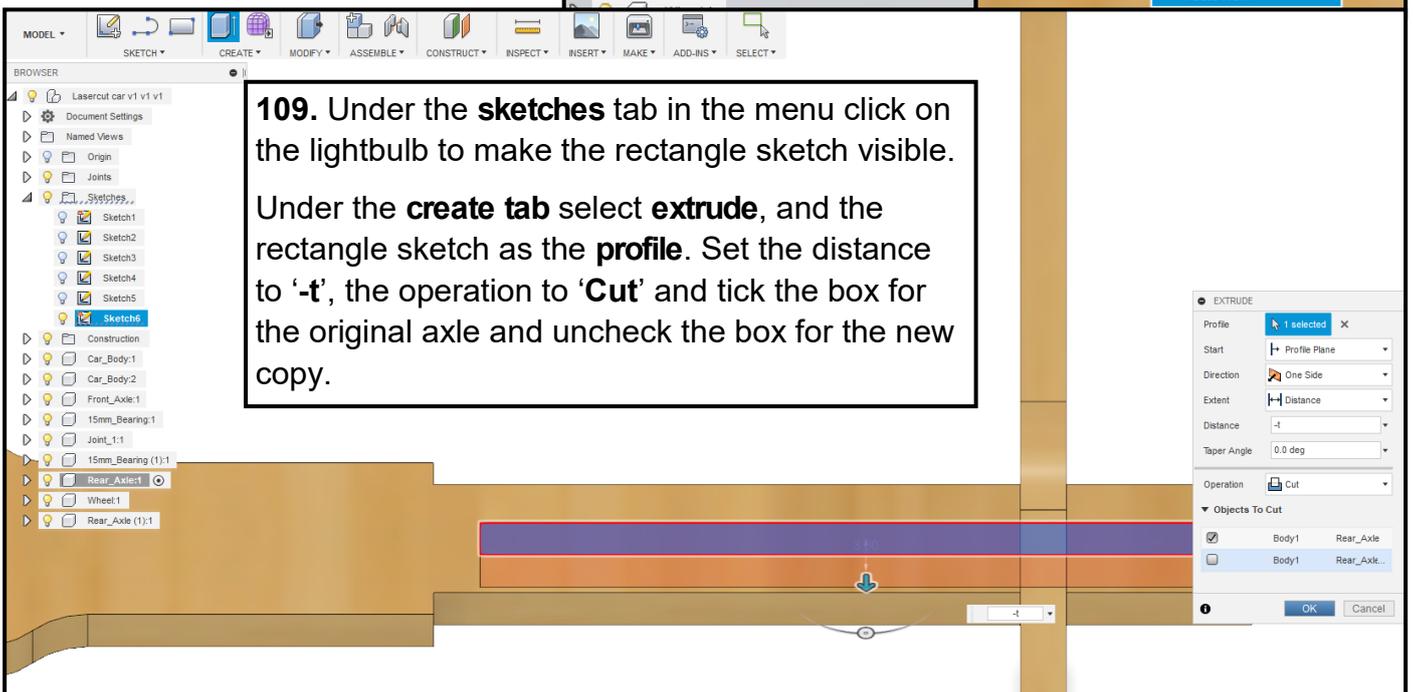


108. Right click on the rear axle in the menu and select 'Copy', then right click on the white area and select 'Paste New'.

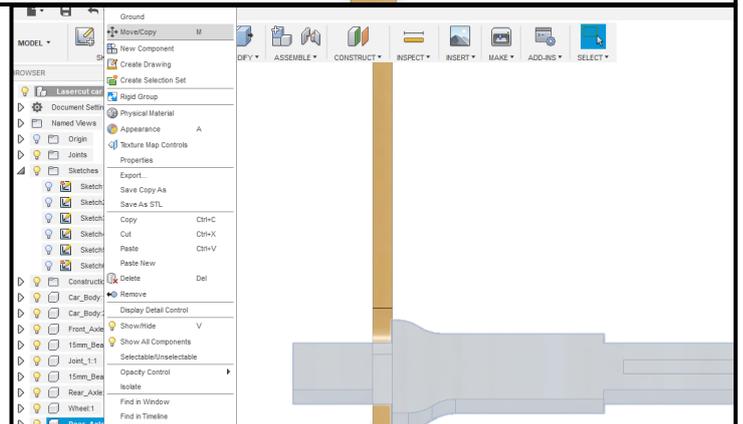


109. Under the **sketches** tab in the menu click on the lightbulb to make the rectangle sketch visible.

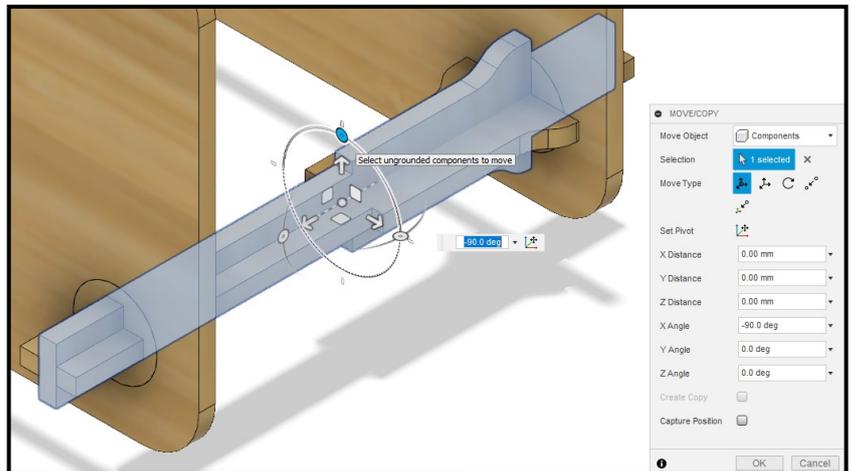
Under the **create** tab select **extrude**, and the rectangle sketch as the **profile**. Set the distance to '-t', the operation to 'Cut' and tick the box for the original axle and uncheck the box for the new copy.



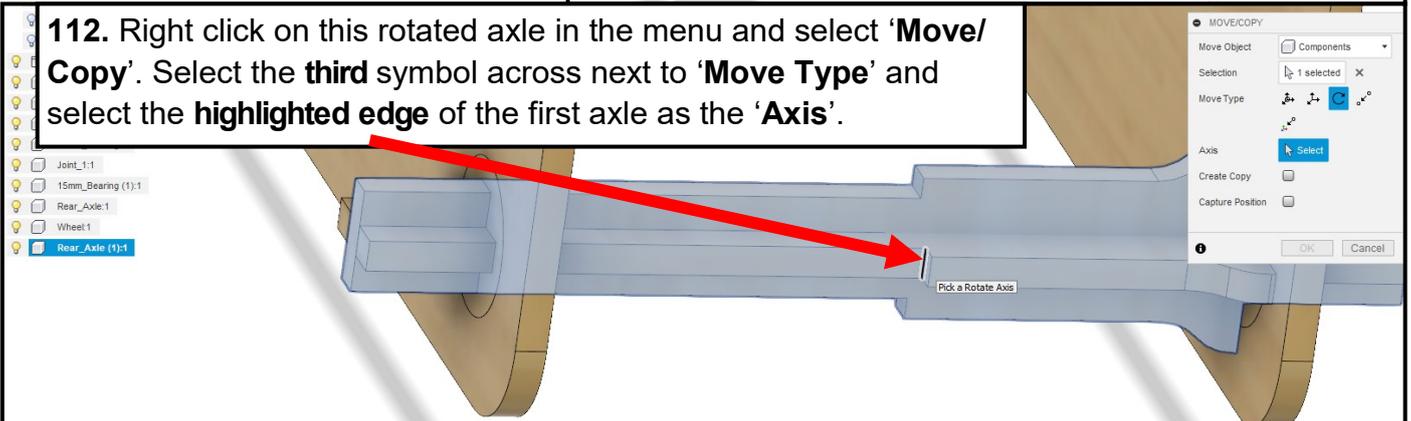
110. Right click on the second rear axle and select 'Move/Copy'.



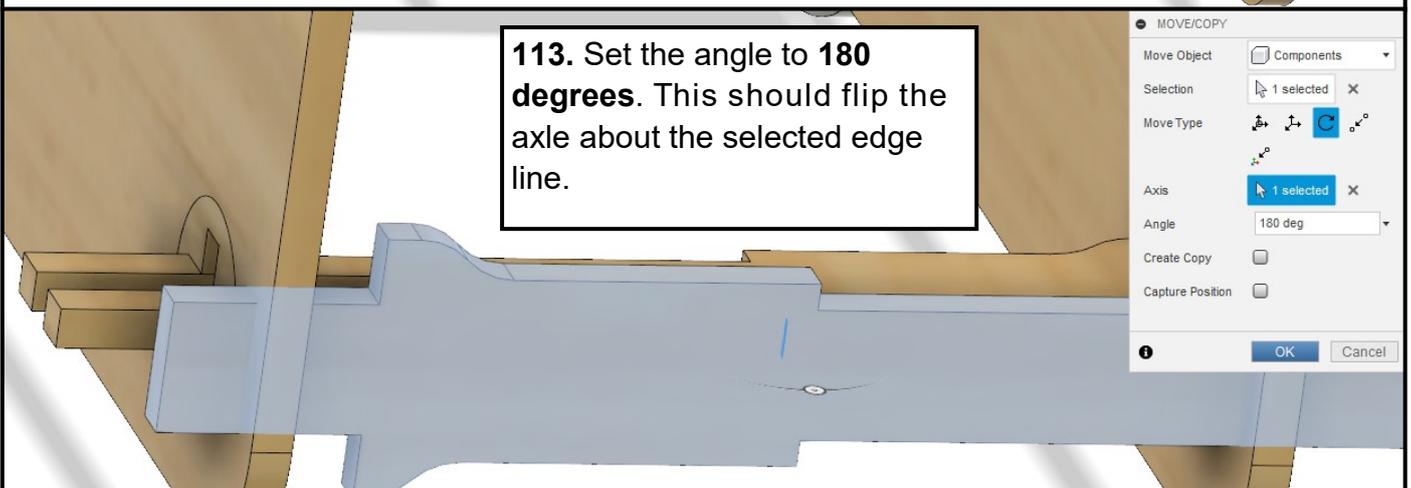
111. Set the **X Angle** to '90 degrees' to rotate this new component.



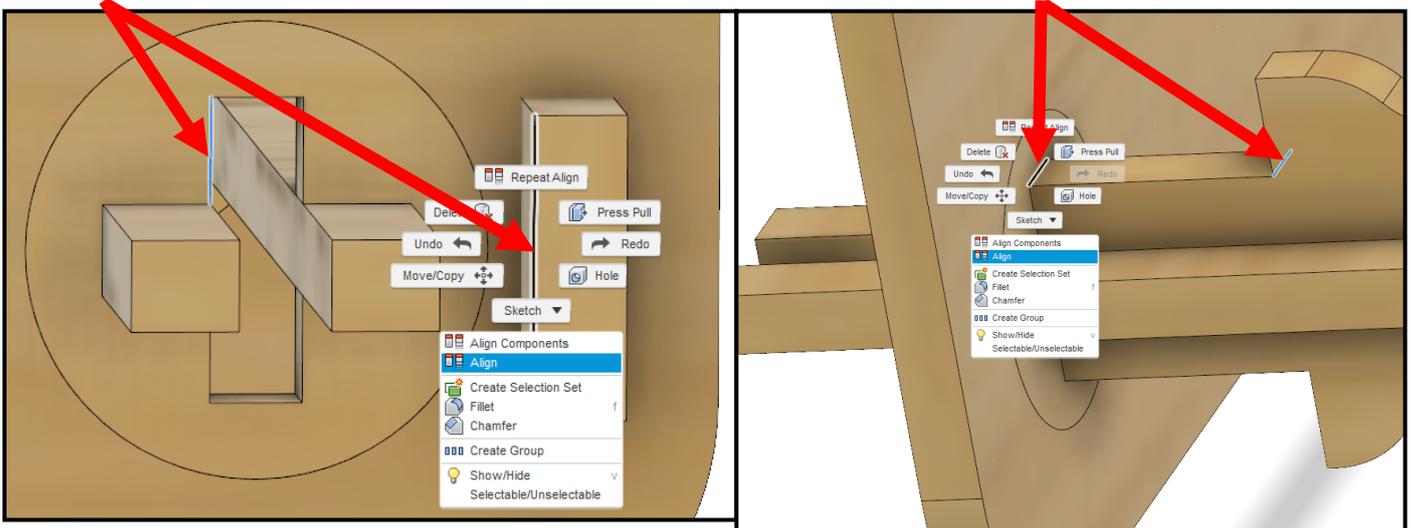
112. Right click on this rotated axle in the menu and select '**Move/Copy**'. Select the **third** symbol across next to '**Move Type**' and select the **highlighted edge** of the first axle as the '**Axis**'.



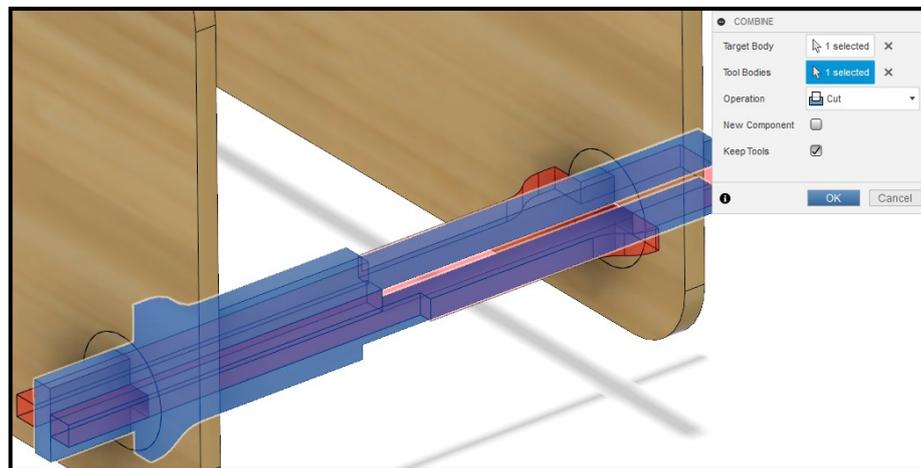
113. Set the angle to **180 degrees**. This should flip the axle about the selected edge line.



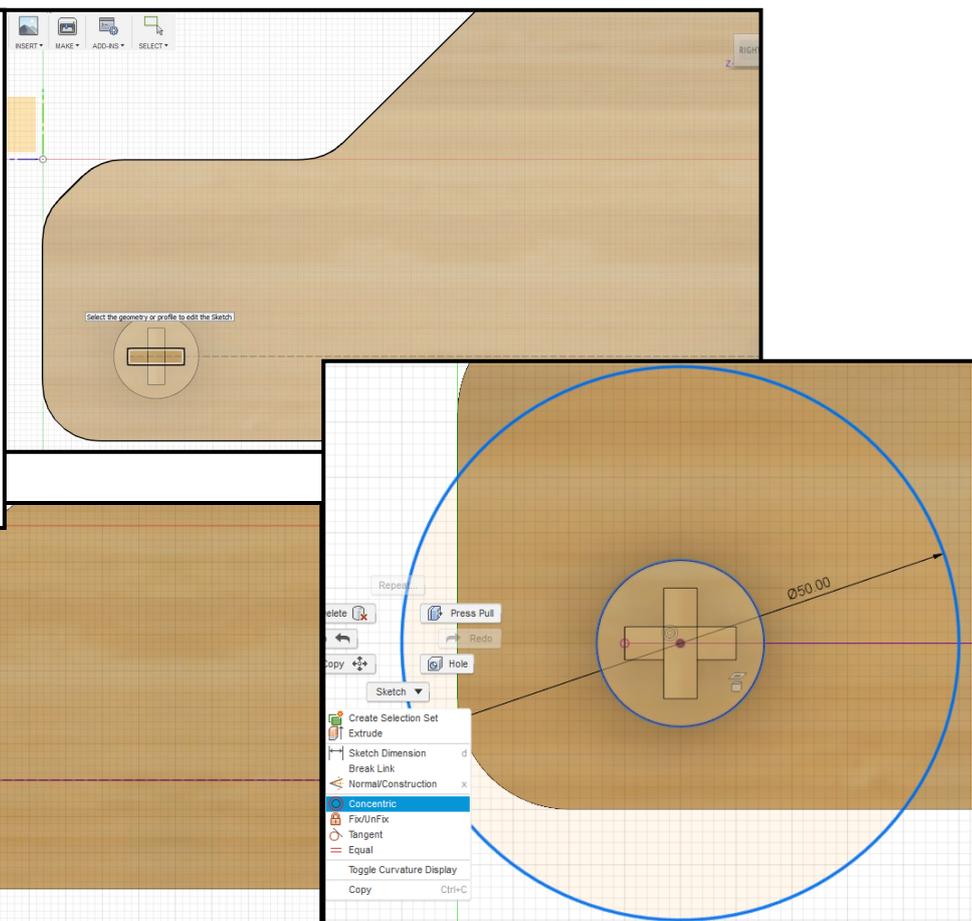
114. For each of the images below select the **blue edge first**, then the **black edge second** (remember to hold down shift to select multiple edges), right click and select '**Align**'.



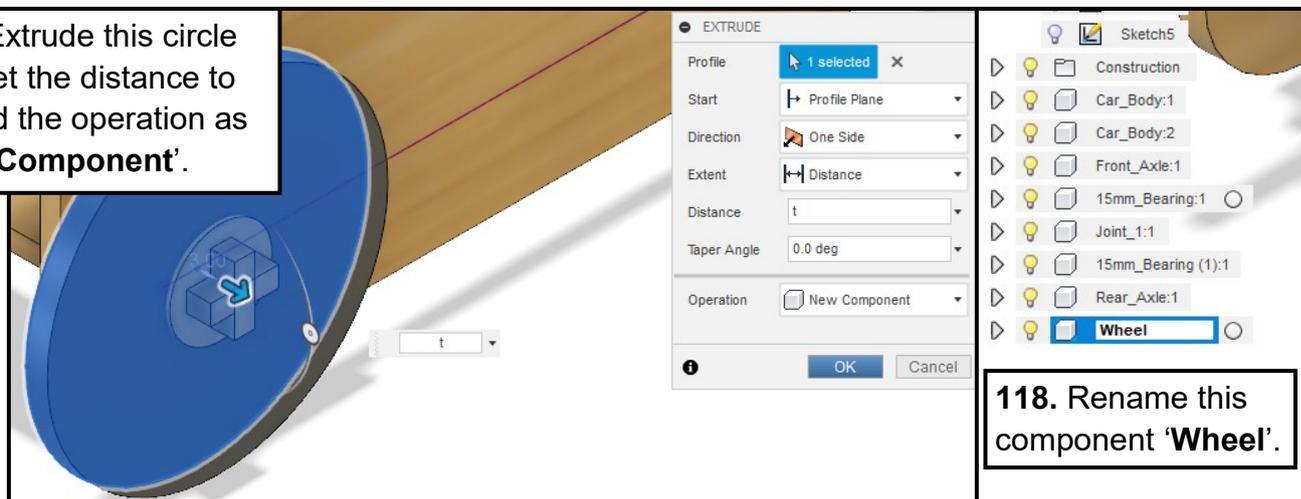
115. Now that the axles are aligned use the **combine** tool found under the **modify** tab, select the second axle in blue as the target body, and select the original axle as the tool body. Make the operation a **cut** and tick the box next to 'Keep Tools'.



116. Start a sketch and select the side of the car body as the sketch plane. Draw a **50mm diameter, center diameter circle** from the centre of the cross of the front axle. If isn't exactly in the centre hold down shift, select the centre of the circle and the centre of the cross, right click and select '**Concentric**'.

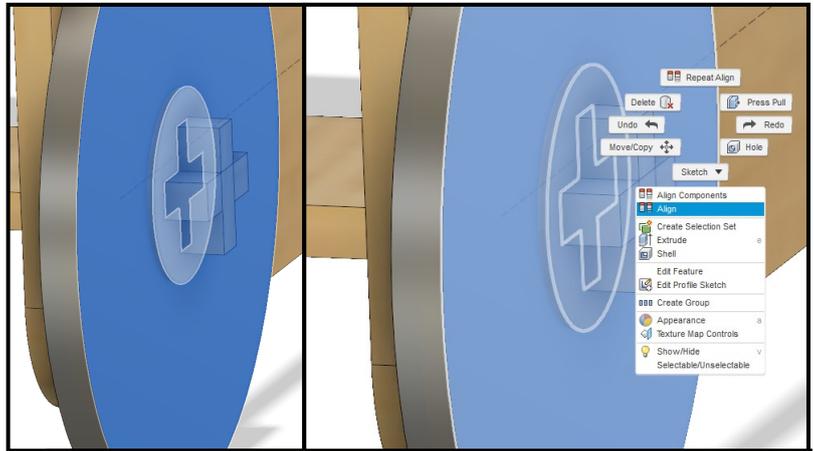


117. Extrude this circle and set the distance to '**t**', and the operation as '**New Component**'.

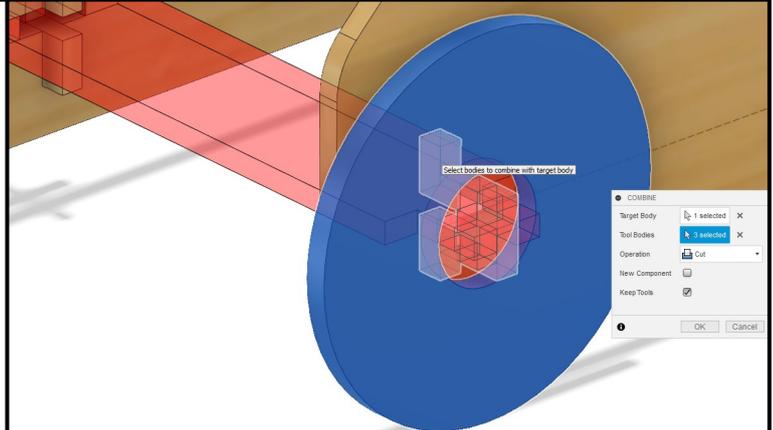


118. Rename this component '**Wheel**'.

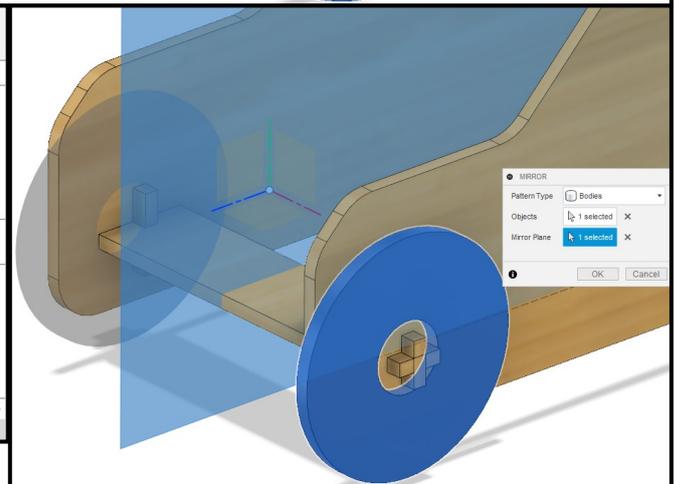
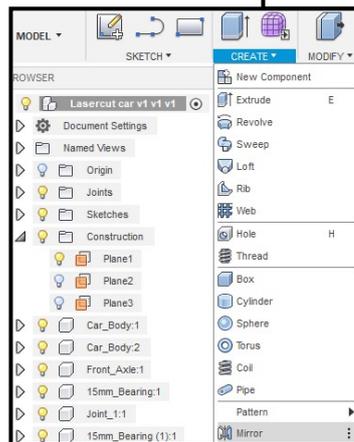
119. Select the face of the wheel, hold down shift and select the front face of the bearing. Right click and select **'Align'**.



120. Using the combine tool, select the wheel as the target body, the **front axle, bearing and joint** as tool bodies and make the operation a **'Cut'**. Tick the box next to **'Keep Tools'**.

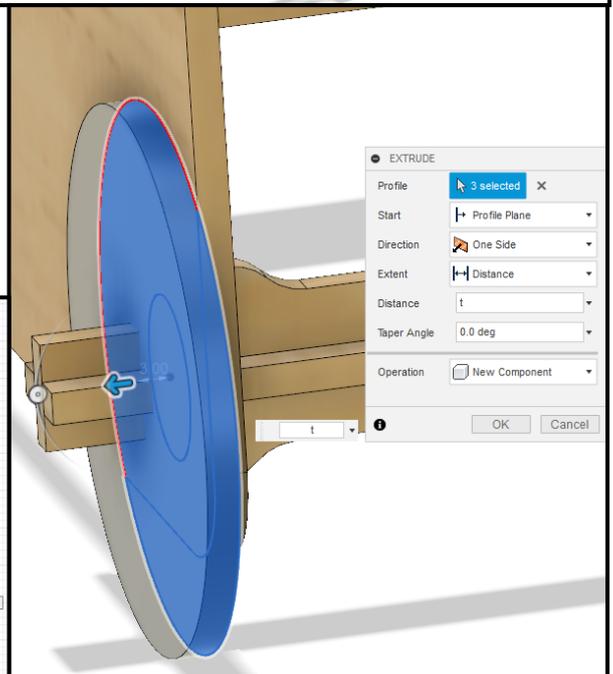
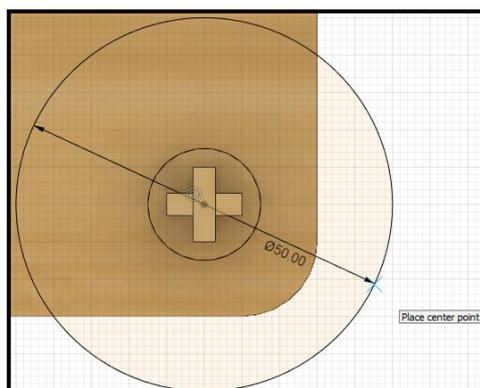


121. Next, under the **create tab** select **'Mirror'**. Click the **lightbulb** next to the midplane shown to view it. Select the wheel as the object and this plane as the mirror plane.

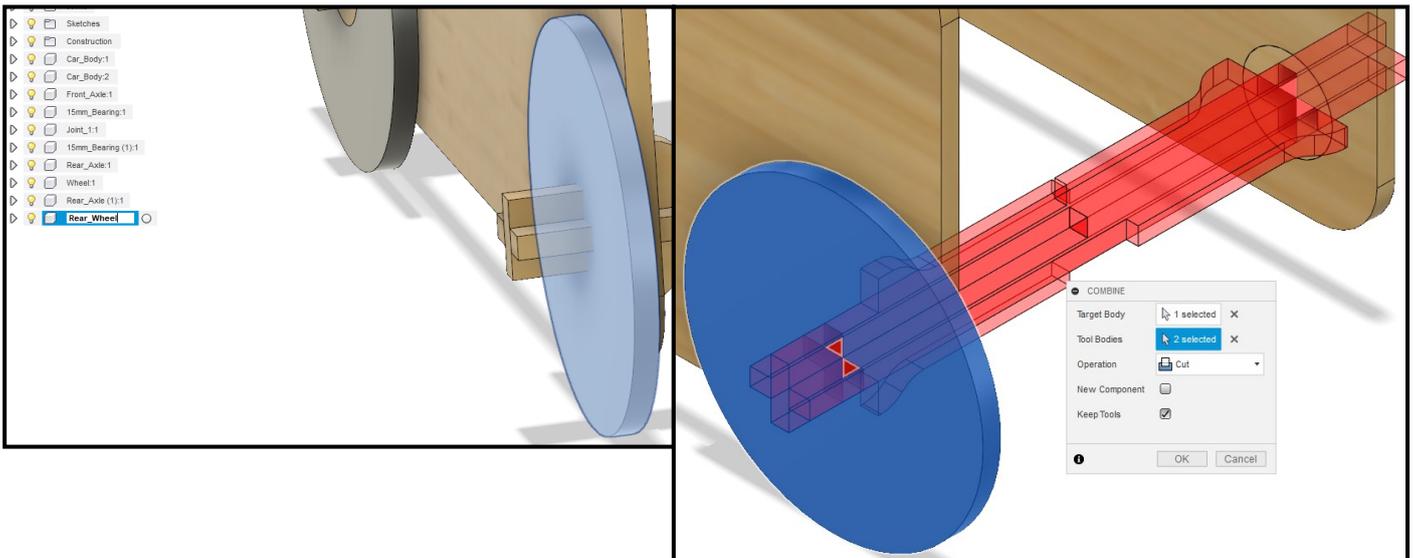


122. Now draw another **50mm diameter** circle on the same plane (the car body) and make sure the centre of the circle is aligned with the cross.

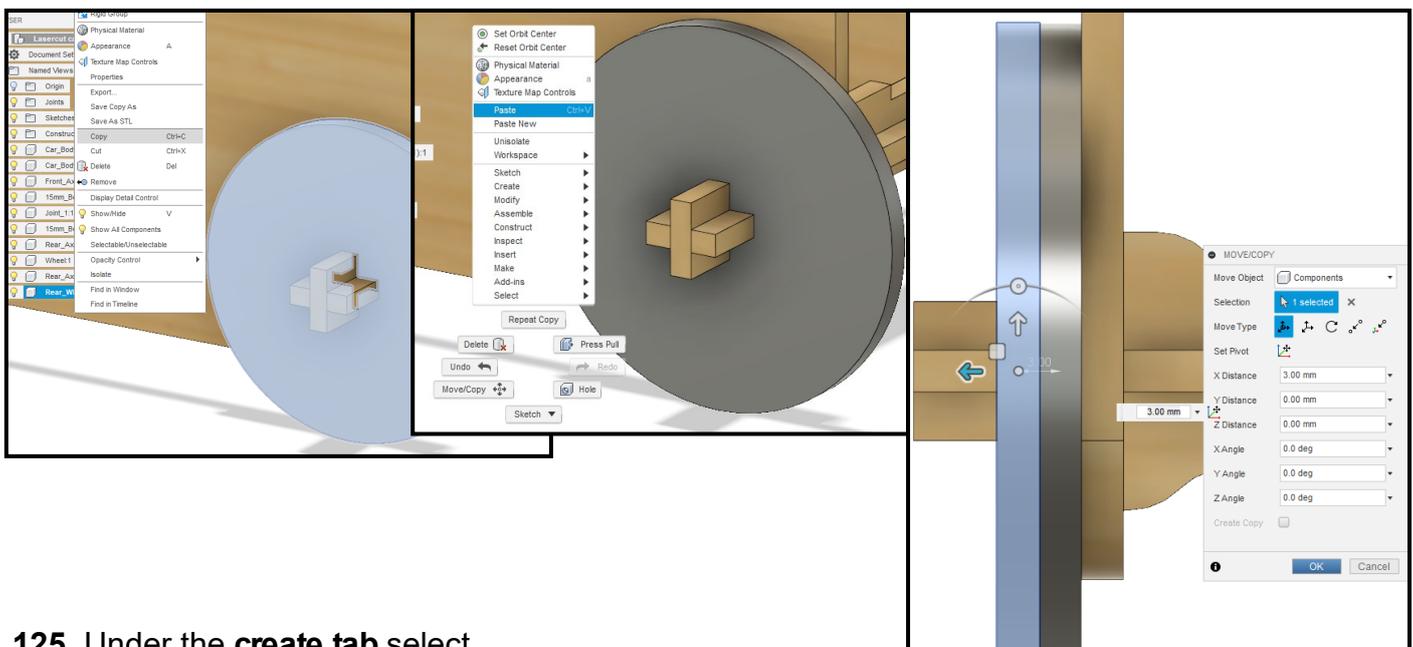
Extrude this whole circle, making sure the full face is selected. Set the distance to **'t'** and the operation as **'New Component'**.



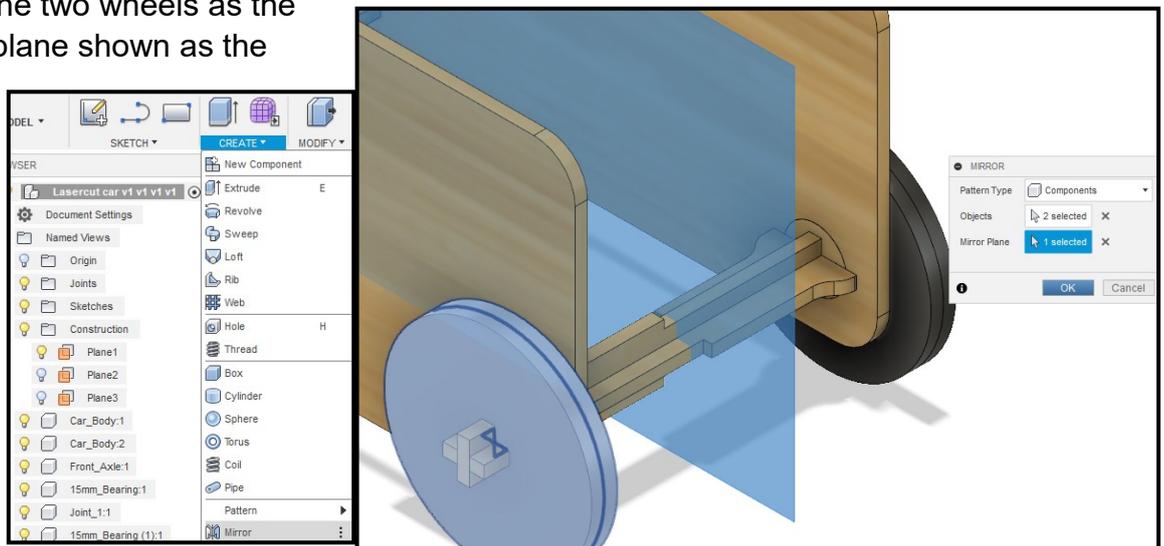
123. Re-name this component 'Rear_Wheel'. Then use the combine tool, select the wheel as the target body and the two rear axles as the tool bodies. Make the operation a **cut** and tick the box next to 'Keep Tools'.



124. Now right click on the wheel and select 'Copy', then right click on the white area and select 'Paste'. Set the X Distance of the copied wheel to 3mm to move it next to the original wheel.

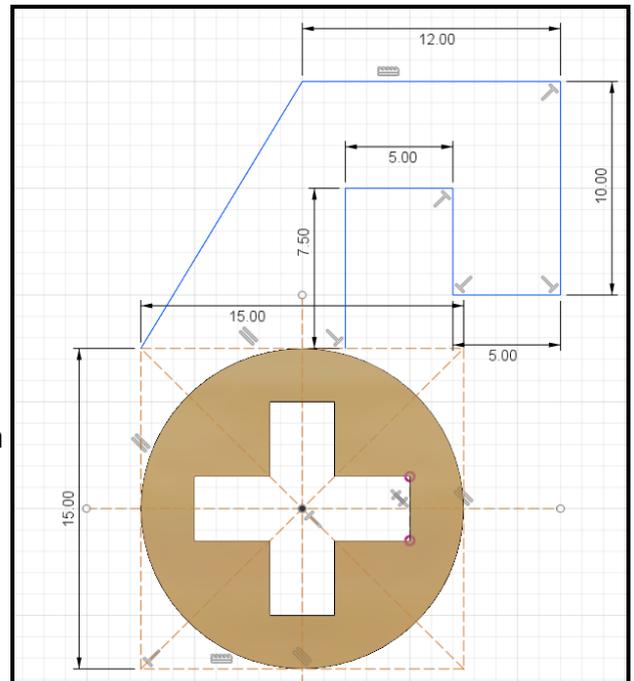


125. Under the **create tab** select 'Mirror'. Select the two wheels as the objects and the plane shown as the mirror plane.

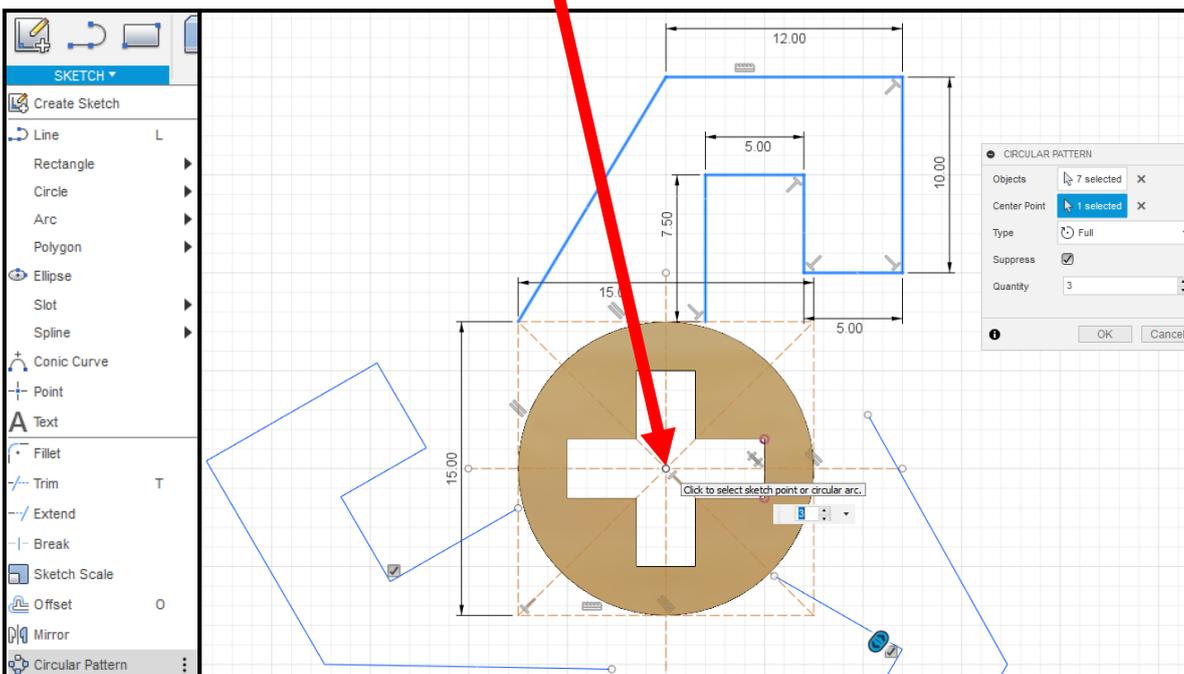


126. Hide all objects around the rear 15mm bearing using the lightbulbs in the left menu. Start the sketch on the same **midplane** used in step **125**. Use a construction center rectangle (draw a normal rectangle and use the rightmost menu to make it a construction piece of geometry), and draw it from the centre of the bearing and make it 15mm long and 15mm high.

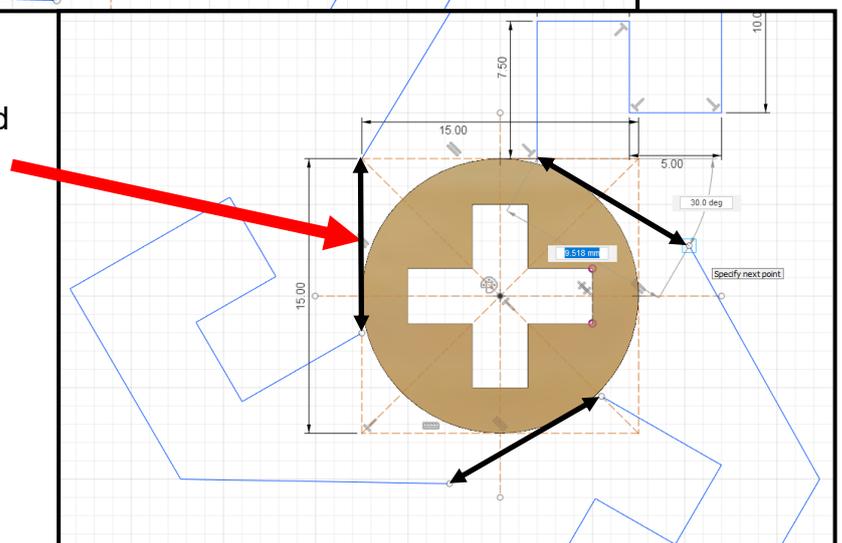
Start the sketch using the line tool, making sure to turn off construction, and draw it from the top left corner of the construction rectangle.



127. Under the **sketch tab** select '**Circular Pattern**'. Select each line of the sketch that was just drawn as the objects. Select the centre point of the bearing as the '**Center Point**'. Set the '**Quantity**' to **3**.

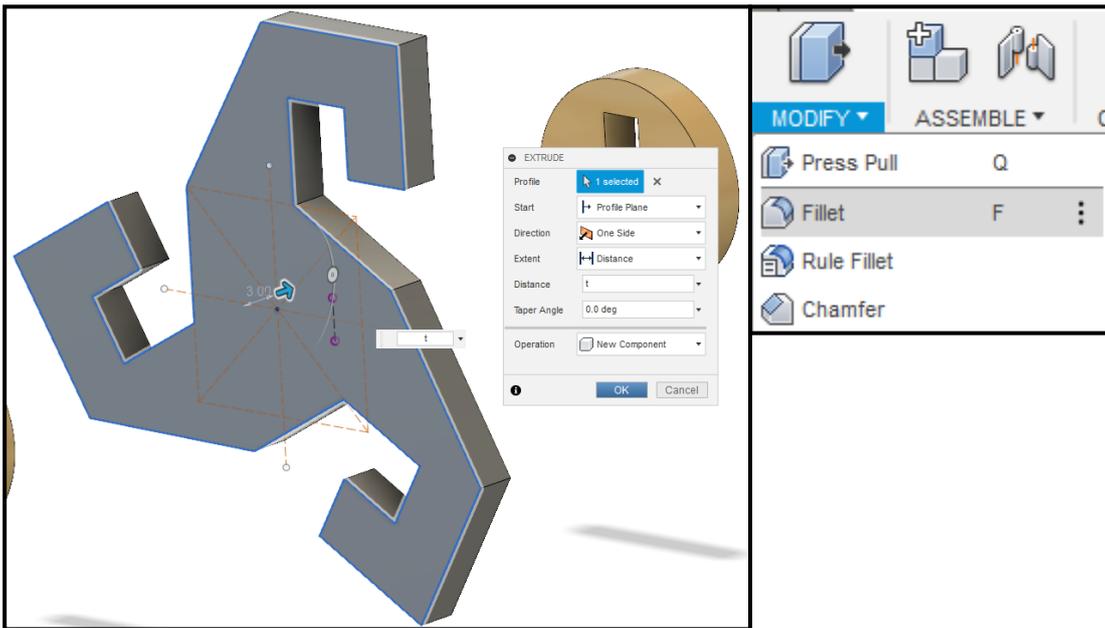


128. Use the line tool, draw a line connecting each pattern. You will need to draw **three** lines in total in the locations shown.

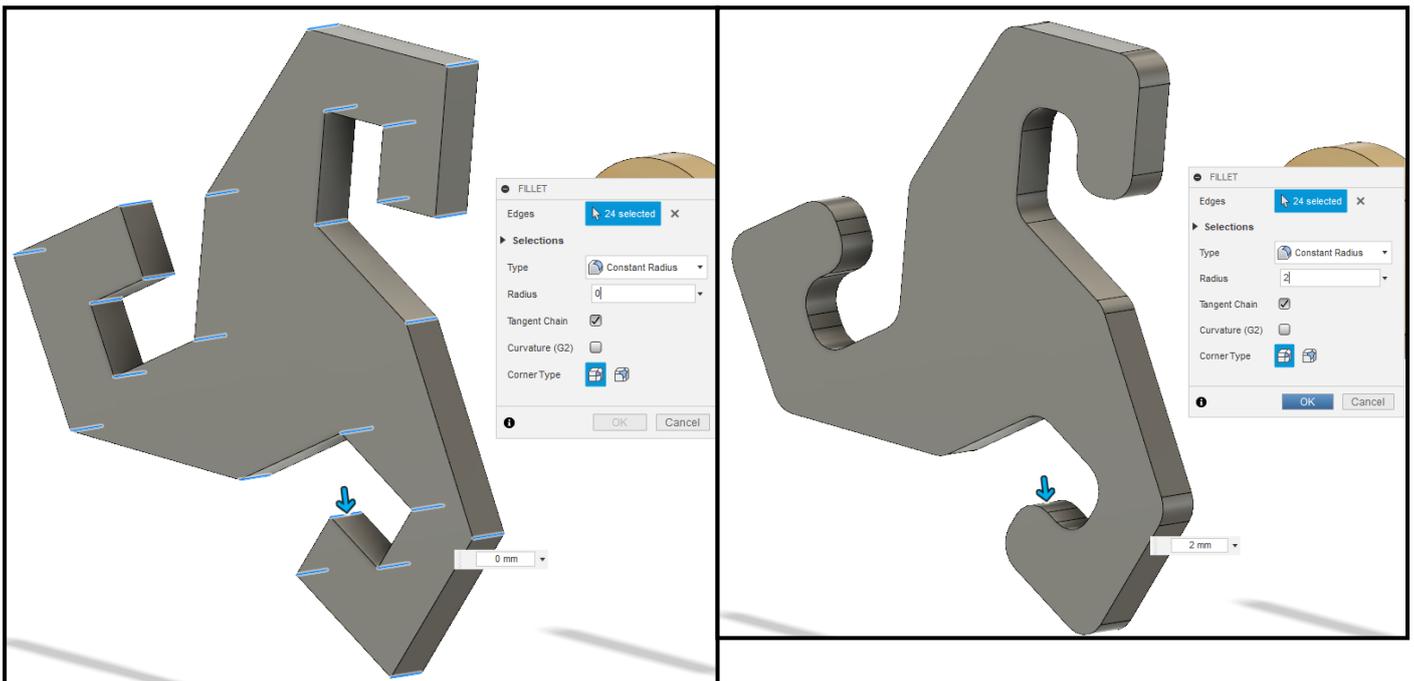


129. Extrude this new sketch by a distance of 't' and make the operation 'New Component'.

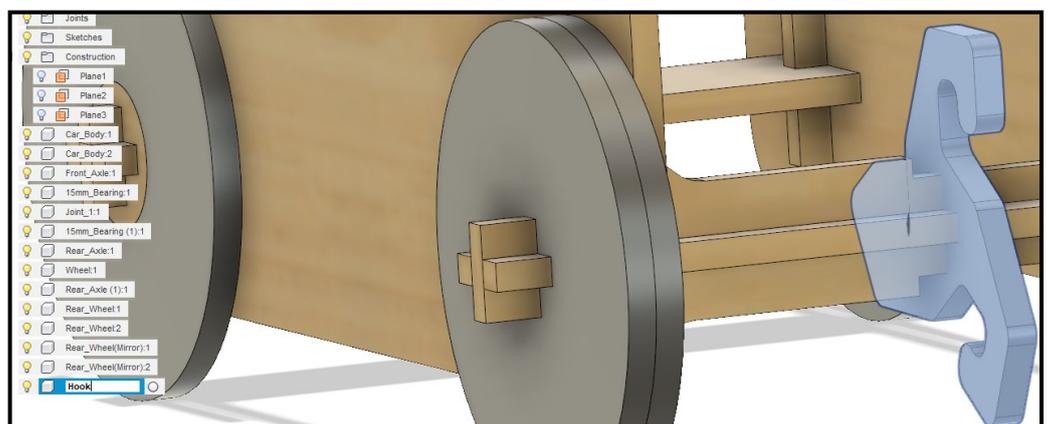
130. Under the modify tab select 'Fillet'.



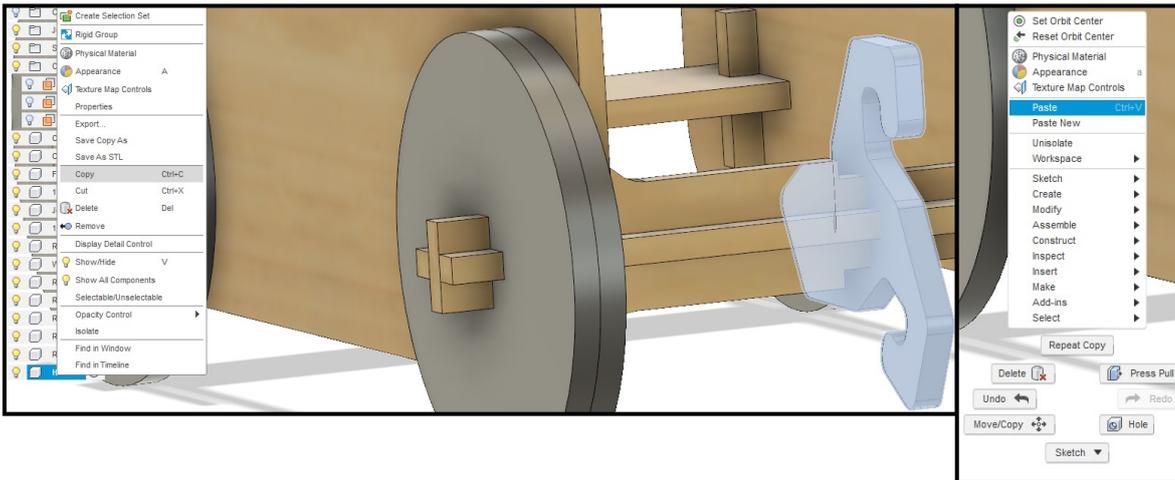
131. Select all of the outer edges of the shape shown below. There are **24 in total**. Once all of the edges have been selected set the **Radius** to **2**.



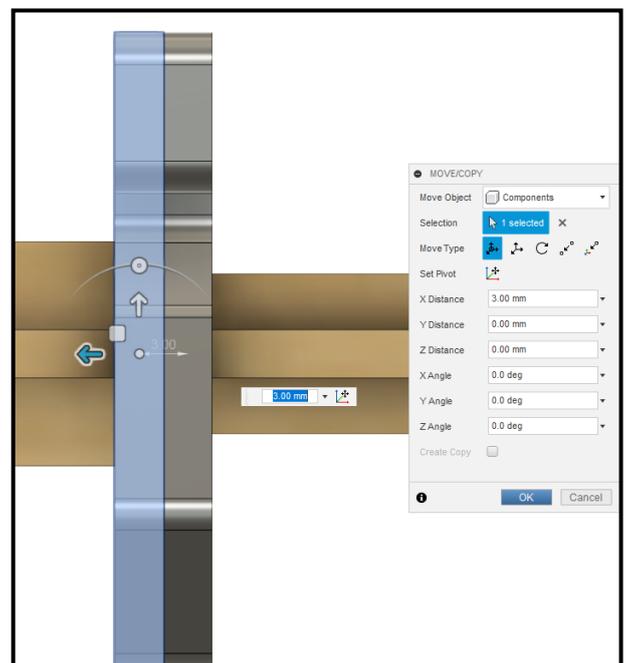
132. Re-name this component 'Hook'.



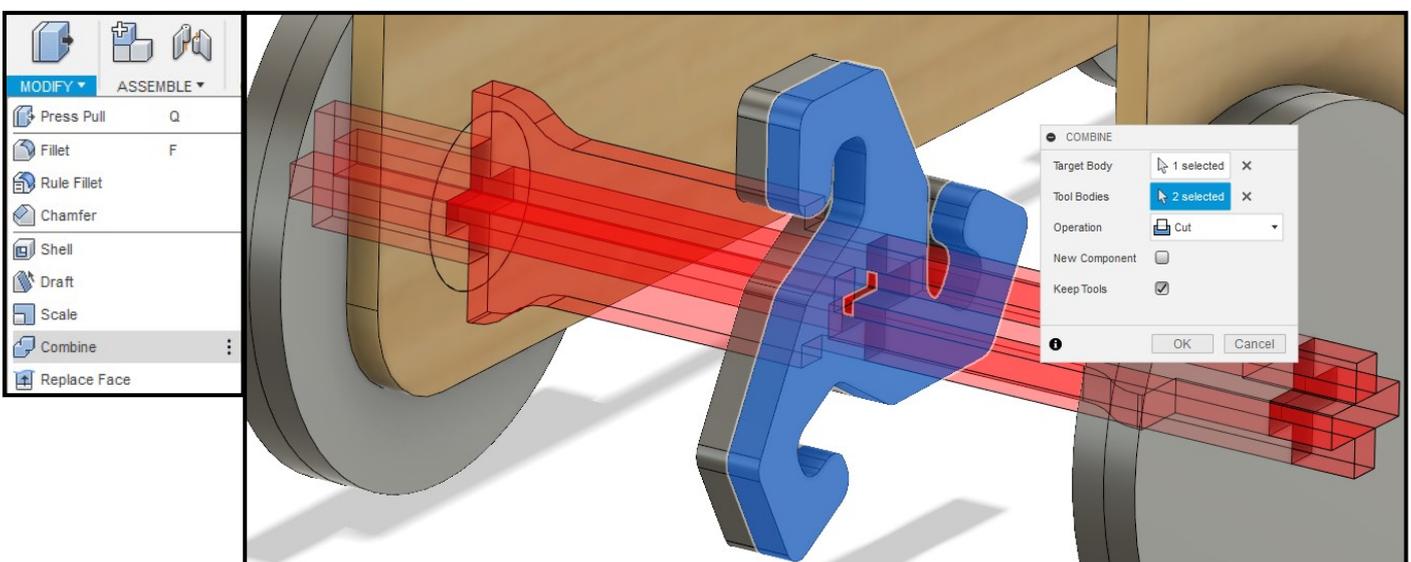
133. Right click on the hook in the left menu and **copy** it. Then as before right click on any white are and select '**Paste**'.



134. Set the **X Distance** to **3mm** for this new hook copy. Both hooks should now be side by side.

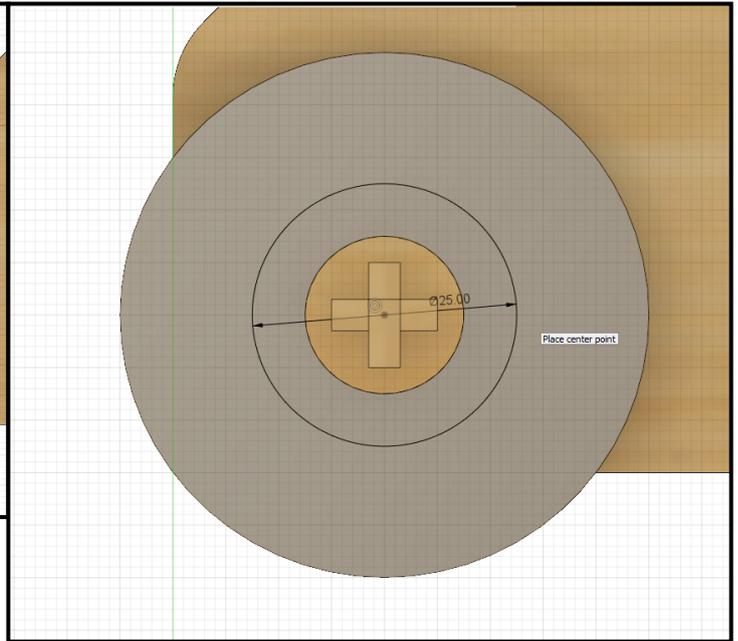
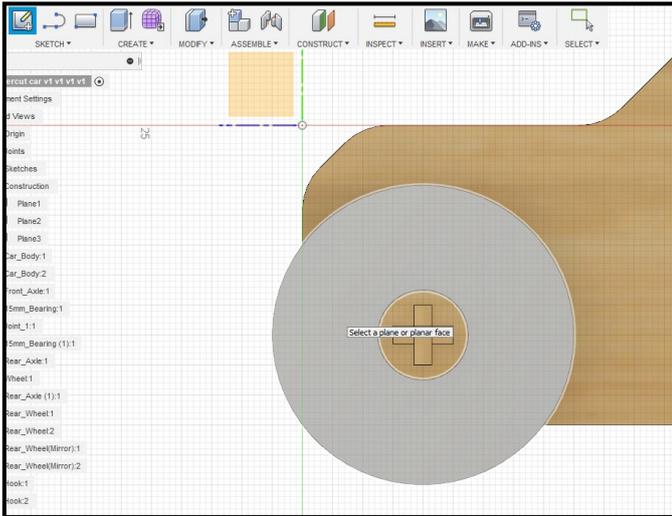


135. Using the **combine** tool, select the original hook as the target body and the **two** rear axles as



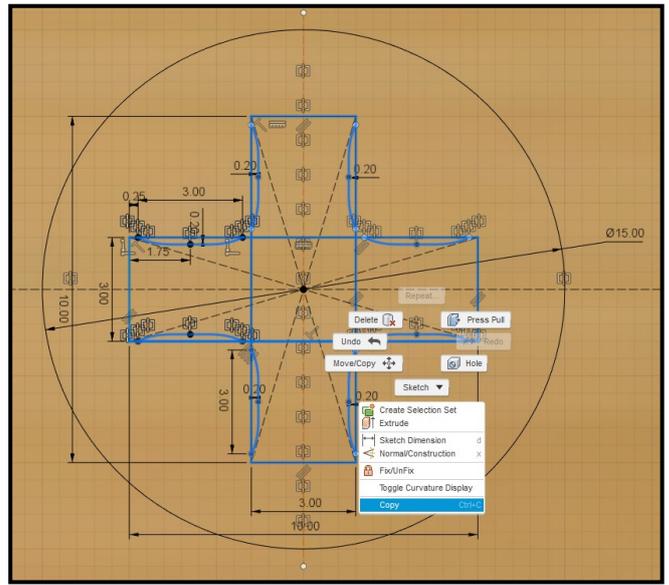
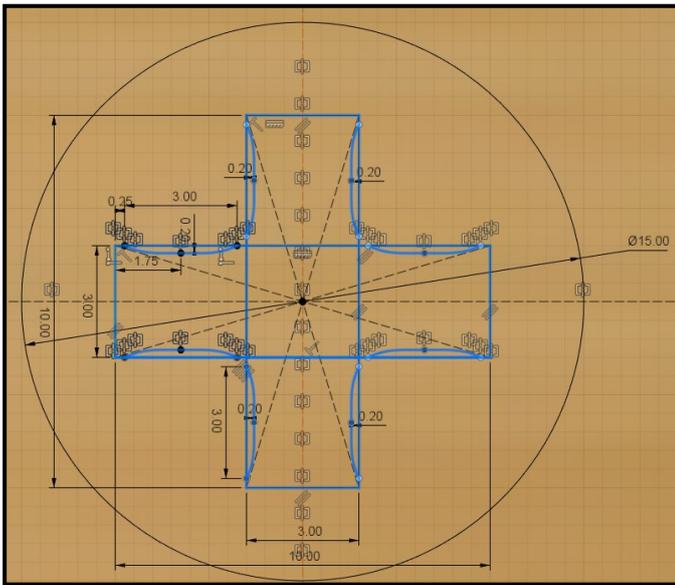
136. Start a **new sketch**, and make the side face of the front wheel the sketch plane.

From the centre of the cross shape draw a **center diameter circle** of **25mm diameter**.

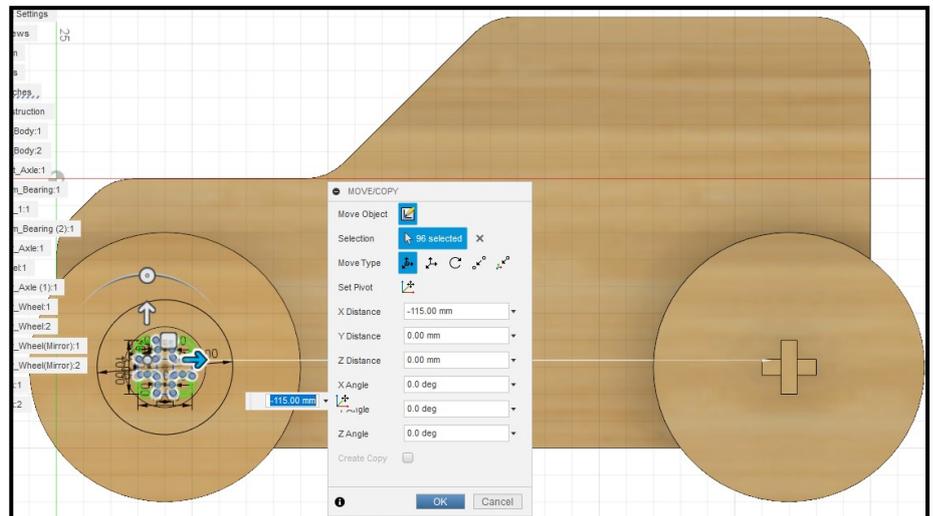


137. In the **left menu** under the sketches tab find the sketch for the **15mm bearing** and view it using the lightbulb. Hold down shift and select all of the blue lines shown below.

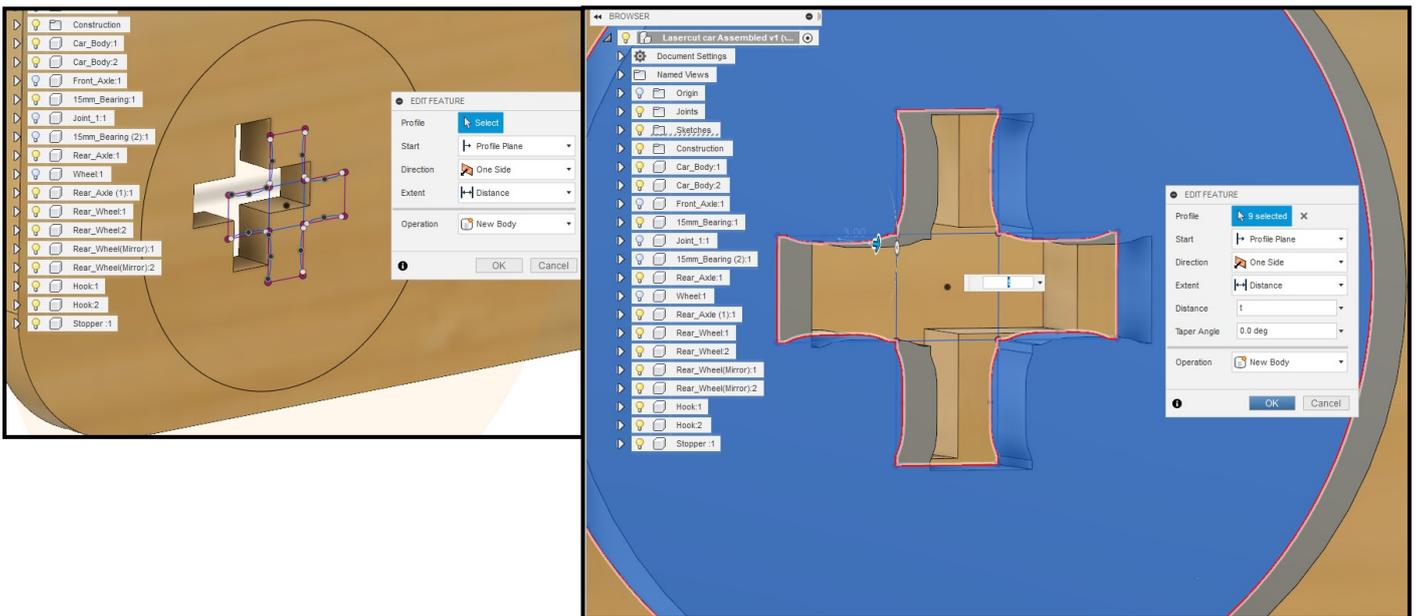
138. Right click and copy the sketch. Then right click on the white area and paste the sketch.



139. Set the **X Distance** of this new sketch as **-115mm** to move it on top of the circle that was just sketched.

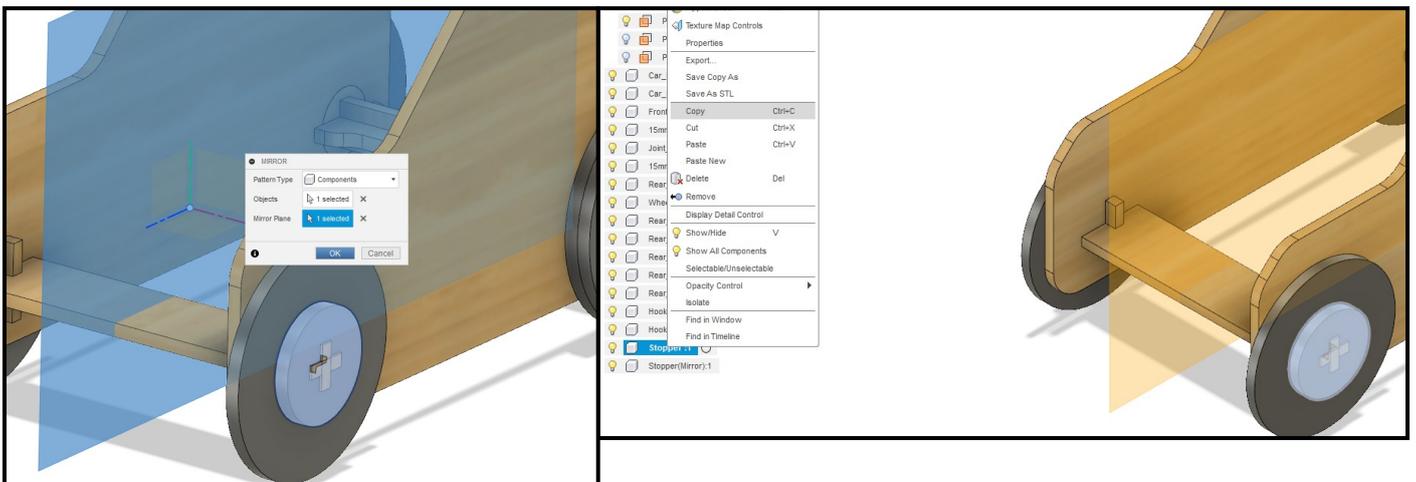


140. Start a **new extrude** and select all of the faces that make up the blue face shown below. There are **9** profiles in total. Set the distance as **'t'** and the operation as **'New Body'**. Re-name the component **'Stopper'**.

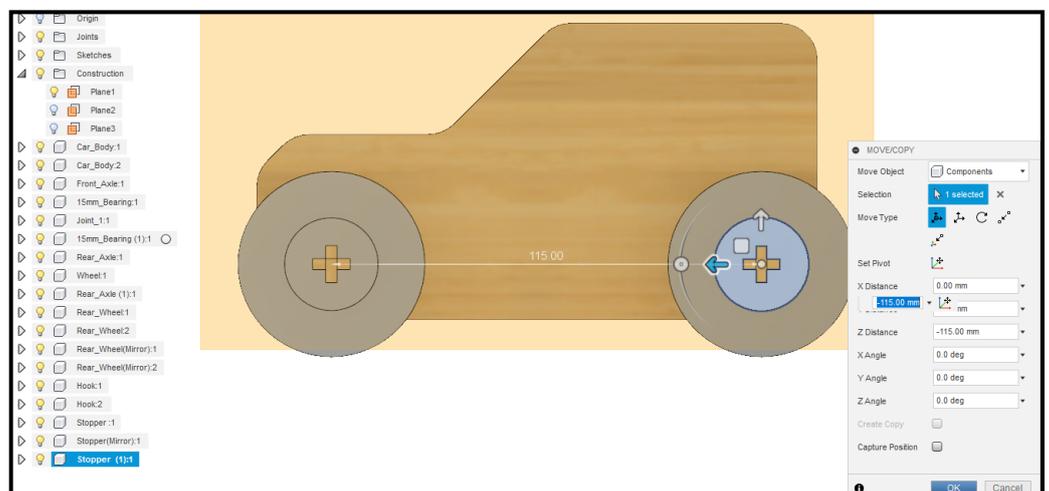


141. Under the create tab select **'Mirror'** and select the stopper as the object and the midplane shown as the mirror plane.

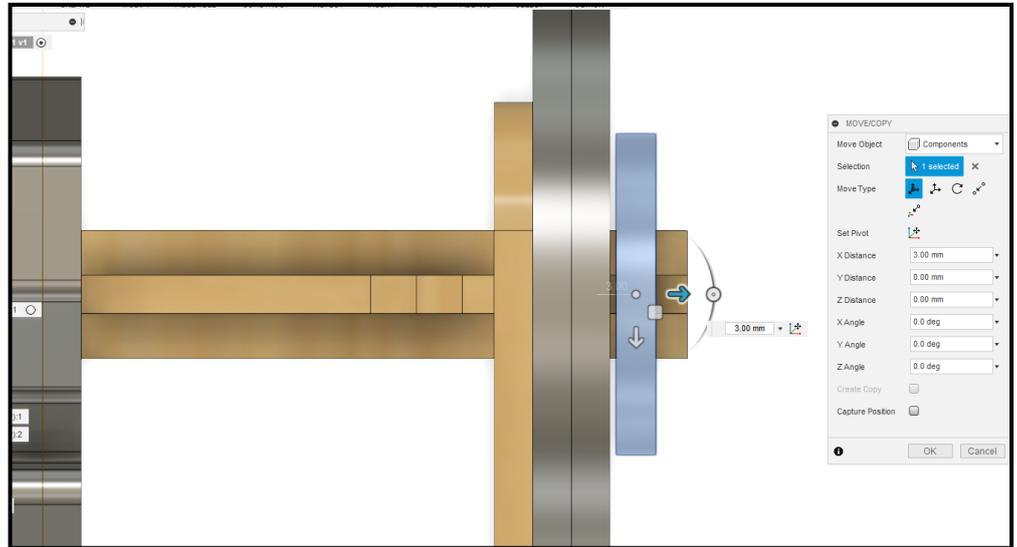
Right click on the stopper in the menu and select **'Copy'**. Then right click on the white area and select **'Paste'**.



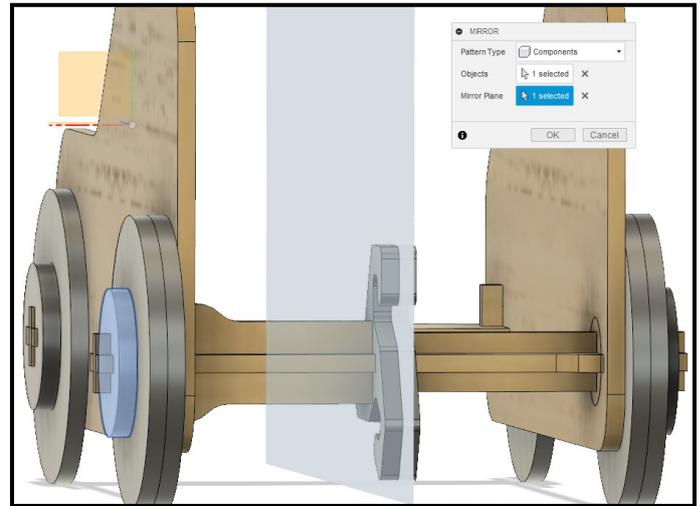
142. Now set the Z Distance of this new copy to **'-115mm'**.



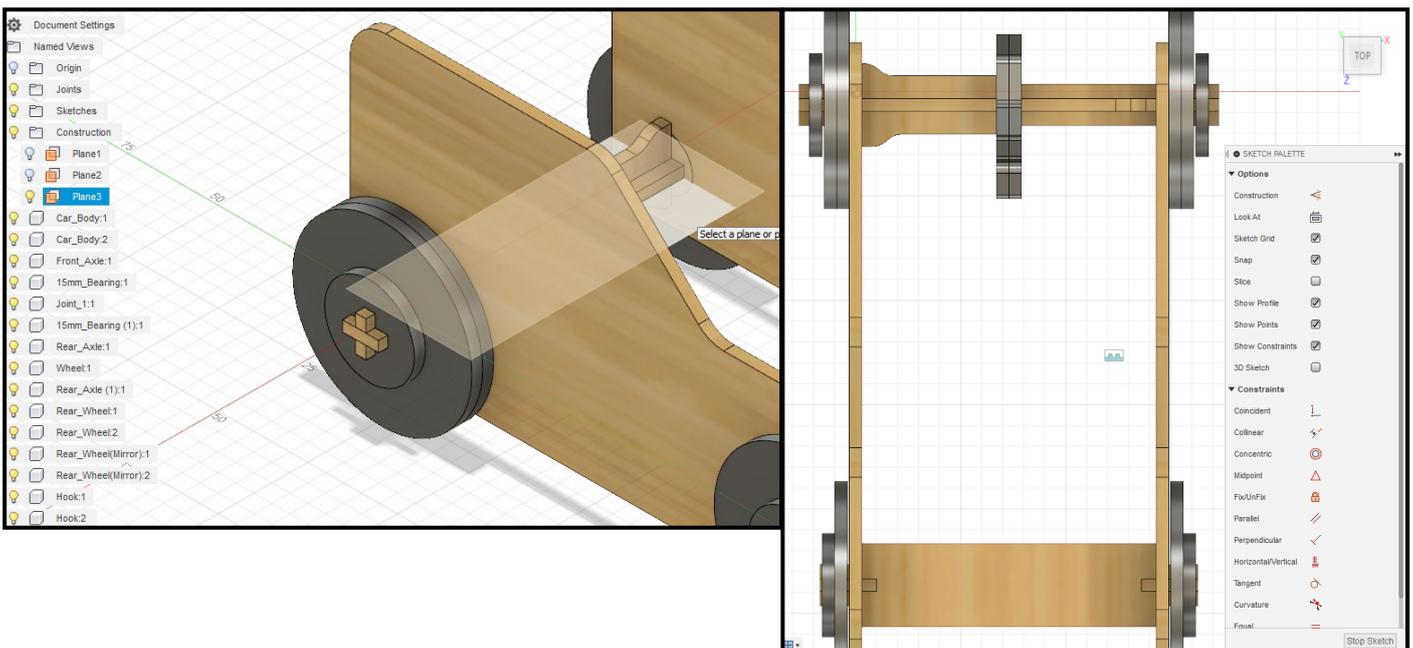
143. Press 'M' on the keyboard and select 'Components' as the move object. Select the copied stopper that you just moved as the selection. Set the X Distance to 3mm.



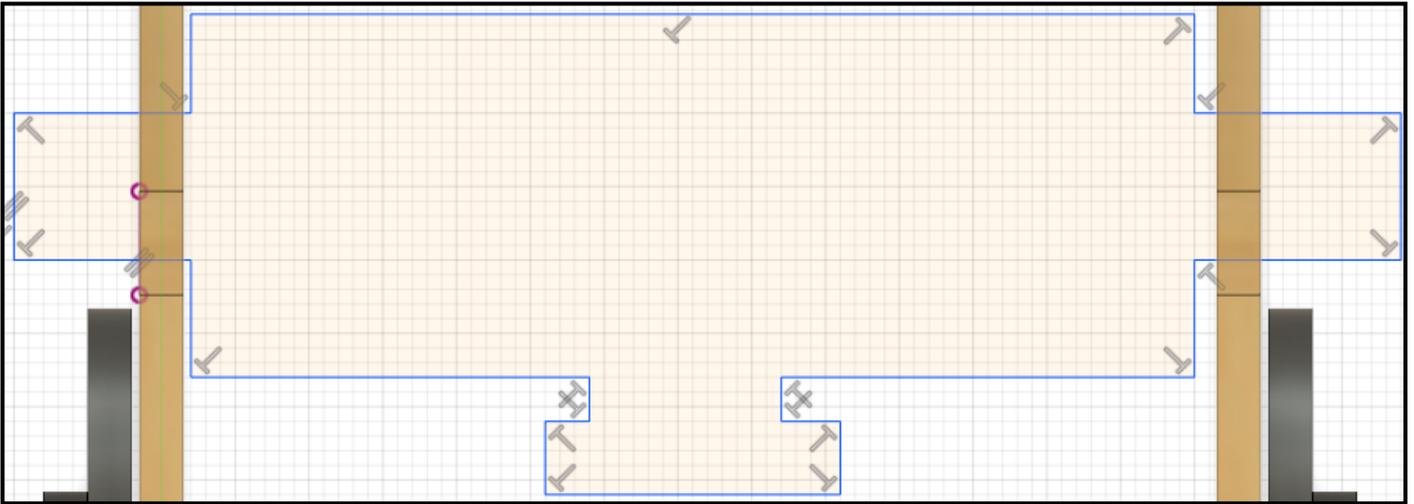
144. Under the create tab select 'Mirror', the pattern type set to 'Components', the rear stopper as the object and the midplane as the mirror plane.



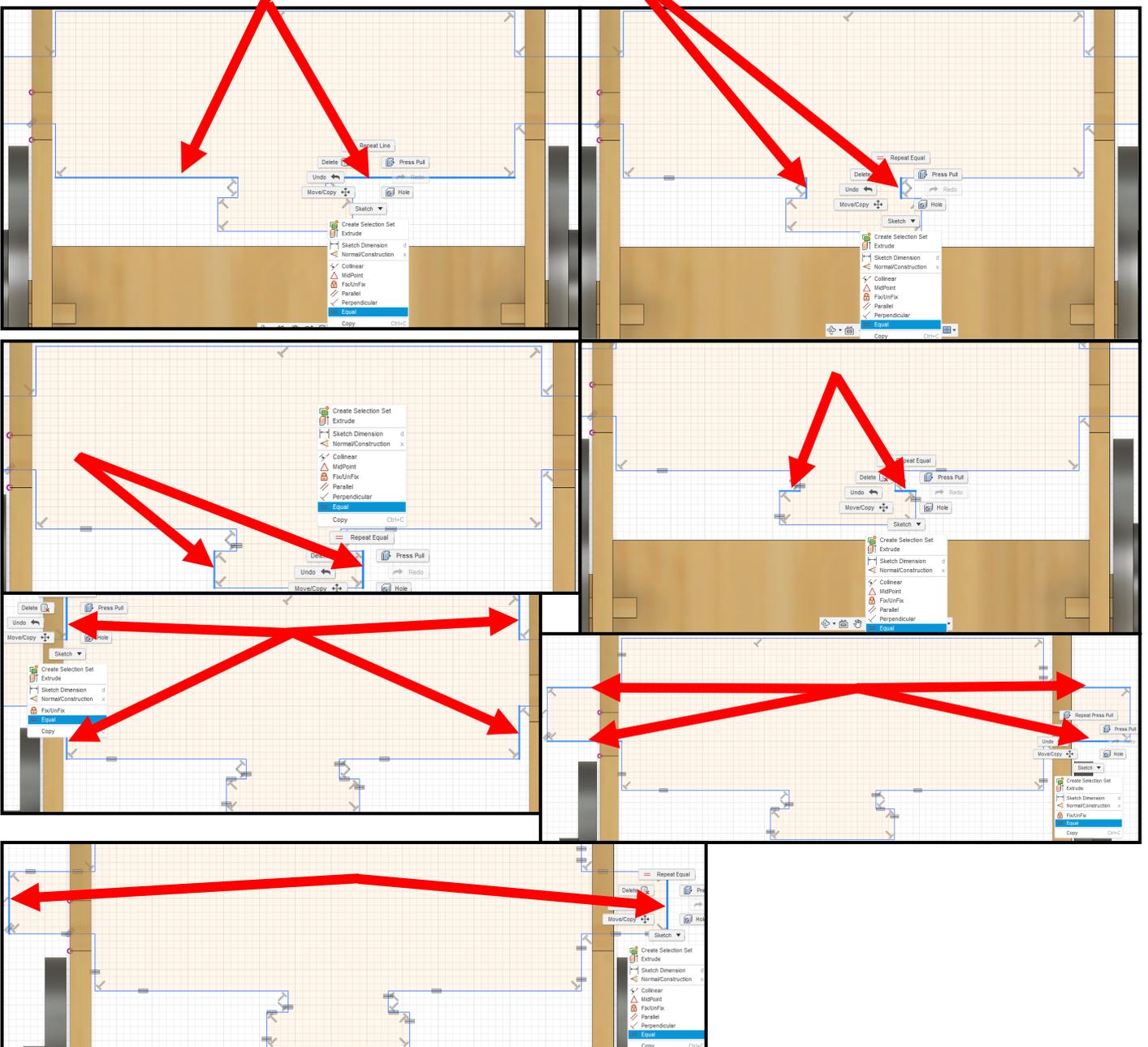
145. Start a sketch on the plane created earlier over the rear axles. You can find this plane under the construction tab on the left. Click the lightbulb next to the planes until it is found. Start from a top view as shown.



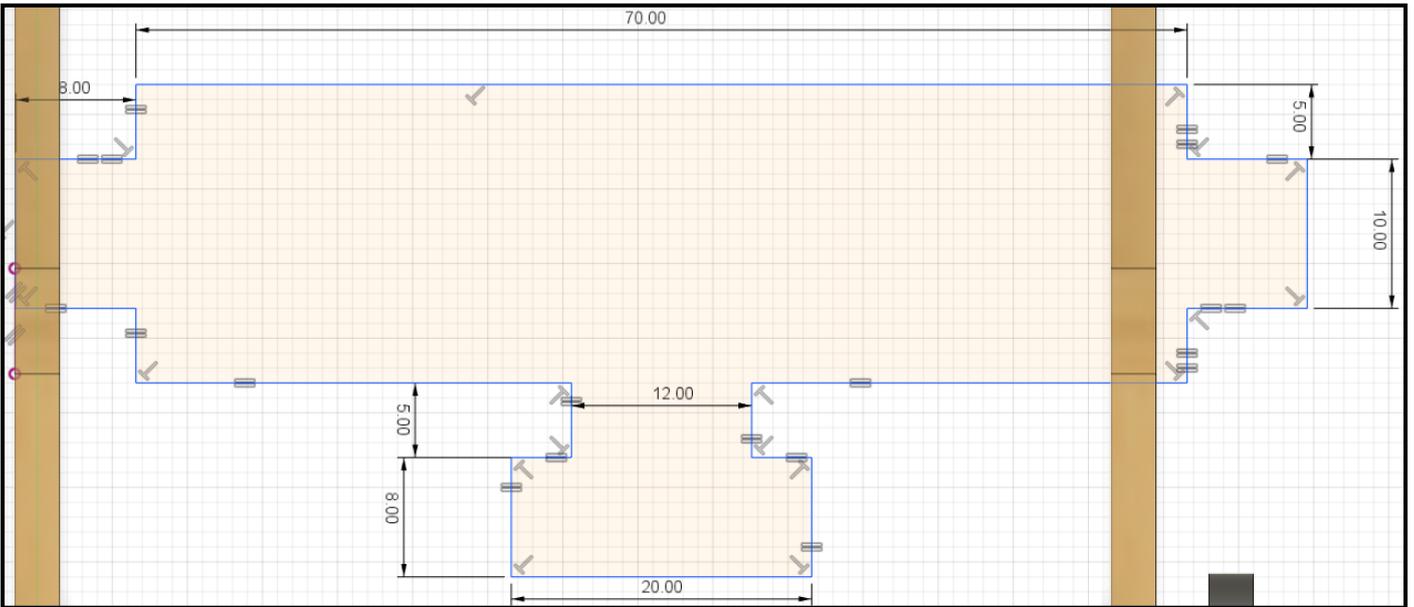
146. Start a **new sketch** and use the line tool to roughly draw this shape below just above the front wheels.



147. Hold down shift and select the highlighted blue edges shown. Right click and select '**Equal**'. Do this for each of the following 7 images.

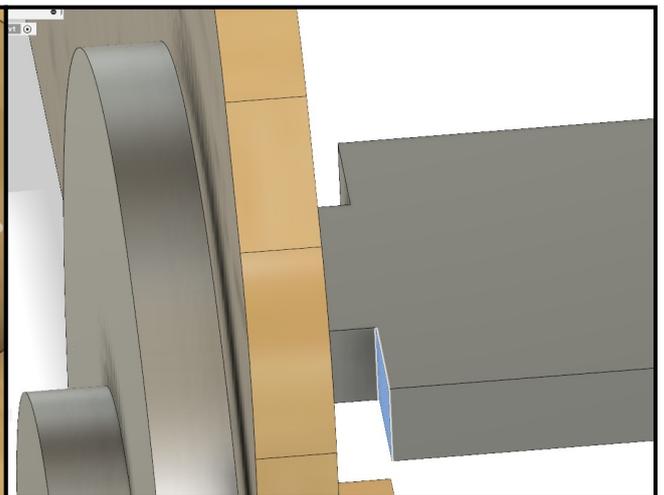
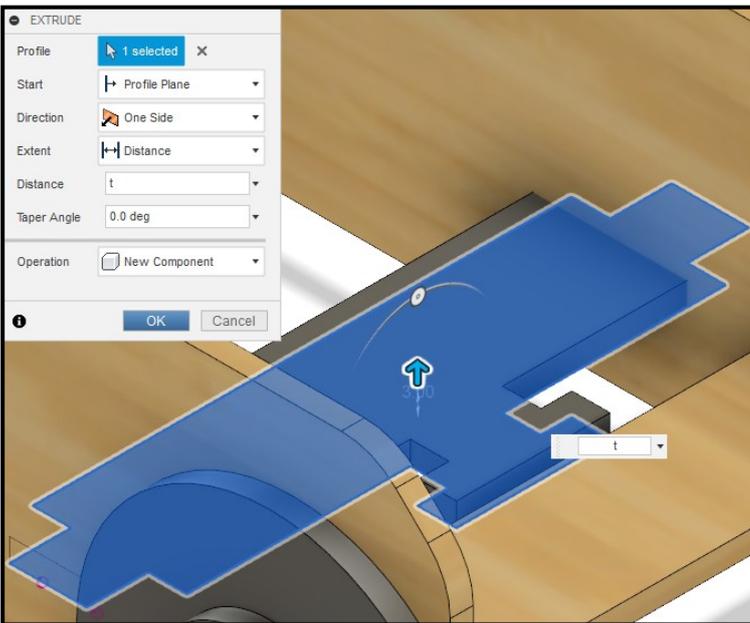


148. Add the following **dimensions** to the sketch.

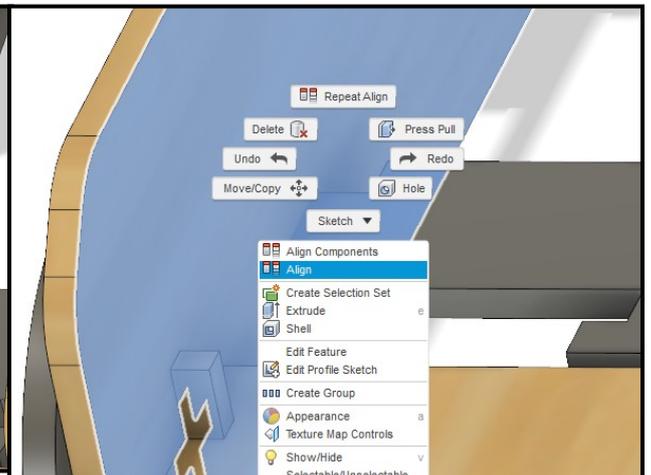
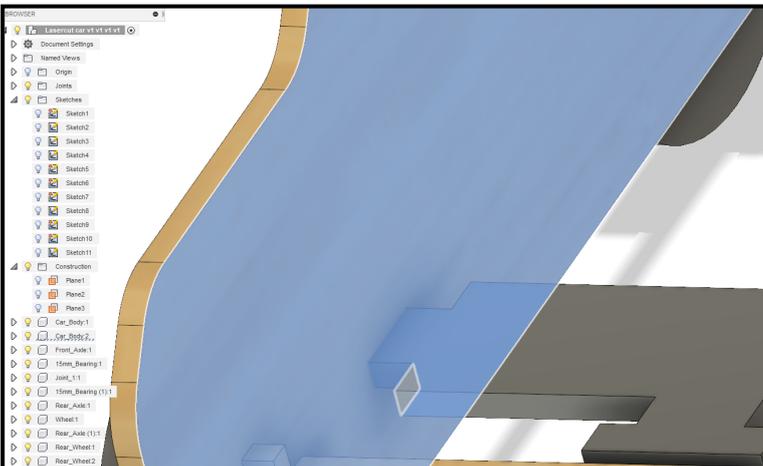


149. Extrude this new shape. Set the distance to 't' and make the operation 'New Component'.

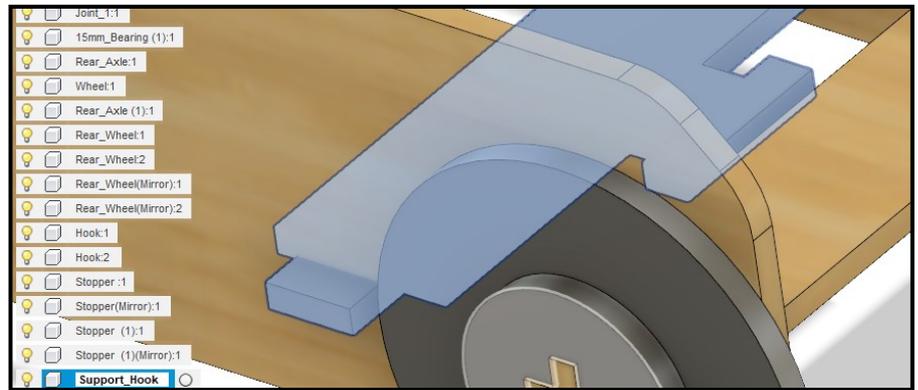
150. Select the face shown to the left of this shape.



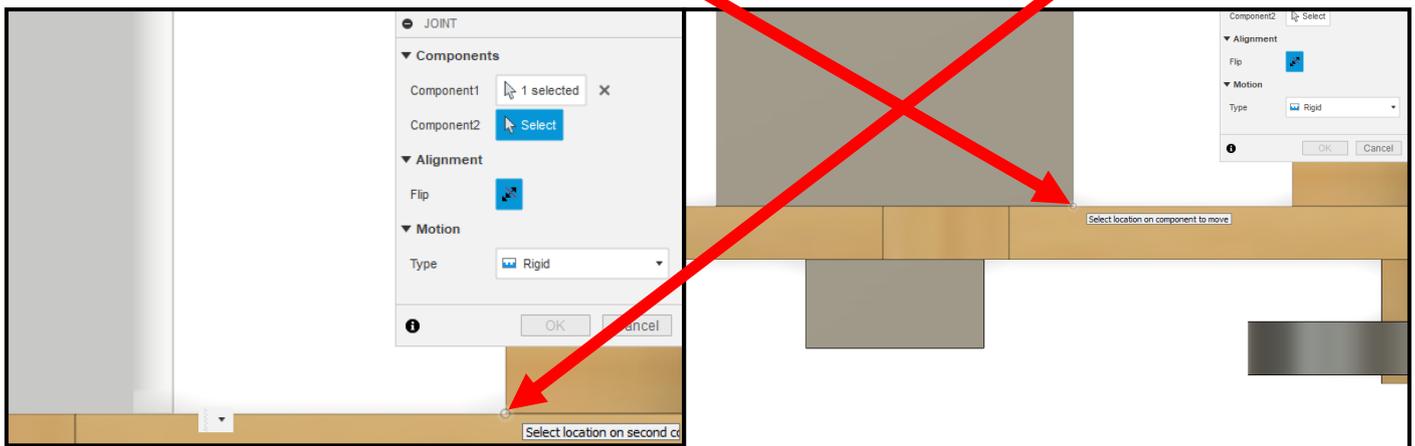
151. Hold down shift and select the inside face of the body shown. Right click and select 'Align'.



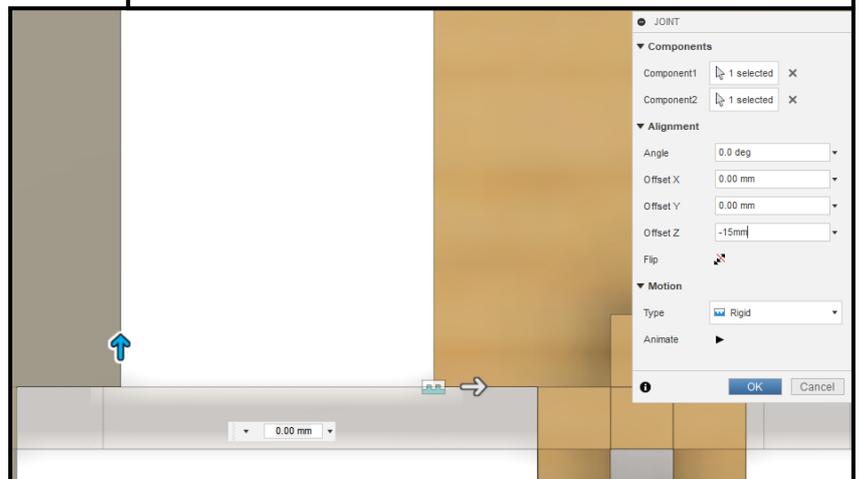
152. Re-name this component 'Support_Hook'.



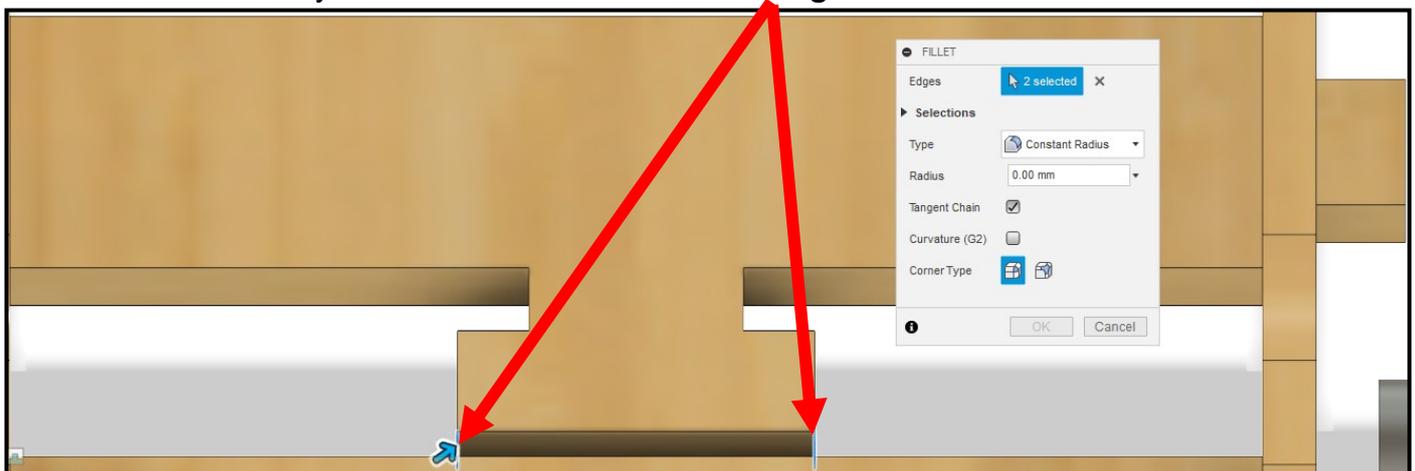
153. Under the 'Assemble' tab select 'Joint'. Select the point on the left as **Component 1**, and select the point on the right as **Component 2**.



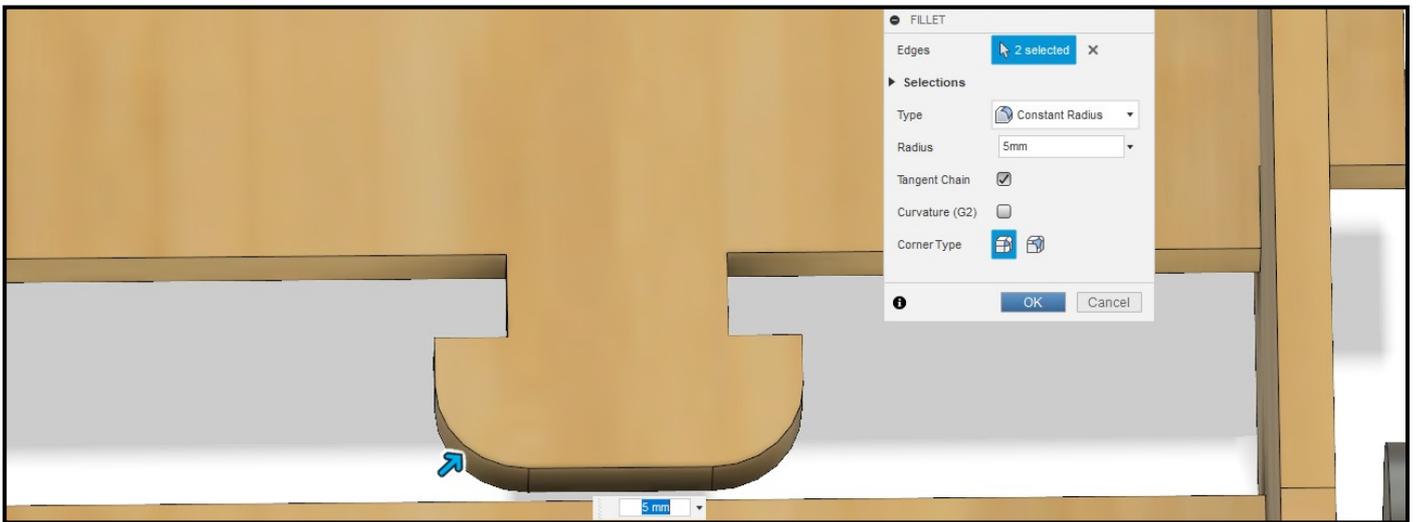
154. Set the **Offset Z** value to **-15mm**.



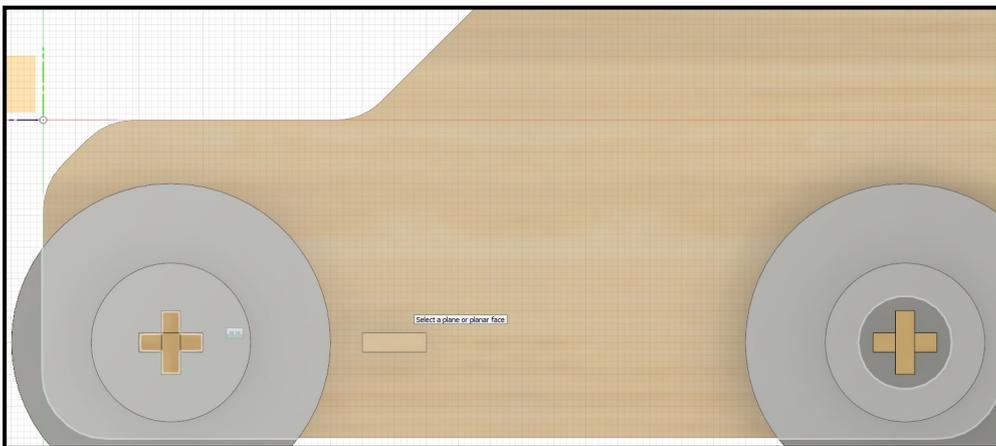
155. Under the modify tab select fillet. Select the **two edges** shown.



156. Now set the **radius** to **5mm**. This will create the rounded edge shown below.

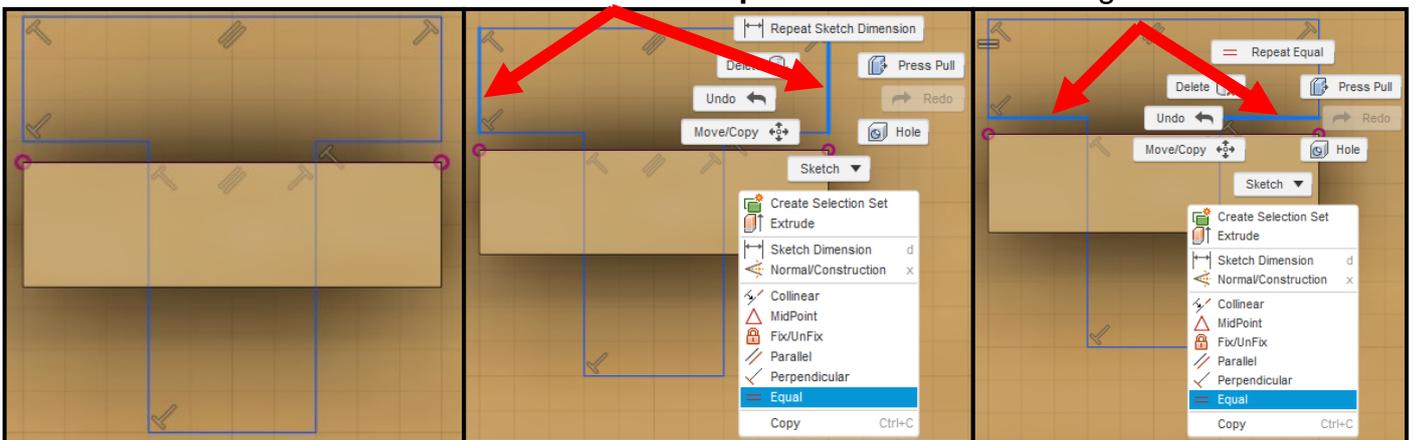


157. Start a **new sketch** on the side face of the car body.

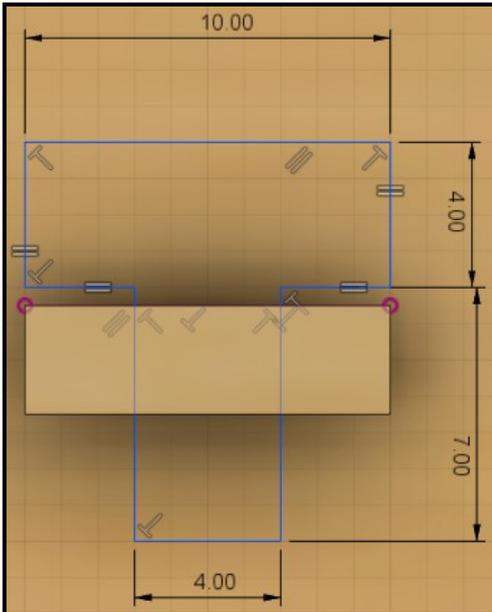


158. Using the **line tool** roughly draw the T shape shown below. This is at the rectangle where the support hook cuts the car body.

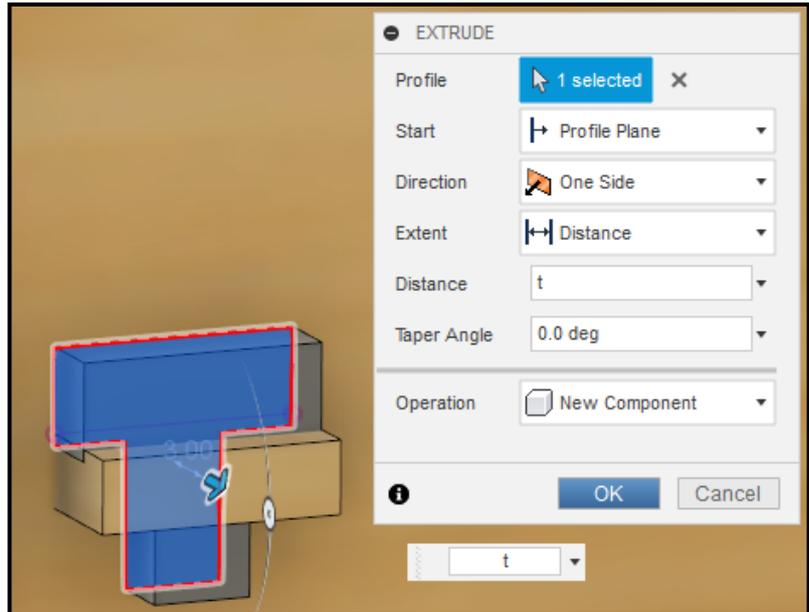
159. Hold down shift and select the lines shown in blue. Right click and select '**Equal**'. Do this for both images below.



159. Add the following **dimension** labels to the T shape.

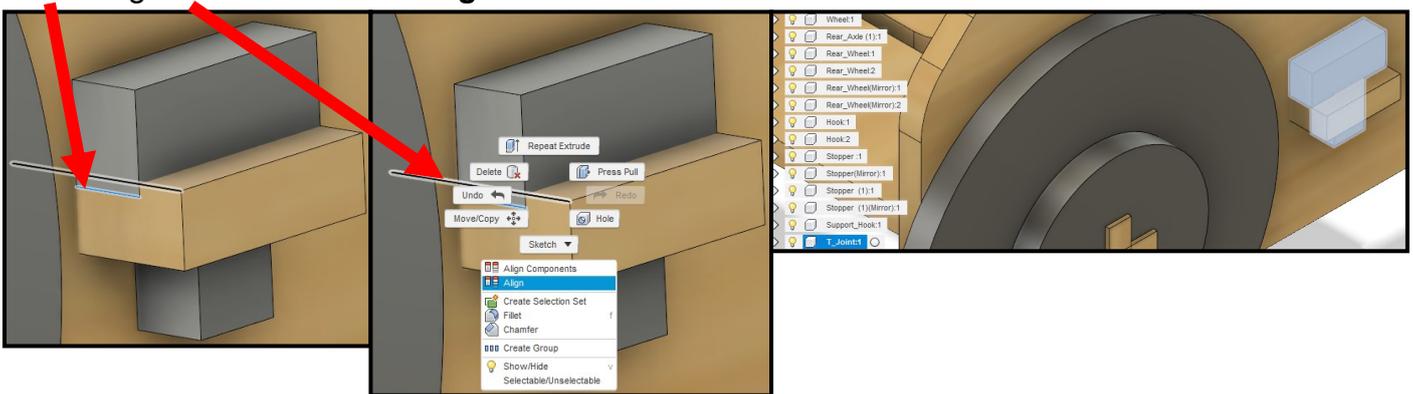


160. Extrude this T shape and set the distance to 't' and operation as 'New Component'.

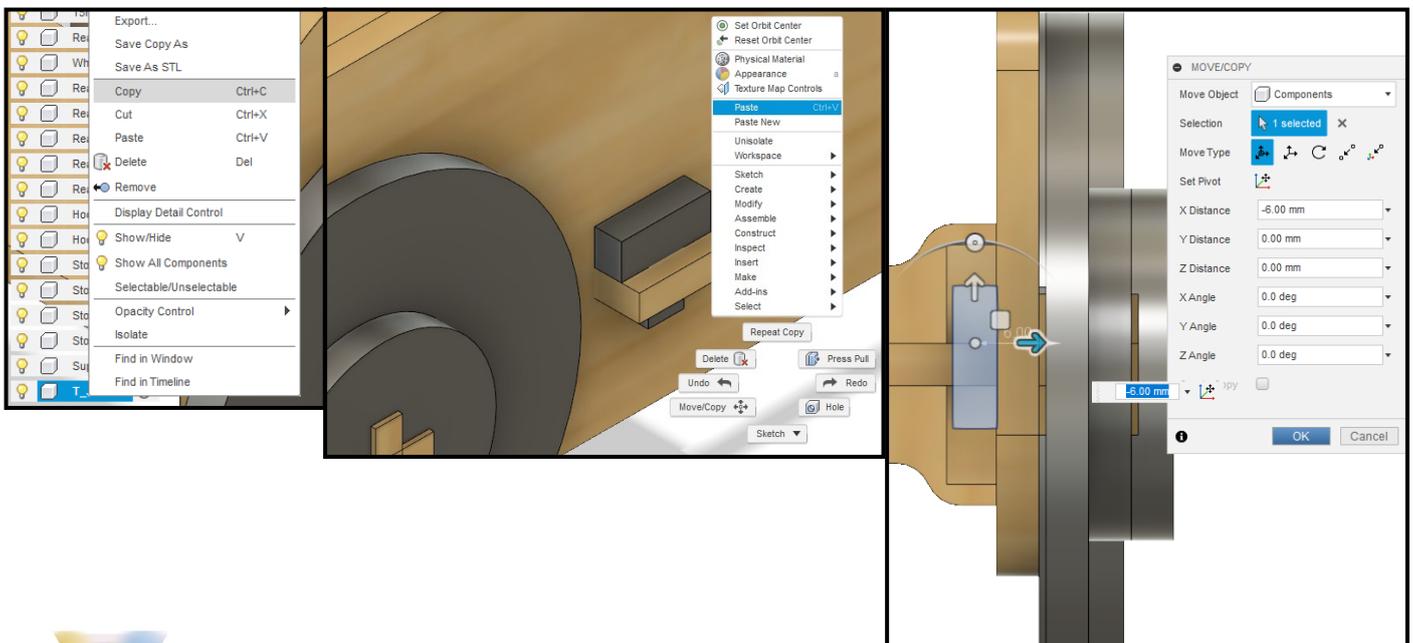


161. Select the edge of the T shown in blue, hold down shift and select the edge line from the support hook. Right click and select 'Align'.

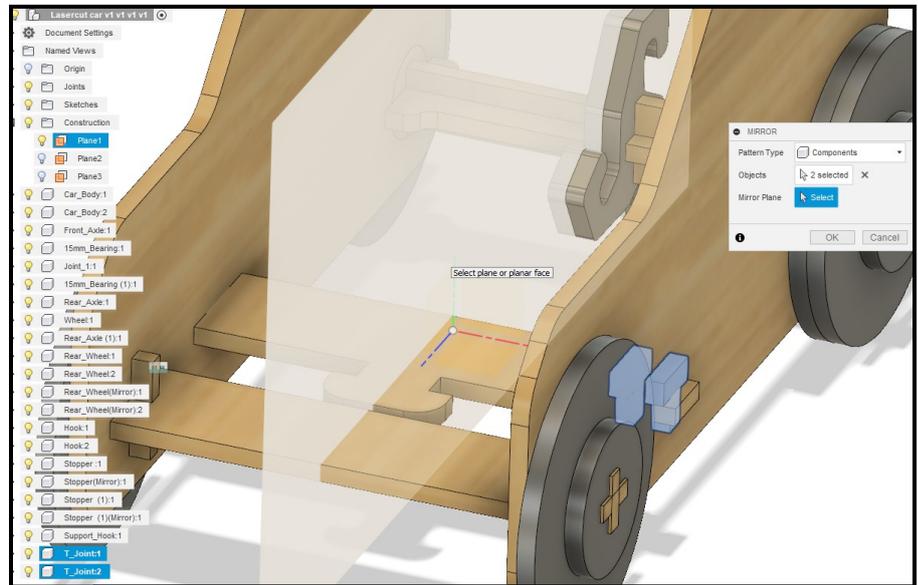
162. Re-name this component 'T_Joint'.



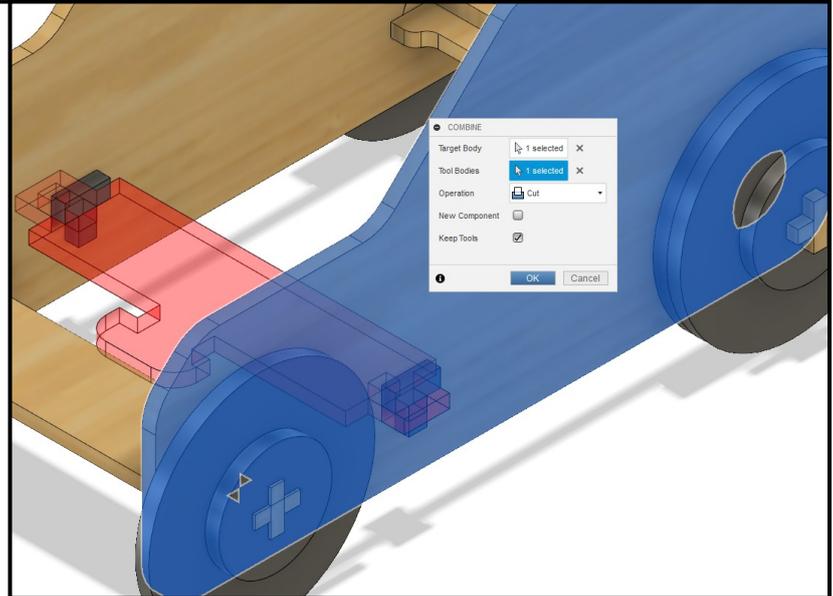
163. Right click on the T joint in the menu and select 'Copy'. Then right click on the white area and select 'Paste'. Set the X Distance to -6mm to move the copied T to the other side.



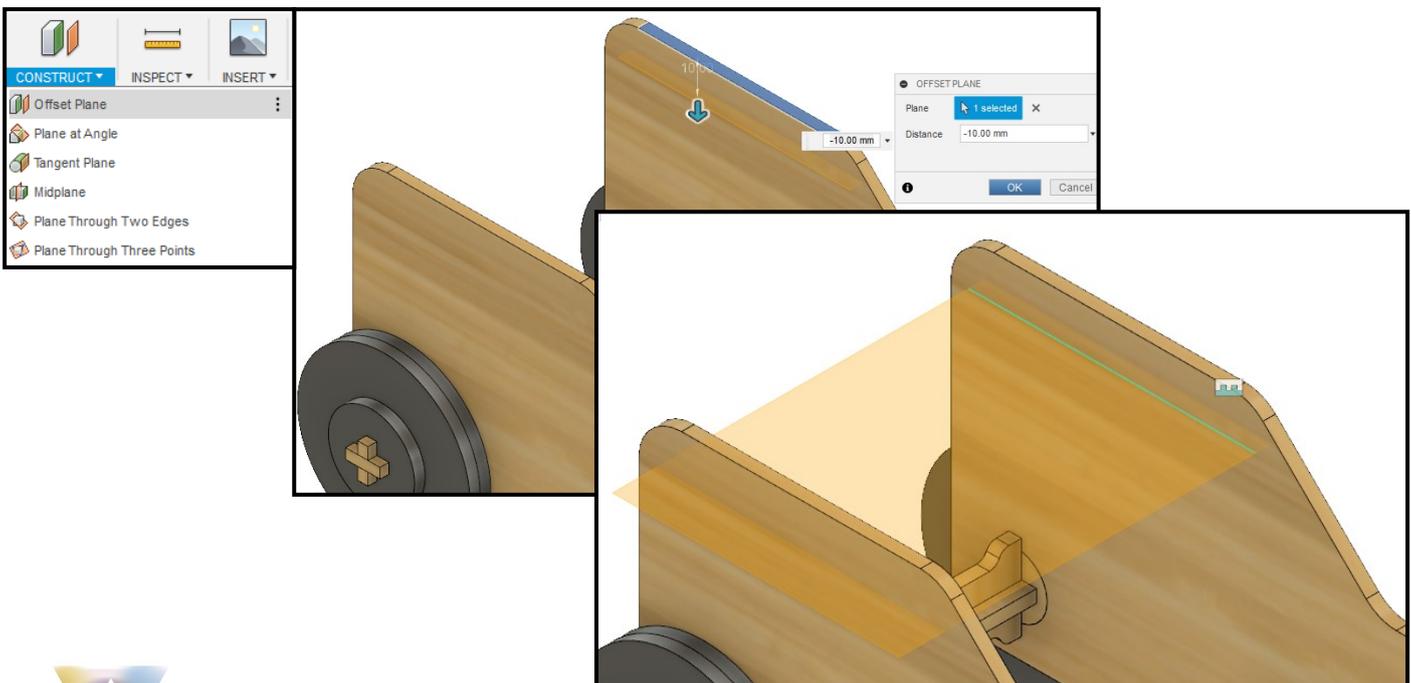
164. Under the **create** tab select '**Mirror**'. Select both T joints as the objects and the midplane used earlier as the mirror plane.



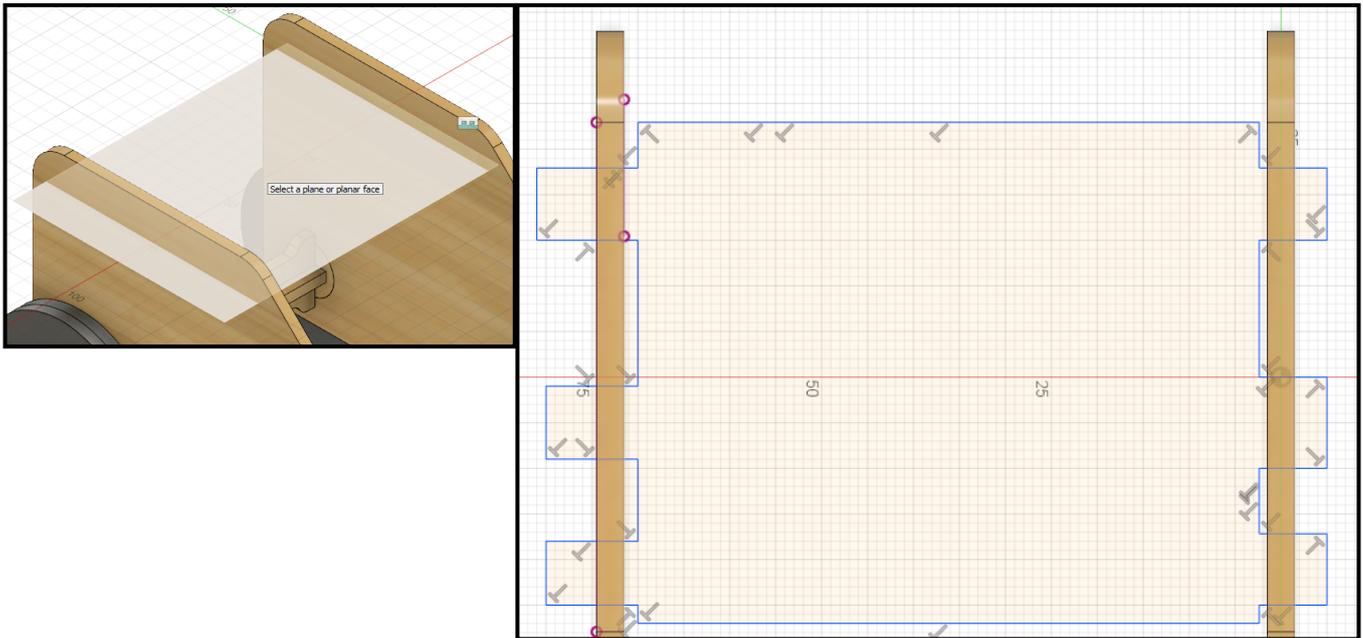
165. Select the **combine** tool and select the car body shown as the target body, and select the support hook as the tool body. Make the operation a cut and choose to keep tools.



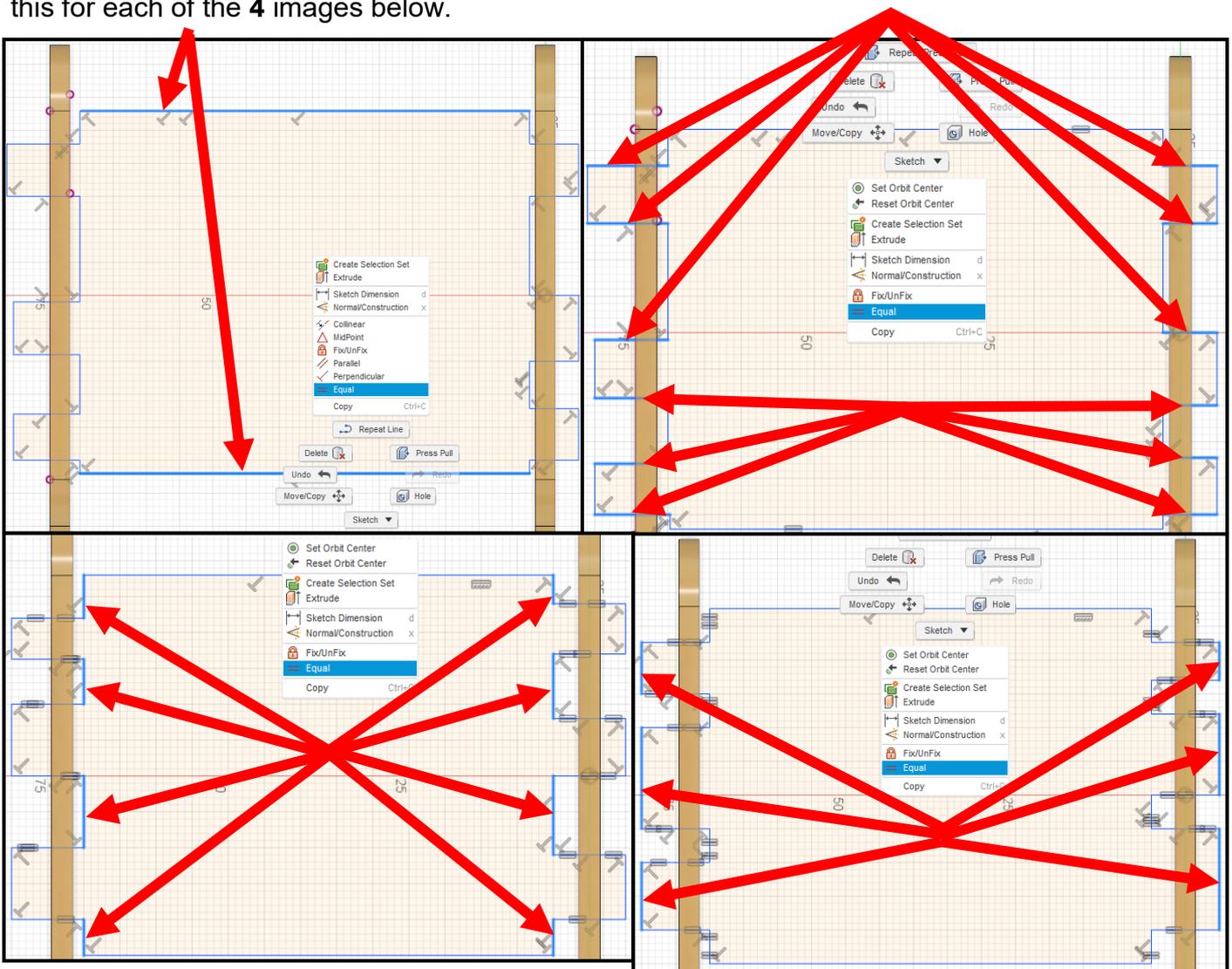
166. Under the **construct** tab select '**Offset Plane**'. Select the face shown at the top of the car body as the starting plane. Then set the **distance** to '**-10mm**'. Drag out the edges of the new plane to make it larger and extend it across the width of the car.



167. Start a **new sketch** and select the new plane as the work plane. Using the lightbulbs in the menu hide the rear wheels, axles etc. This is just to make the sketching area less cluttered. From the **top view**, use the **line** tool to roughly draw the same sketch below.

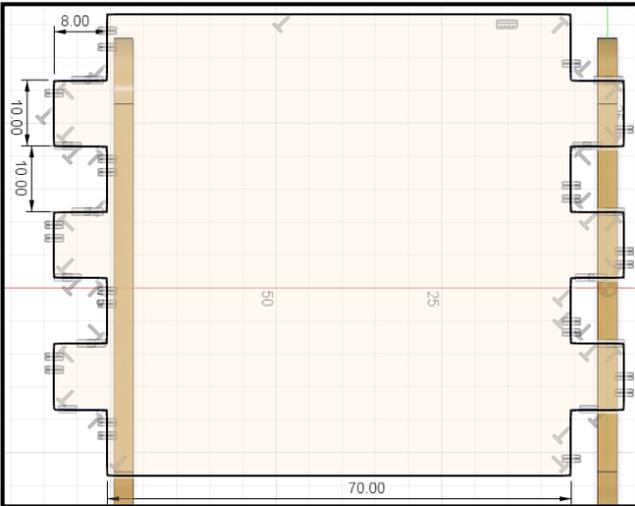


168. Hold down shift and select the lines highlighted in blue. Right click and select 'Equal'. Do this for each of the 4 images below.

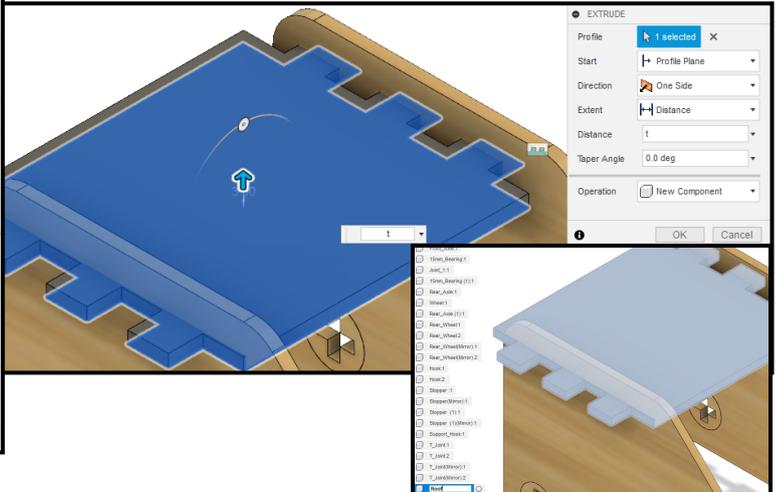


169. Add the **dimensions** to this new sketch 170. Select this sketch to extrude. Set the distance to 't' and make the operation 'New Component'.

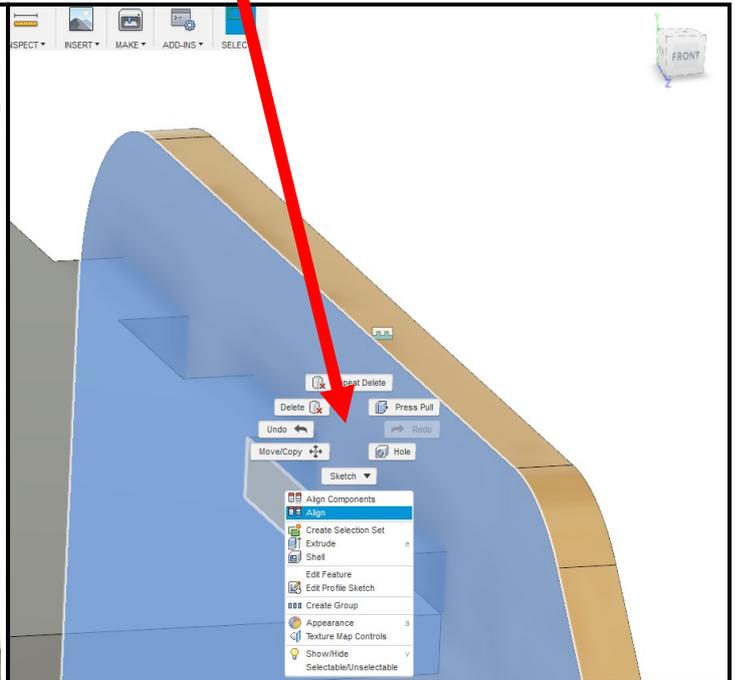
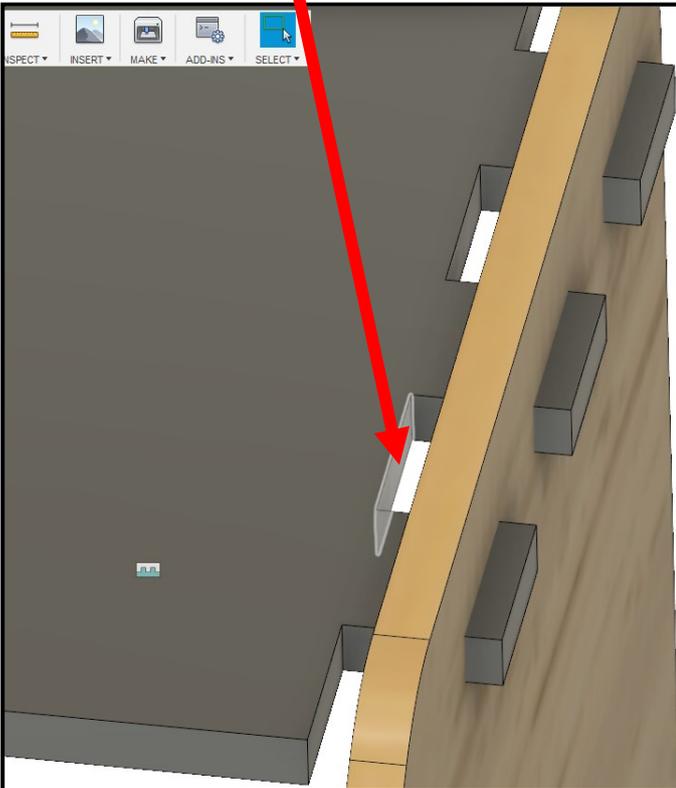
170. Select this sketch to extrude. Set the distance to 't' and make the operation 'New Component'.



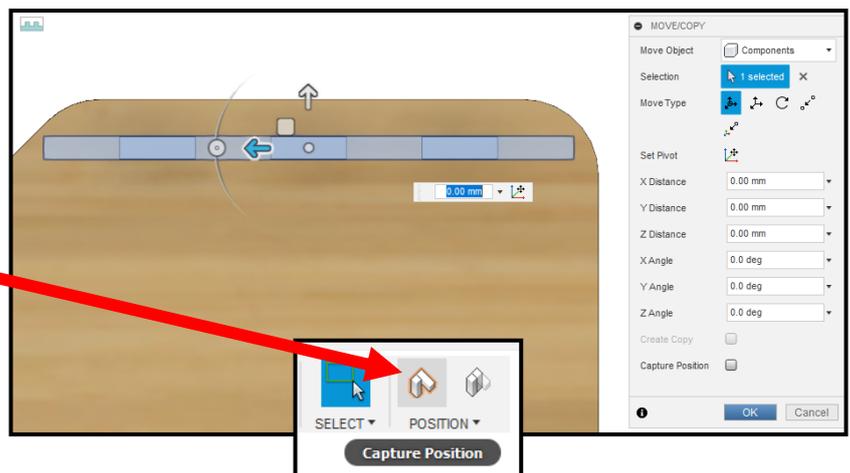
Rename this component 'Roof'.



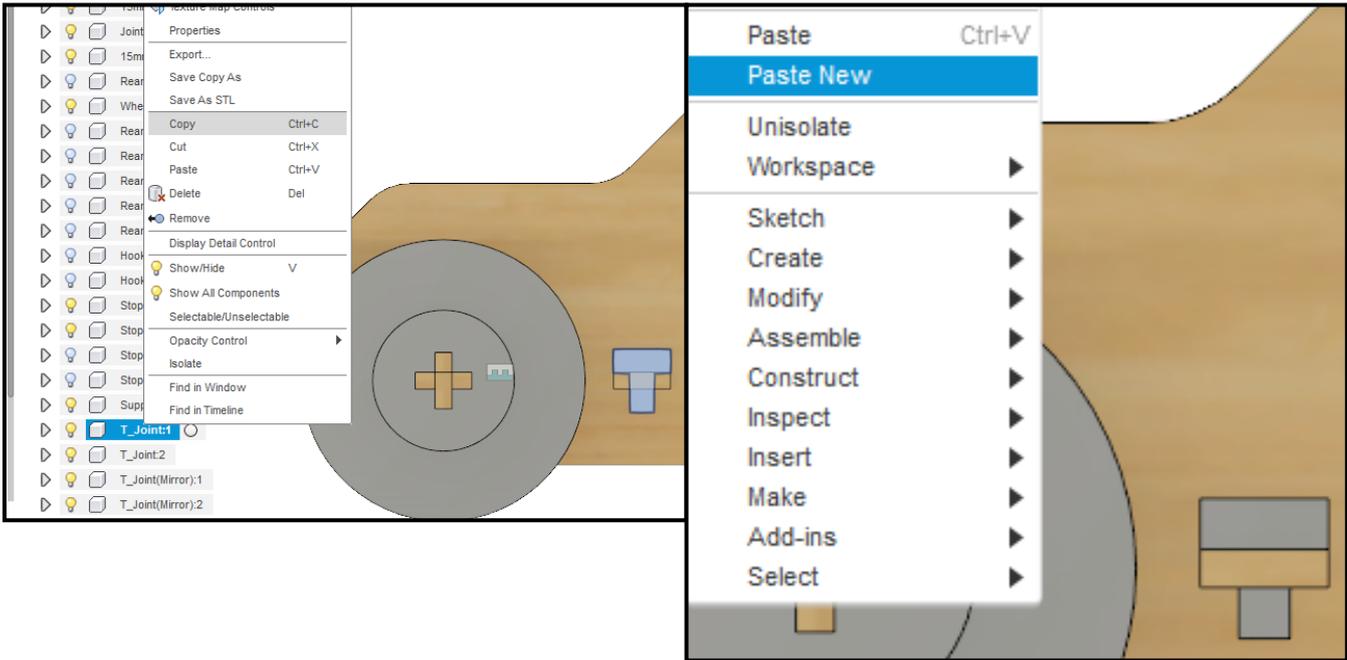
171. Select the face shown on the roof. Hold down shift and select the inner car body facing the previously select face. Right click and select 'Align'.



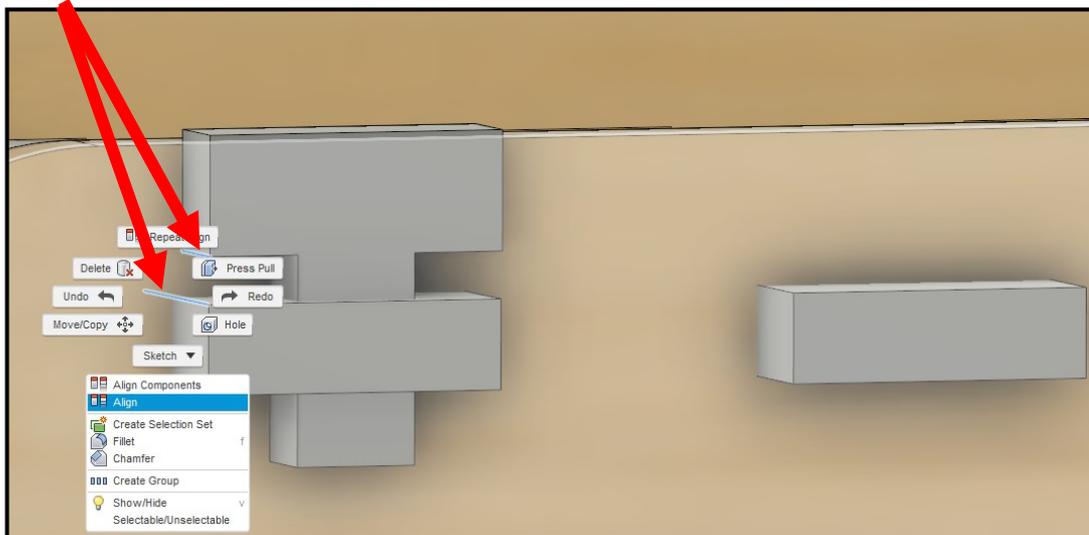
172. Press 'M' on the keyboard and select the roof as the 'selection' component. Use the **two arrows** to move the roof roughly into the position shown. Click on **capture position** to fix it in place.



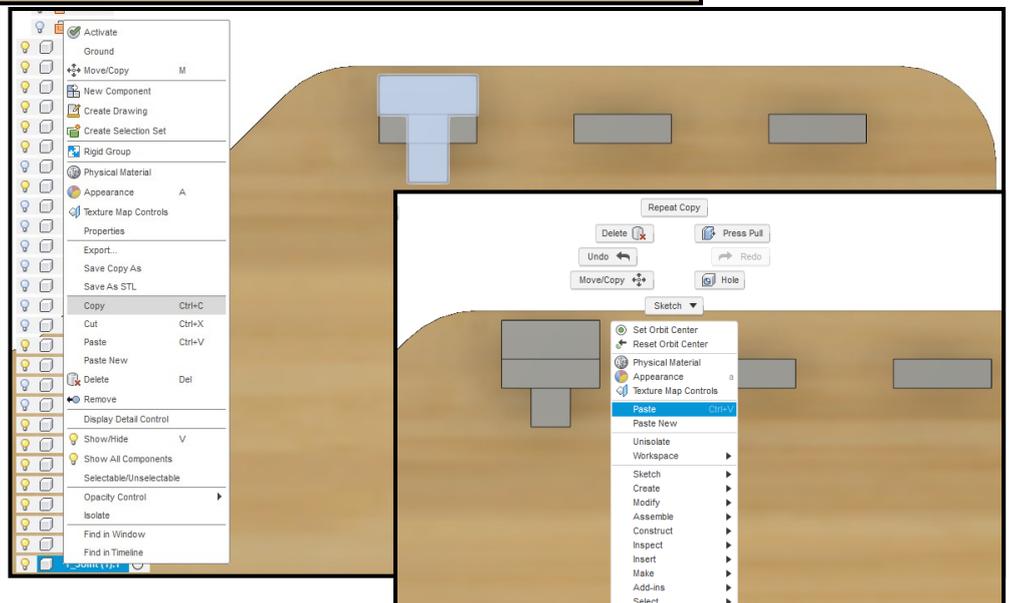
173. Right click on the T joint in the menu, **copy** it and **paste a new one**. Move the T joint using the same arrows as before to roughly place this new T joint onto the first of the three rectangle nodes of the roof.



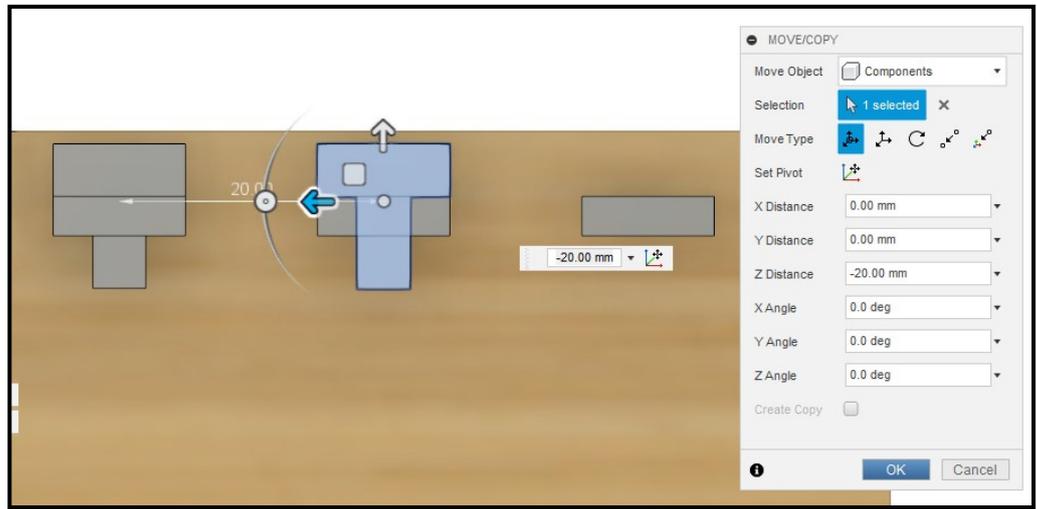
174. Hold down shift and select the **edge line** on the T joint, then select the **edge line** on the roof. Right click and select '**Align**'.



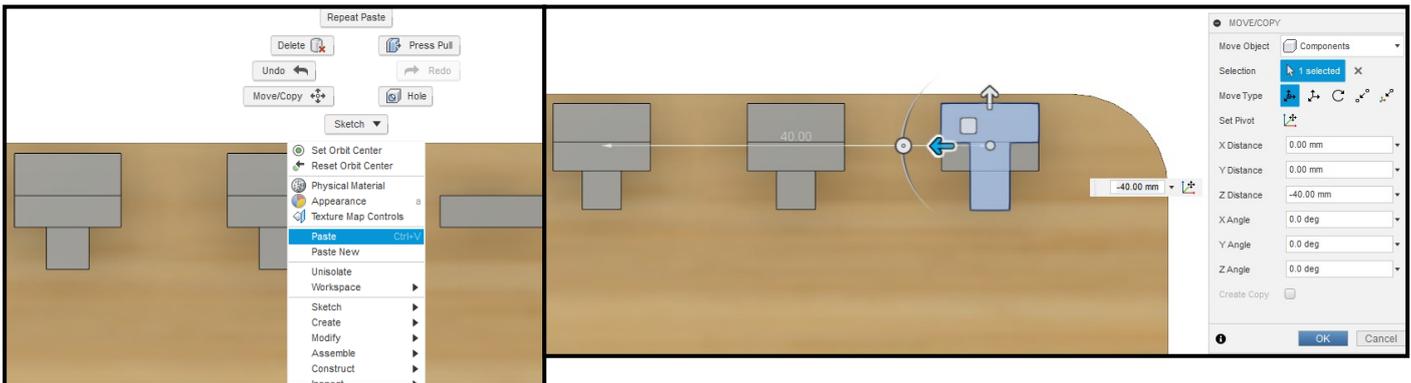
175. Now that the T Joint is in place, right click on it in the menu and select **copy**. Then right click on the white area and select **paste**.



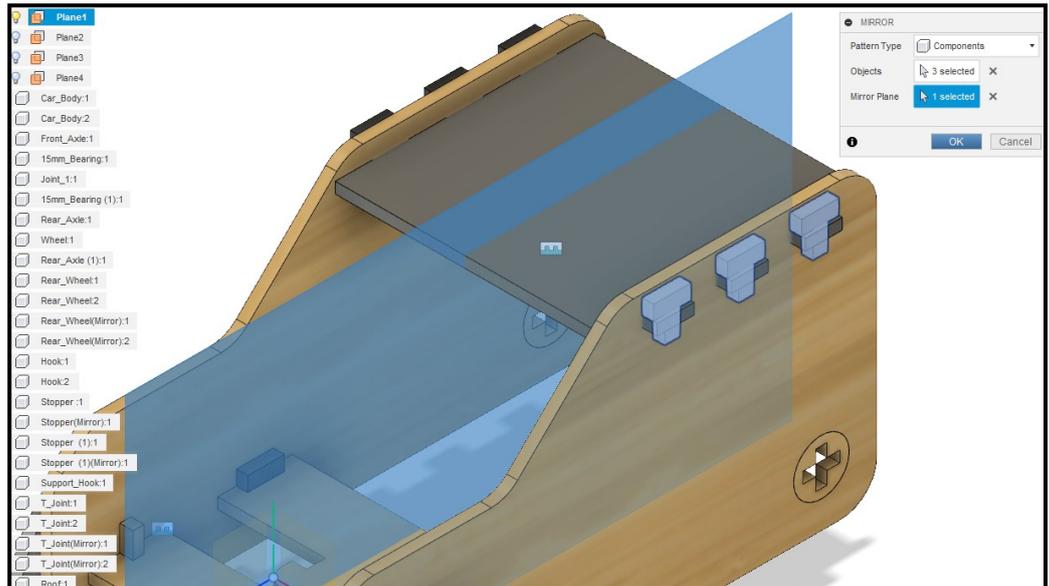
176. Set the **Z Distance** to '**-20mm**' for this copied T joint. This should move it into the position shown.



177. **Paste** another T joint, then set the **Z Distance** as '**-40mm**'.

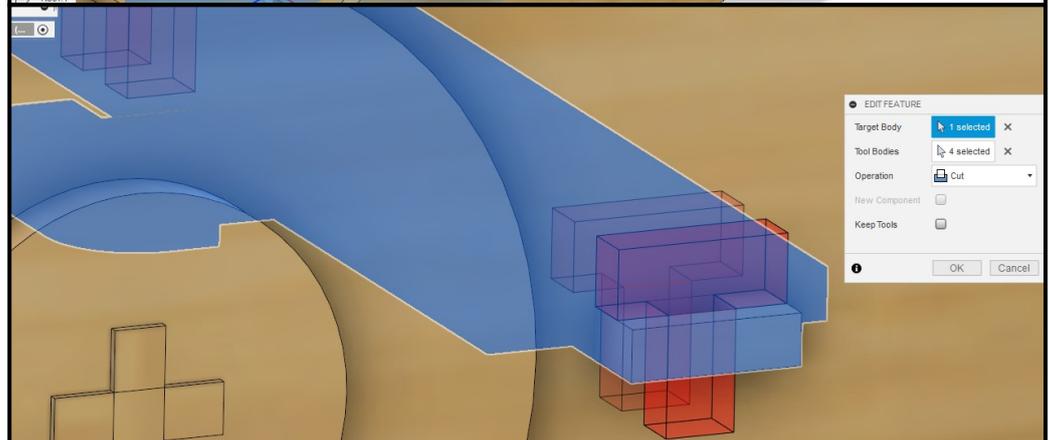


178. Under the **create** tab select the '**Mirror**' tool. Select the **three T Joints** shown and the midplane as the mirror plane.

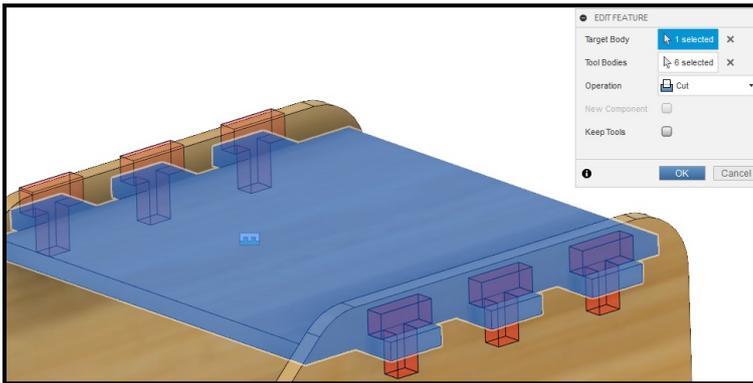


179. Select the **combine** tool, make the target body the support hook and make the **4 T joints** on either side the tool bodies.

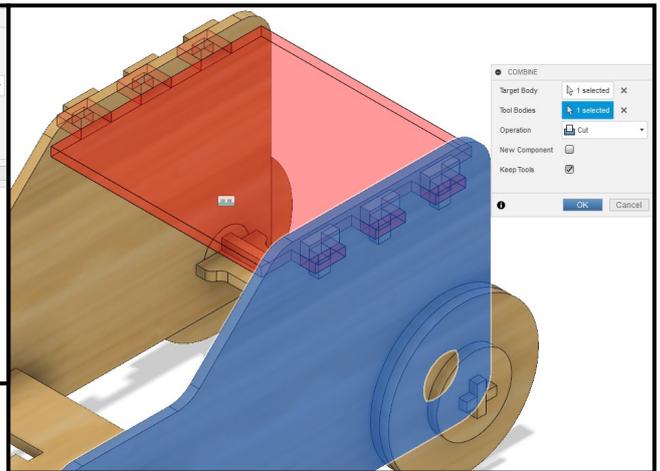
Make the operation a '**Cut**' and **uncheck 'Keep Tools'**.



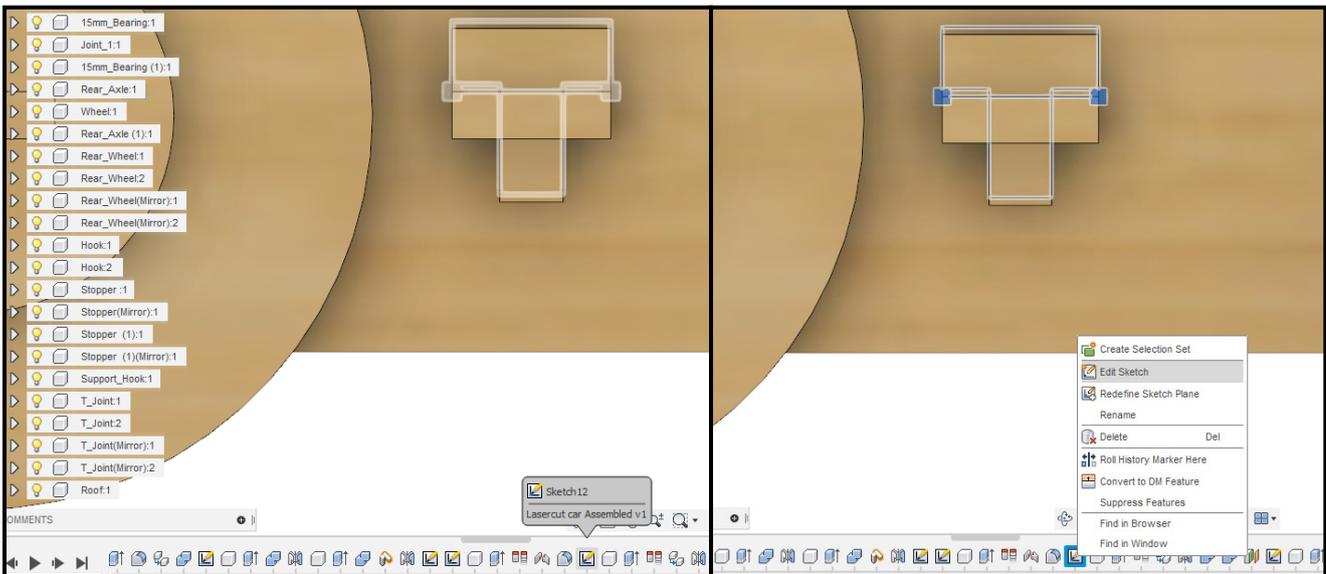
180. Use the **combine** tool again, select the roof as the target body, and the **6 T joints** as the tool bodies. Set the operation to 'Cut' and **uncheck** the box for 'Keep Tools'.



181. Using the **combine** tool select the car body as the **target body**, and the roof as the **tool body**. Make the operation a 'Cut' and tick the box for 'Keep Tools'.

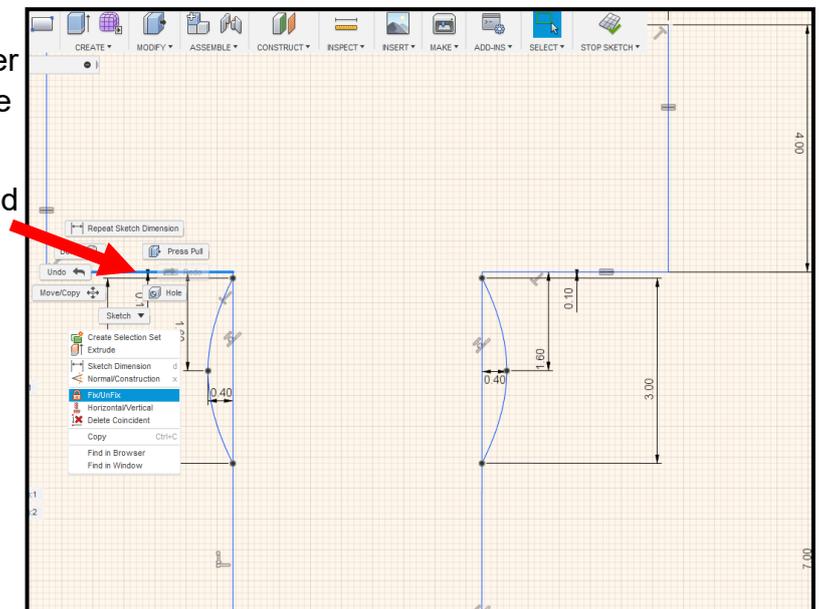


182. If a component needs to be fixed or amended, you can use the **timeline** at the bottom to edit or change components. In the timeline find the **original sketch for the T joint**. Once it is found right click on the sketch symbol and select 'Edit Sketch'.

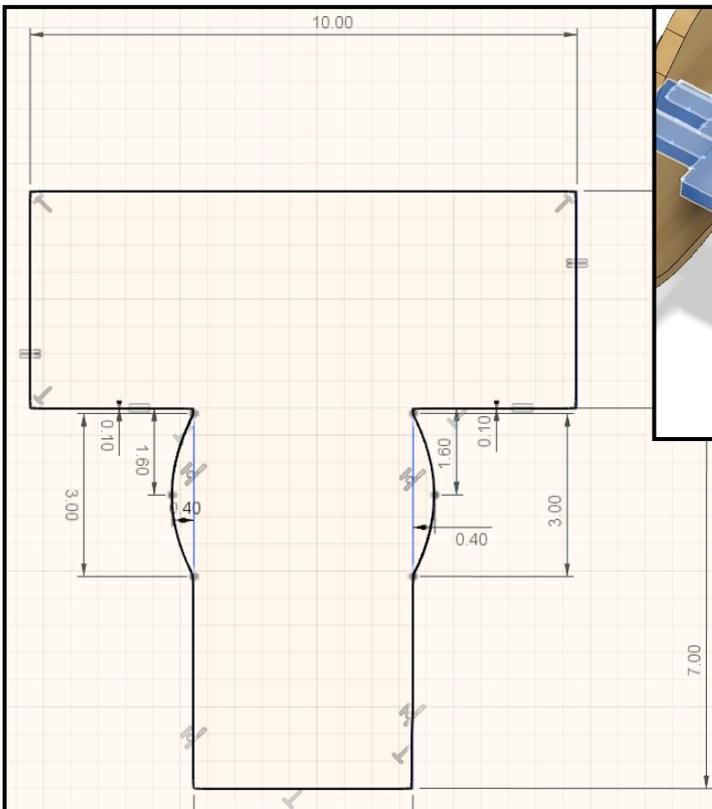


183. Use the same spline tool used earlier to draw the **two** splines on the stem of the T shown on the right.

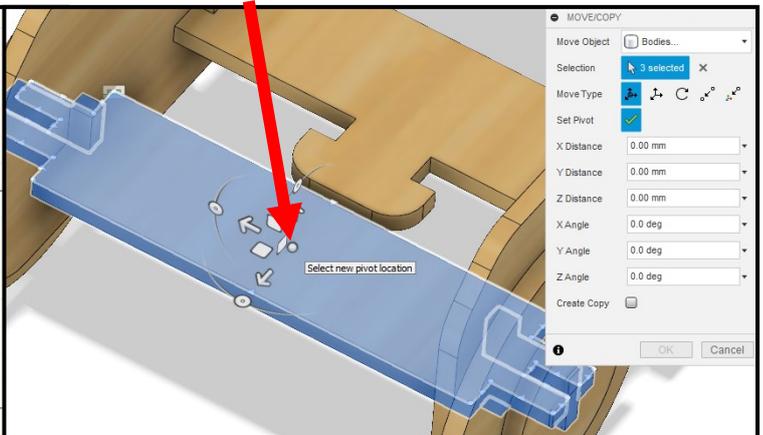
Right click on the blue line highlighted and select 'Fix/Unfix'. You can do this for any line that moves parts of the sketch when adding dimensions. This will fix the line in place.



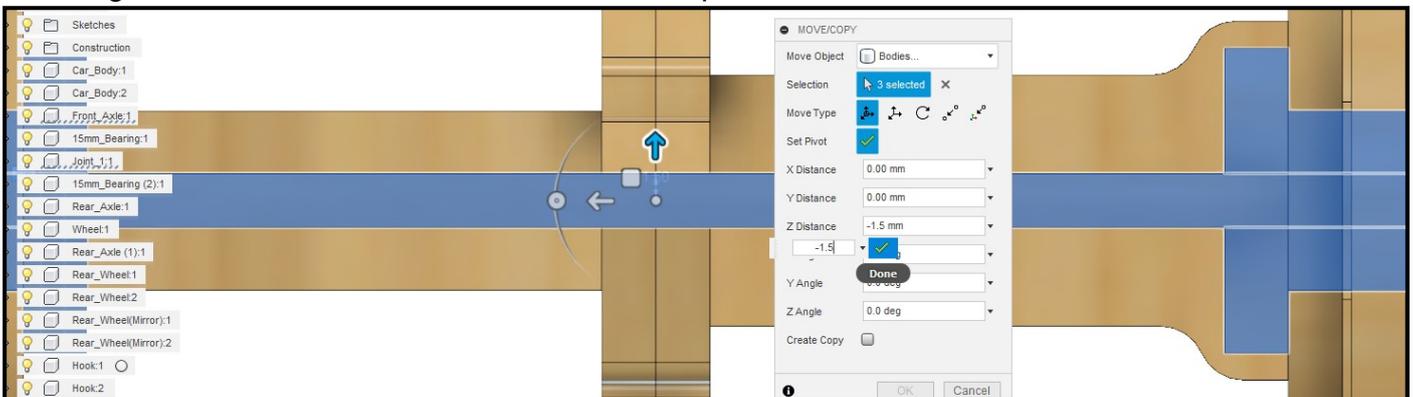
184. Add the following **dimensions** to the splines that were just drawn. Now repeat steps **160 - 164** and steps **173 - 178** for this **new T joint**. Remember to include the spline area when extruding.



185. Activate '**Move/copy**' by typing '**M**' on the keyboard. Select the front axle and the **two** joints either side as the '**Selection**'. Click the symbol next to '**Set Pivot**', you can now click and drag the pivot from the circular node to the centre of the front axle.



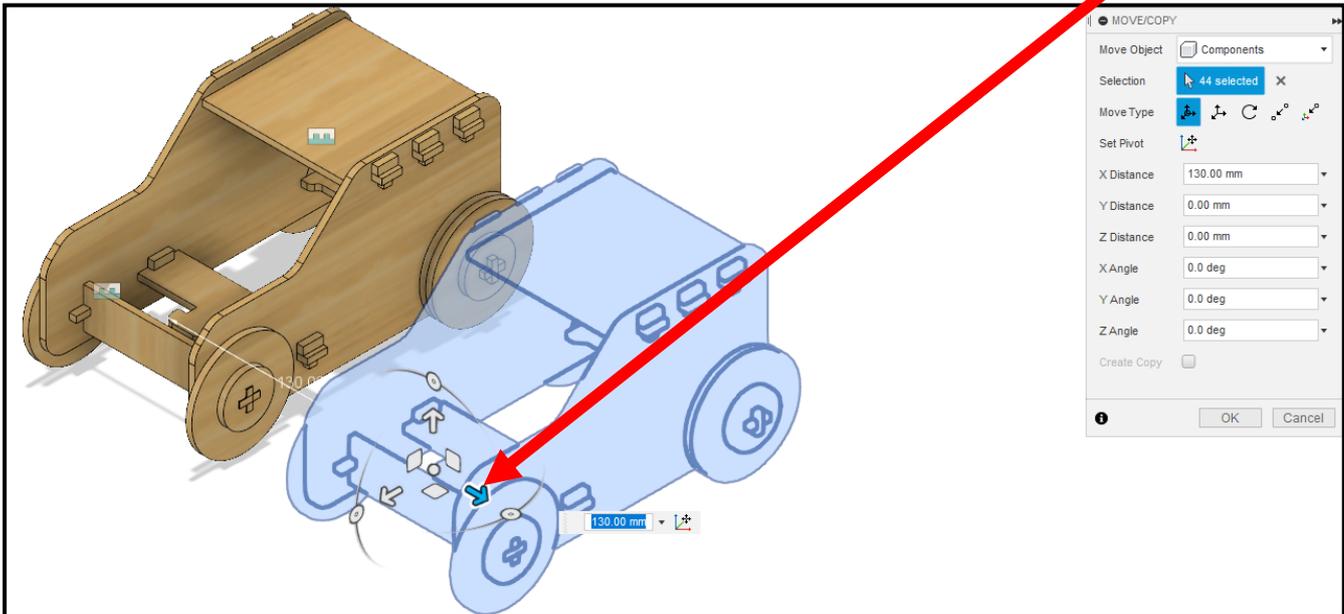
186. From the front, make sure the node is placed in the centre of the front axle. Do this by making the **Z Distance** '**-1.5mm**'. To secure the pivot click on the tick next to '**Set Pivot**'.



187. From the side view, set the **X Angle** to **90 degrees**. This should rotate the axle and the joints together.

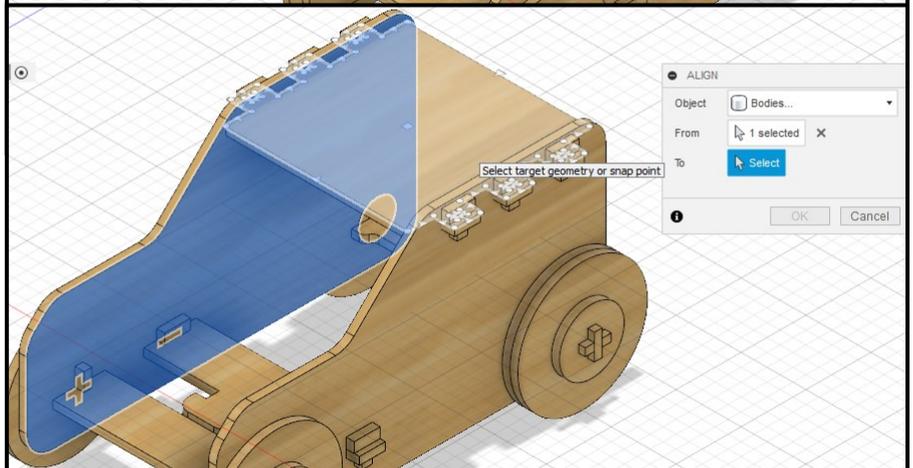
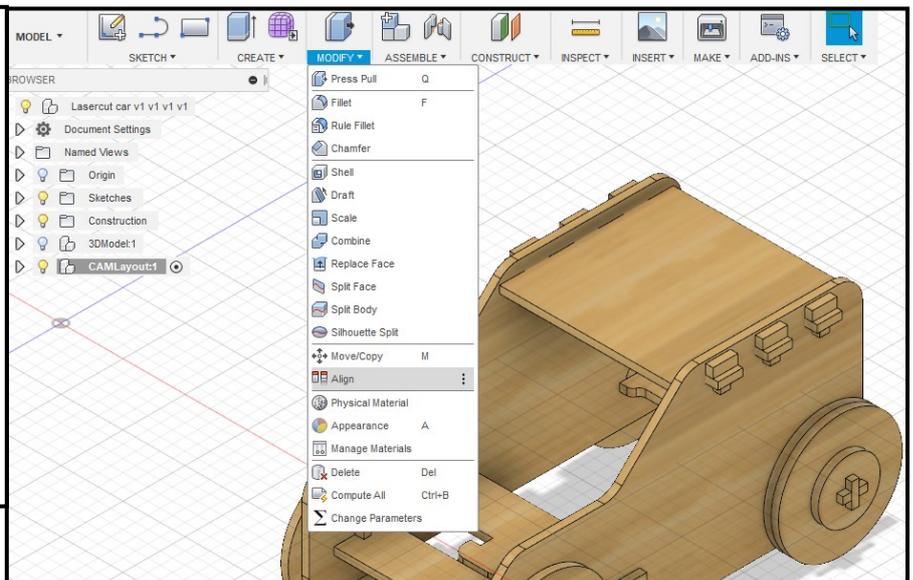
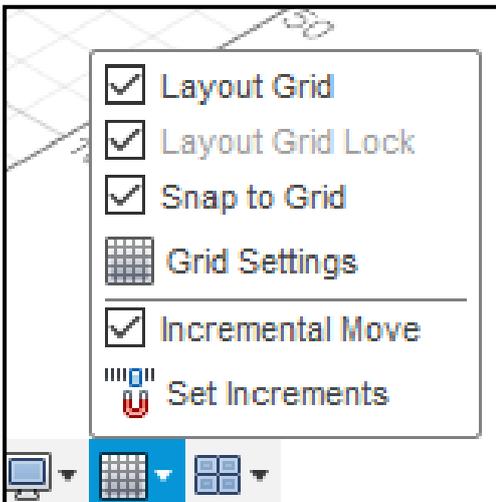


191. Now move this new pasted car to the side, out from the original using the **arrow**. Hide the 3DModel original using the lightbulb.

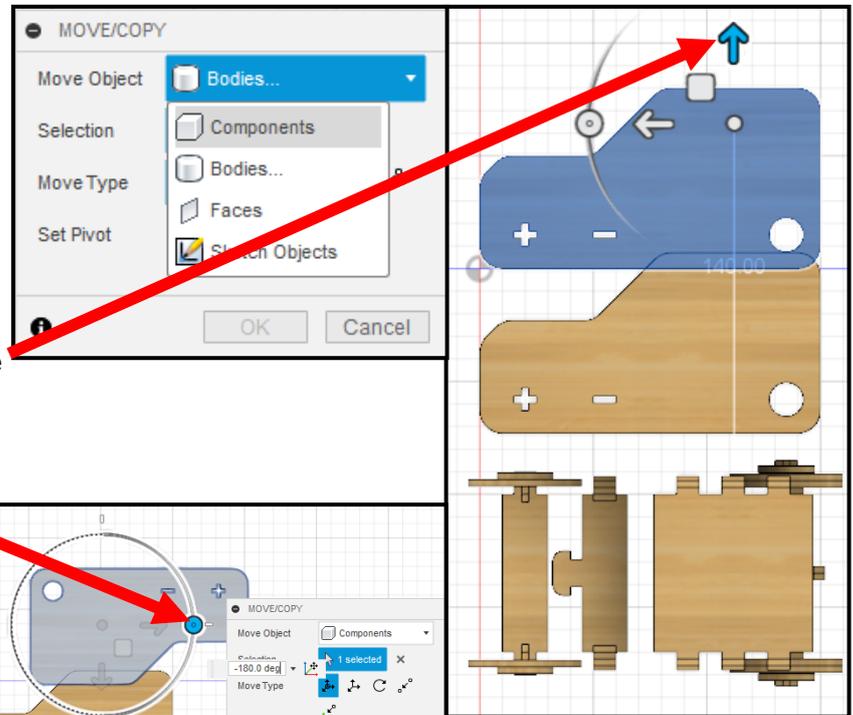


192. At the bottom click on the grid symbol and tick the box for layout grid. This is just to help with aligning parts in a future step.

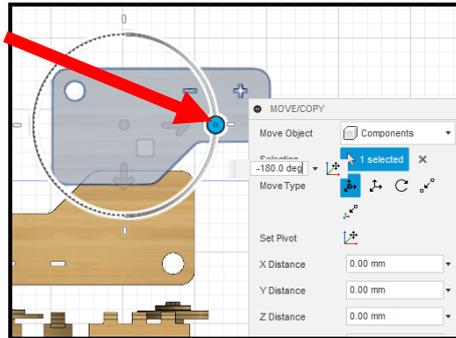
193. Now you are going to create the flat layout that can be printed out on a laser cutter. Under the **modify** tab select 'Align'. We want to align every part with the roof, so select the side face of the car as the 'From' and when doing this with all other parts always make it 'To' the **flat face of the roof**.



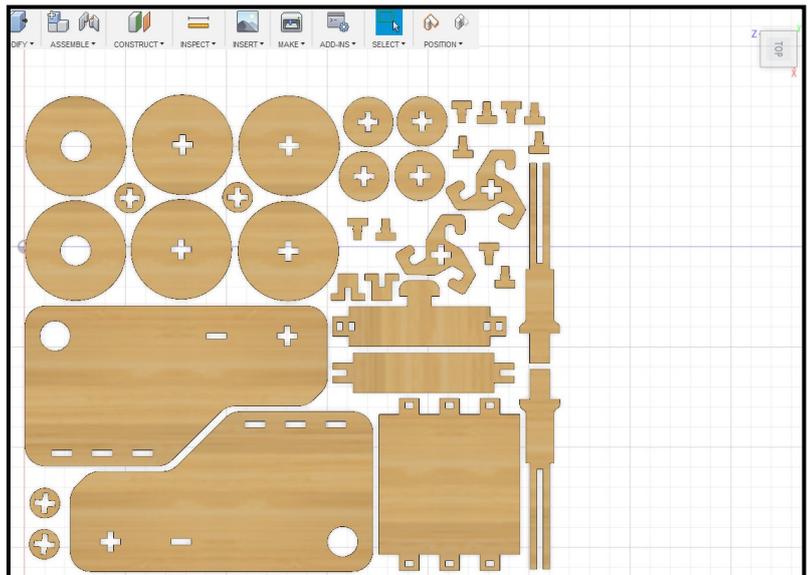
194. Press 'M' on the keyboard to activate 'Move/Copy' and make the **move object** 'Components' (Some pieces will move together occasionally, when this happens select 'Bodies' instead of components), then select the car body as the '**Selection**' piece you want to move. Then use the arrow to move the car body so that it doesn't overlap with another piece of the car.



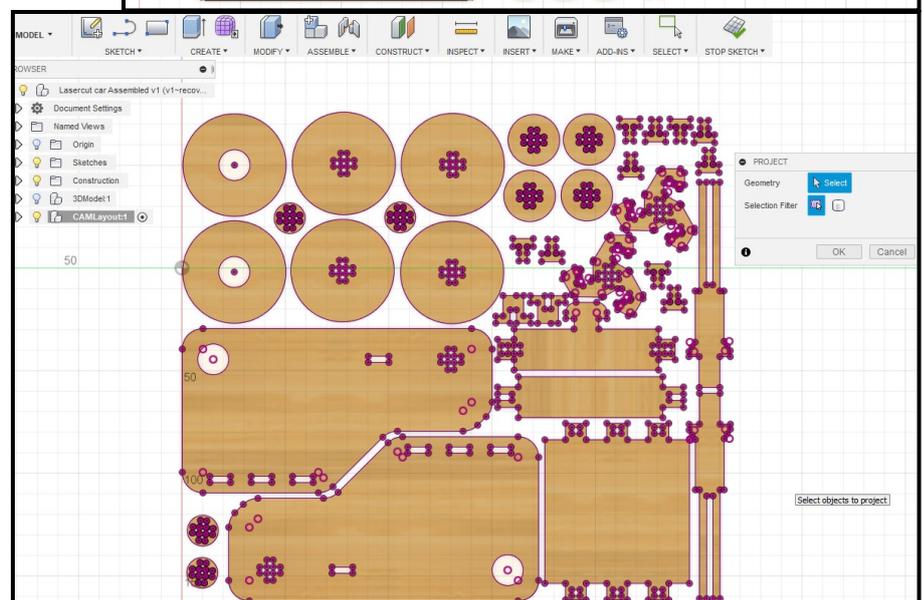
195. You can use the circle nodes to **rotate** objects. This is very useful when you want all of the objects to be close together so that you use as little material as possible to cut out the parts.



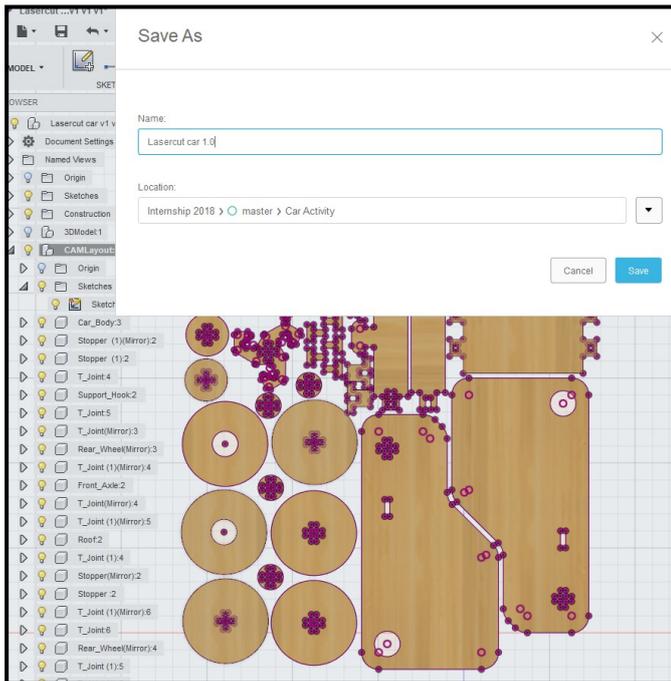
196. **Align and move all of the parts** until they are laid out in a similar fashion to the image on the right. Remember to leave some gaps between each part. They **should not** be right next to each other.



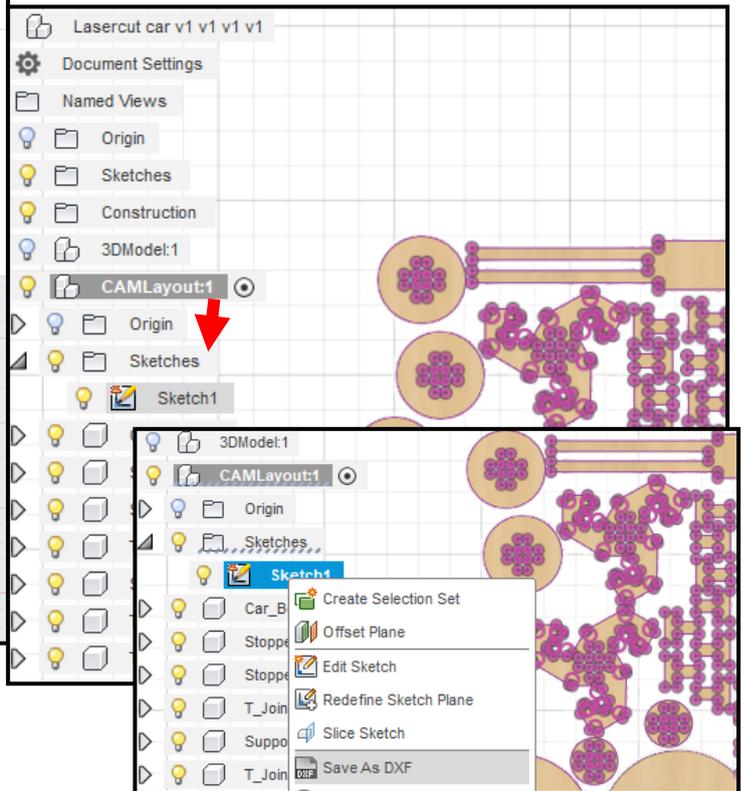
197. Start a **new sketch**. Press 'P' on the keyboard to project, select each and every part as the **geometry** you want to project, this will give all parts a pink outline.



198. Go to file and save as 'Lasercut Car 1.0'.

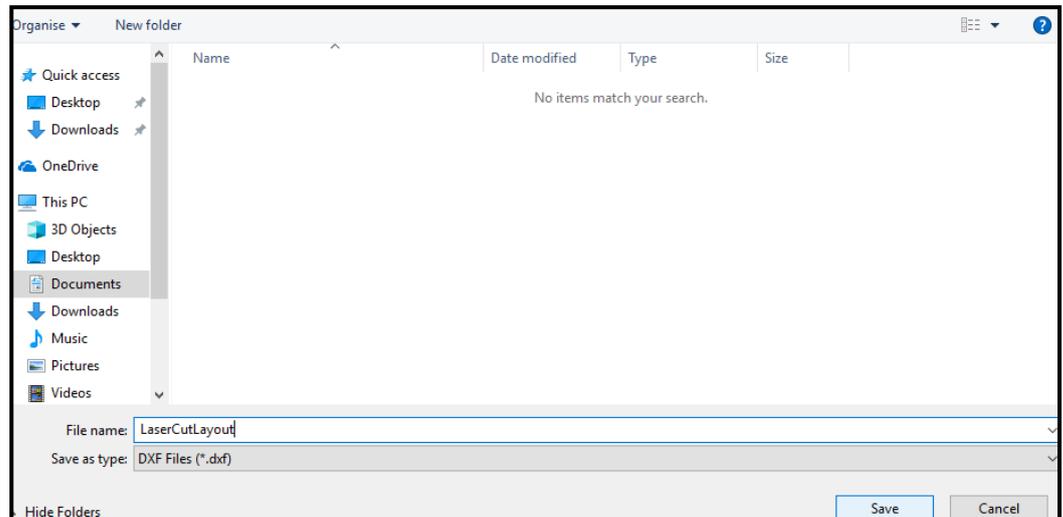


199. In the menu, under CAMLayout, in the sketches section there will be a sketch called Sketch1. Right click this sketch and click on 'Save As DXF'.



200. Save this file as 'LaserCutLayout' in documents where you can find it again.

Now you can open this DXF file on your laser cutter software and print it out.



ASSEMBLY

The T joints and stoppers may be difficult to fit onto the car due to the nodes that were created to make them fit tightly. You may need to use a **long thin file**, or **some sandpaper** to reduce the size of the nodes slightly. **Be careful not to over file the nodes** or the parts will fall out easily.

For the rear wheels take a **thick rubber band**, **cut it**, and **stick it with glue** so that it goes around

