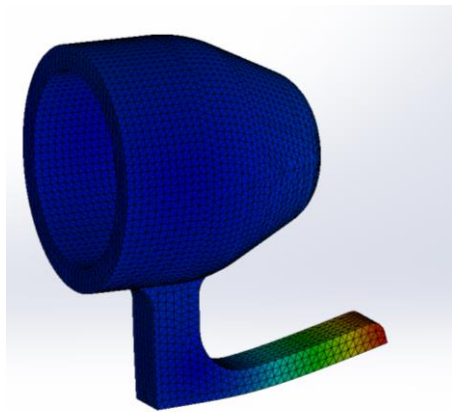


Frequency Boundary conditions exercise – Modelling assemblies and joining them

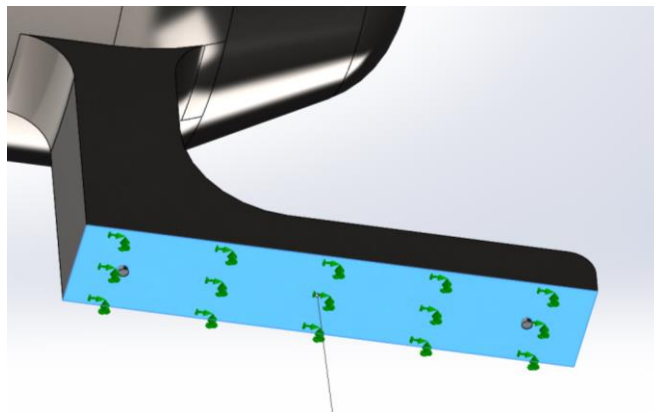
In the last exercise we saw how to assess a structure to understand its natural frequency when unrestrained. We are now going to look at ways of modelling the boundary conditions.

Assess the first 7 free modes of wing_mirror_stp2.sltprt made from steel as shown below, using a 3.3mm mesh so that you get 3 elements through the thickness of your part. You should see the following

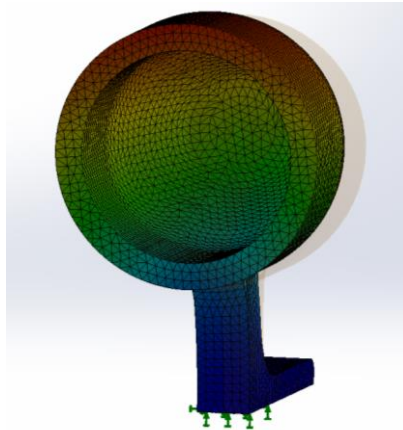


The first 6 modes are zero Hz modes and the first structural mode is as above at 1348Hz.

We are now going to assess this structure where we fix the base which would be attached to the car/door in 6 dofs as shown below.



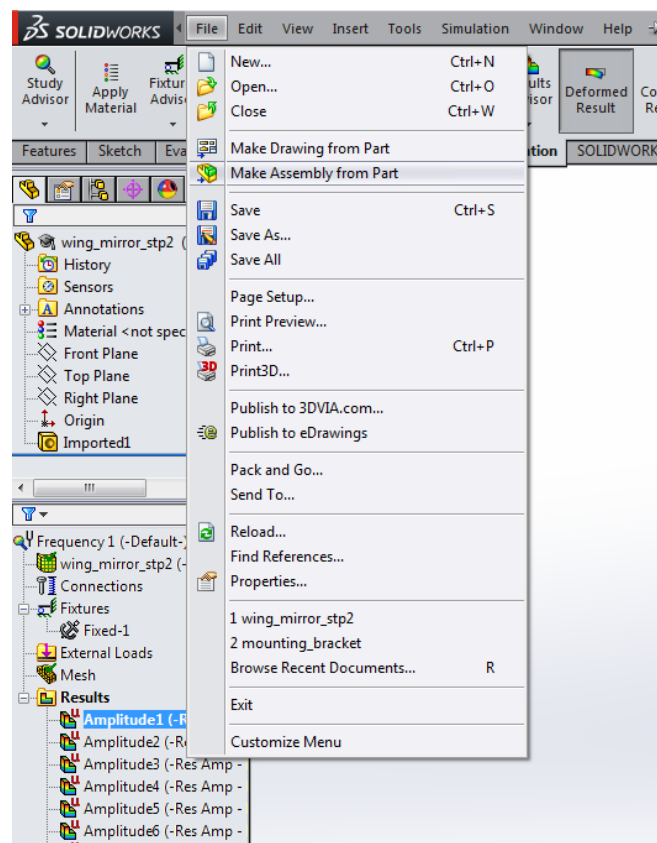
Reassess this structure and you should see the following



Note that the **1st mode** has now increased to 306Hz (Notice that since this structure is now constrained that there are no zero Hz modes). Note also, however that this is assuming that the fixing point to the car has zero compliance!

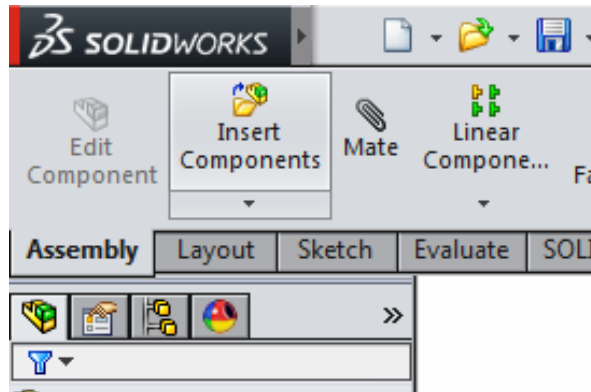
In order to address this we will now also model the fixing bracket and attach the mirror to the bracket.

To do this we will first create an assembly



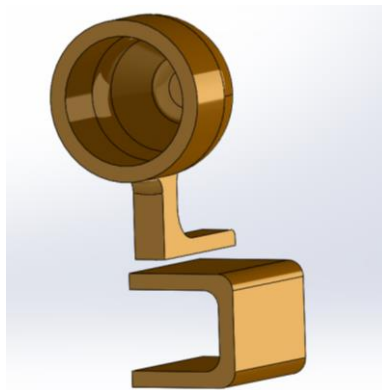
OK

Now we will add the mounting bracket

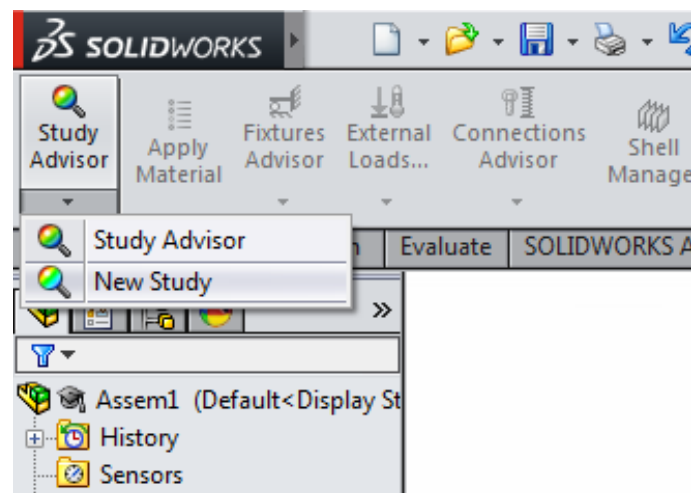


Browse to “mounting_bracket.sldprt” > Open > OK

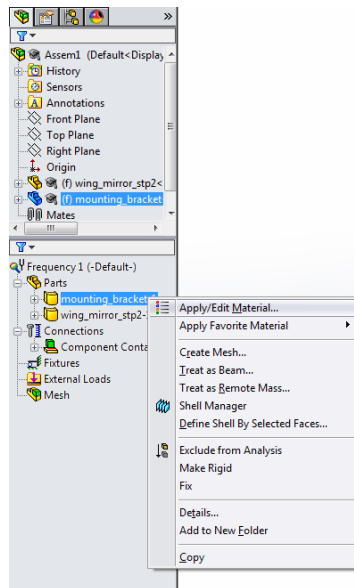
This will insert the part at the origin as shown below



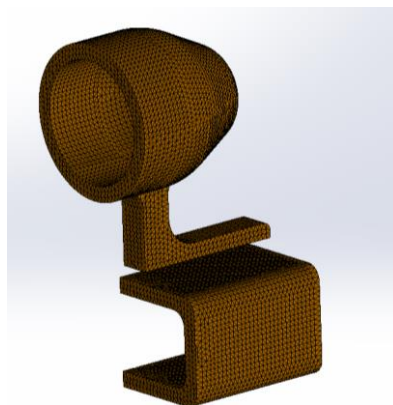
We will now rigidly attach the bracket to ground and then rigidly attach the mirror to the bracket.



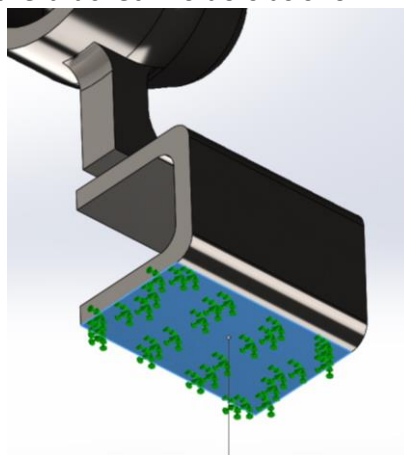
Create another new frequency study



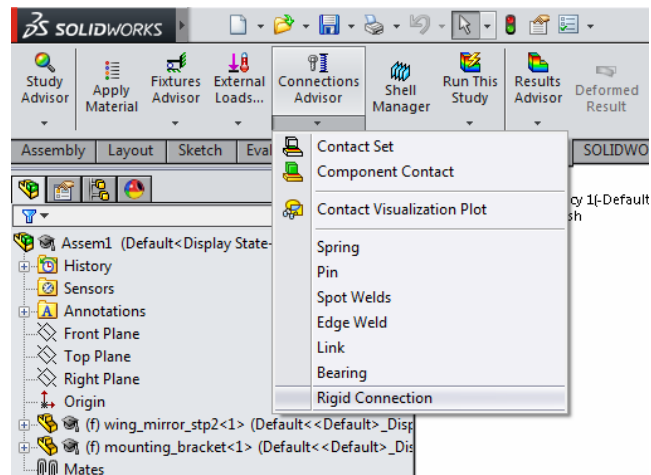
Notice in the tree structure for the assembly there are now two parts. Right click on the mounting bracket and set the material to steel. Now do the same for the mirror. Now put a 3.3mm mesh on the assembly. (Notice that if you want different size meshes on different components you can use mesh controls on each part within the assembly. Your assembly should now look as follows.



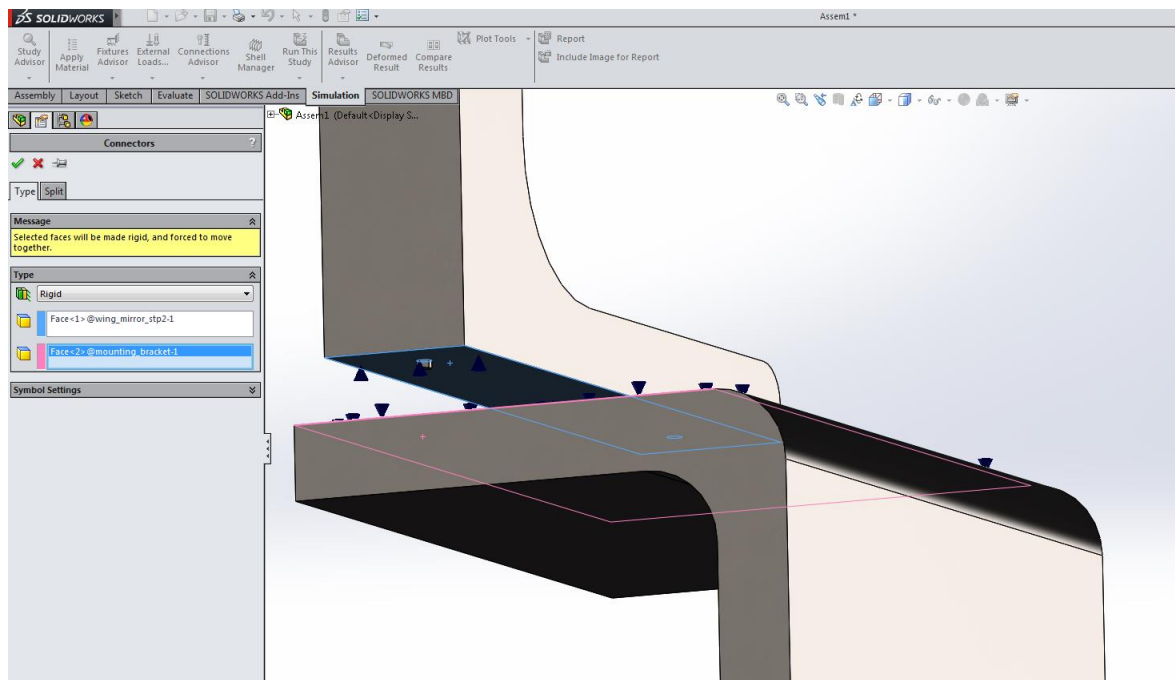
Now constrain the bottom of the bracket in 6 dofs as shown.



We now need to attach the mirror to the bracket. We will do this using a rigid connection.

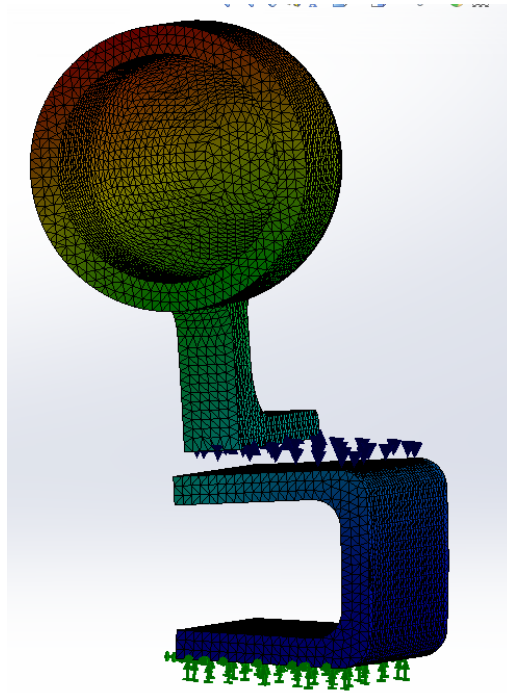


Select the two faces that you want to rigidly connect.



OK.

Solve this model. You should see the following results.



Notice that the 1st mode has now reduced from 306Hz to 119Hz because we are now taking into account the stiffness of the supporting structure. This is vital when trying to simulate the SYSTEM response rather than just the component response and is often the reason why test results and FE models do not correlate.

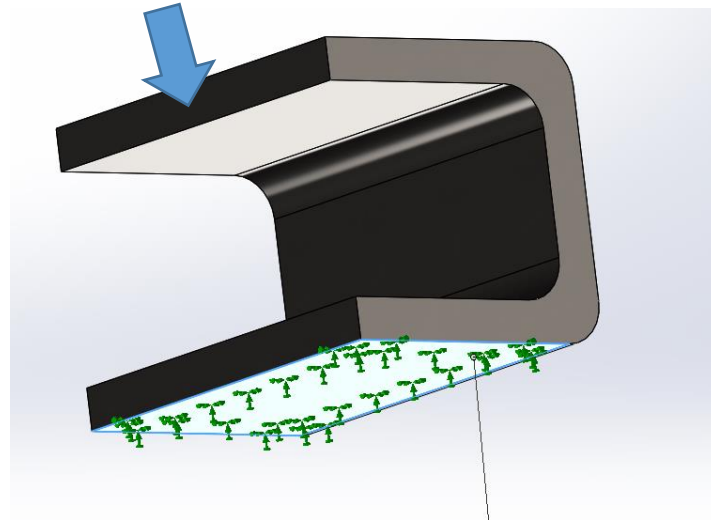
However, modelling the supporting structure obviously adds to the size and the complexity of the model. How many nodes, elements, dofs are in this model and how long did it take to solve?

You should see the following....

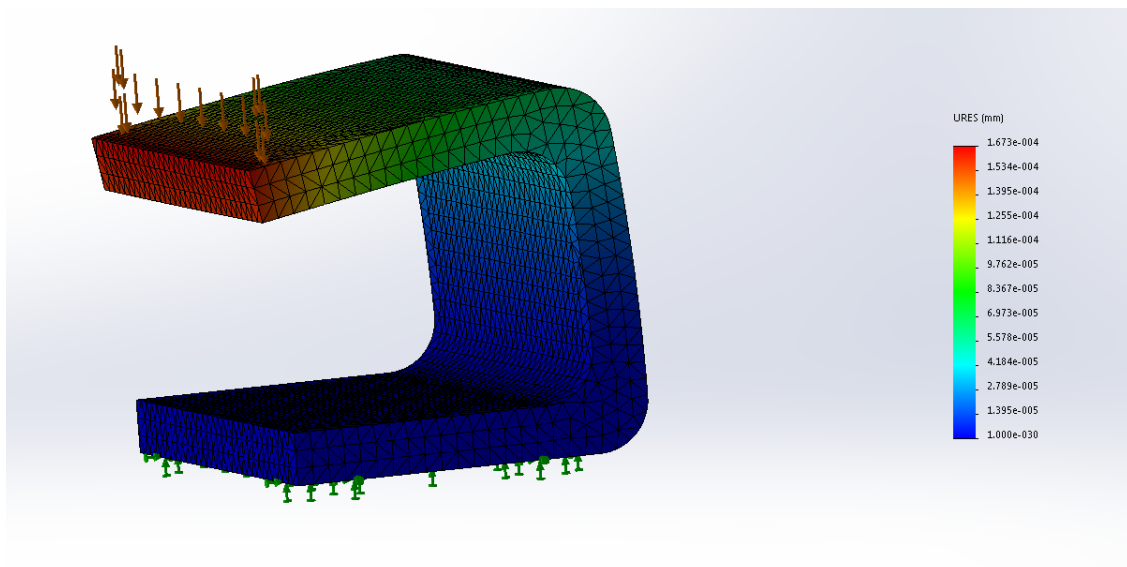
Solver Message	
No. of nodes	136843
No. of elements	87585
No. of DOF	413586
Total solution time	00:00:23
OK Save Help	

In order to reduce this instead of modelling the actual structure we can simply model the stiffness of the structure. In order to do this first we must do a static assessment of the attachment point.

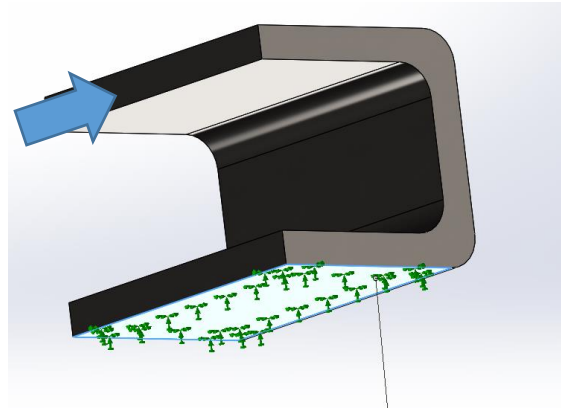
In order to do this create a static model of the bracket and assess the stiffness of the bracket when constrained and loaded as shown.



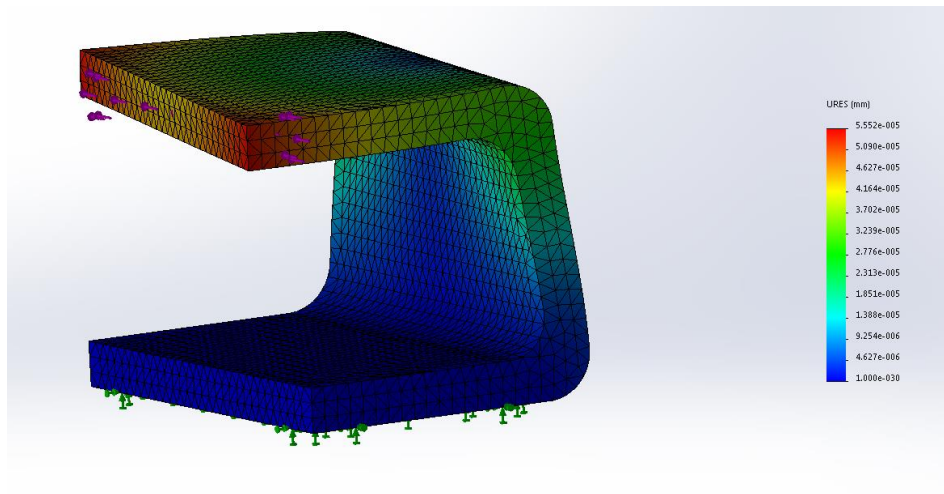
1. We need to constrain the lower face and apply a load on the face shown in the direction shown so that we can understand the compressive stiffness of the bracket. Because the model is linear the tensile stiffness of the bracket will be equal and opposite. We should see the following



The compressive/tensile stiffness of the bracket can thus be calculated using $F=kx$. You should find that the stiffness is 5977N/mm.



1. We need to constrain the lower face and apply a load on the face shown in the direction shown so that we can understand the shear stiffness of the bracket. We should see the following

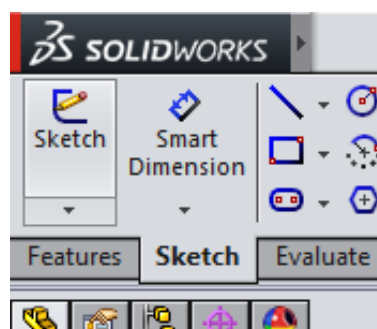


The shear stiffness of the bracket can thus be calculated using $F=kx$. You should find that the stiffness is 18011N/mm.

Now that we understand the stiffness of our attachment point we can now model the mirror only and attach this to springs of the same stiffness that are attached to “ground”.

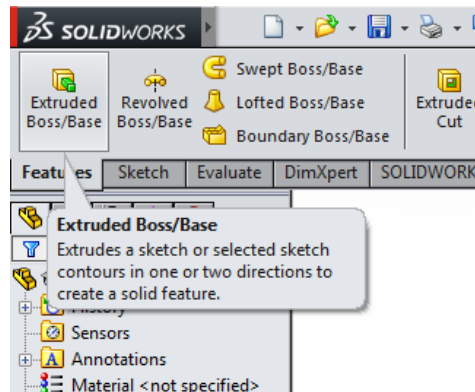
Reopen the mirror.

The first thing that we need to model is our “ground” plane.

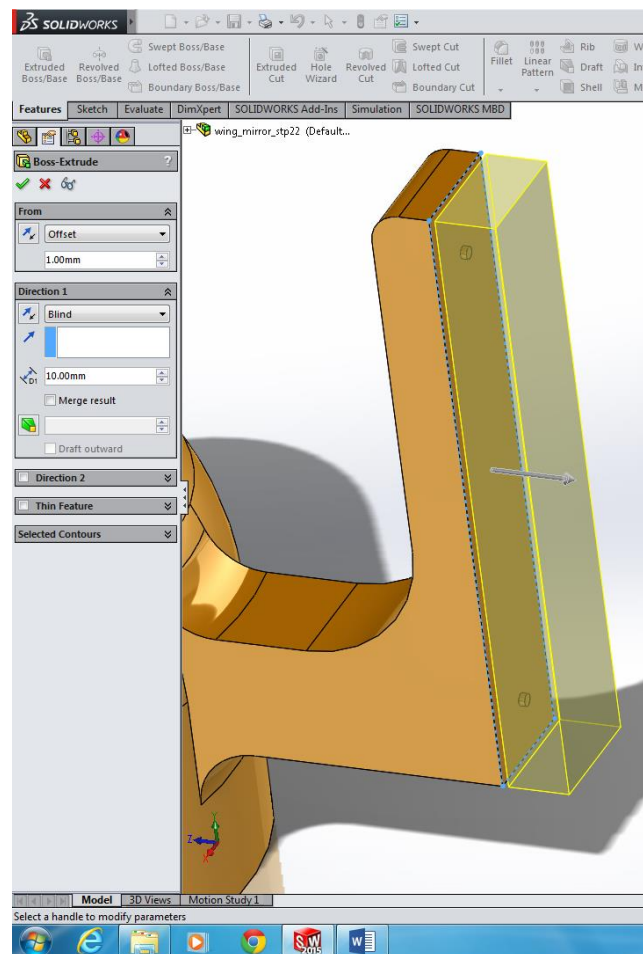


Choose the plane on the bottom of the bracket and create a rectangle at the base of the mirror that we will use as “ground”

OK > Exit sketch



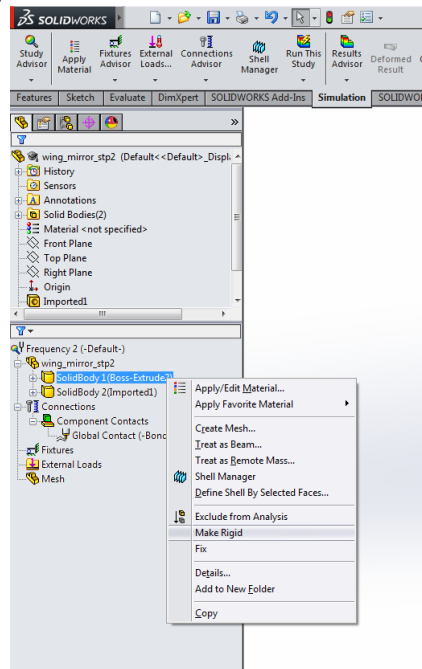
Now extrude this away from the mirror by 10mm with a 1mm offset and create a new part



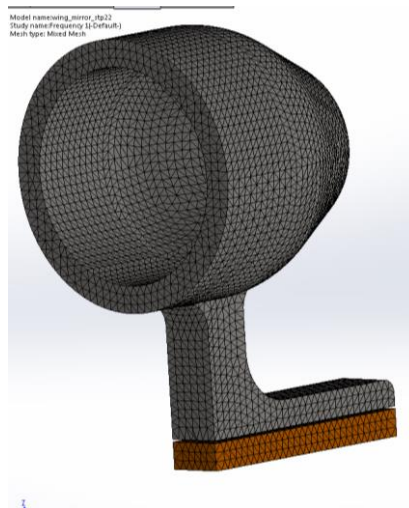
When you do this make sure that you uncheck “merge result” so that we have two different parts.

We will now create springs with the same stiffness between the ground plane and the mirror to represent the stiffness of the bracket. We can make the ground plane rigid and attach this to ground.

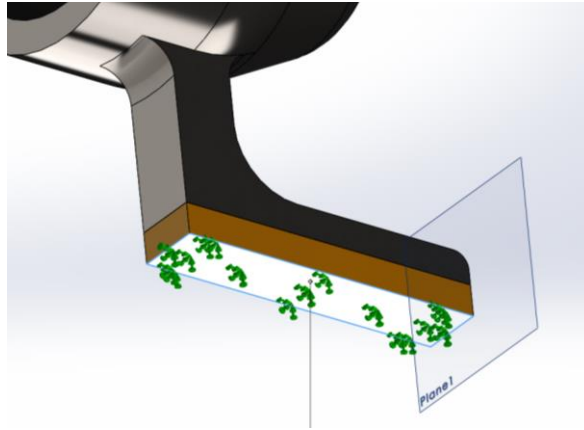
Create a new frequency study.



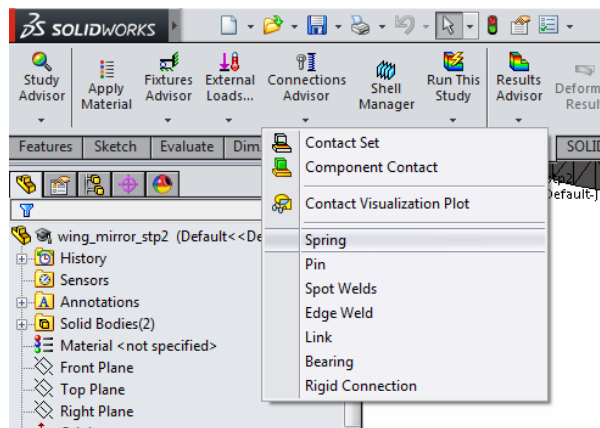
First make the ground plane rigid. Then make the mirror from steel and mesh the part using 3.3mm elements as before so that your part should look as follows



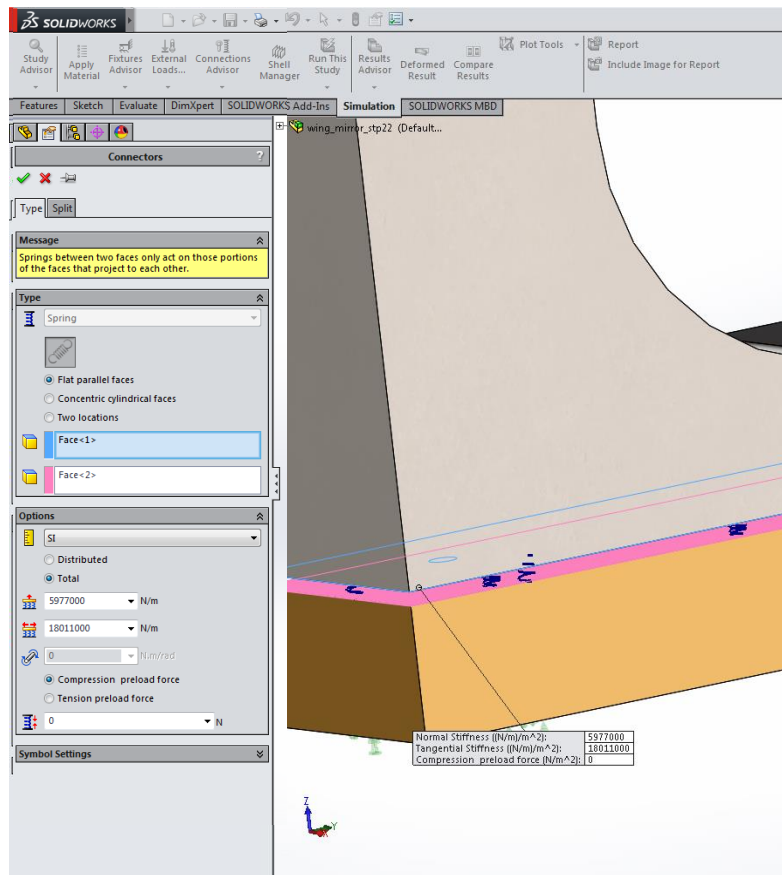
Now constrain the bottom of the ground plane in all 6 dofs.



We will now define the springs between the mirror and the ground plane with the same stiffness as the bracket.



Note we calculated 5977N/mm in tension/compression and 18011N/mm in shear. However the stiffness between the two surfaces is given as N/m so we must multiply our values by 1000. Also, make sure that you define the total stiffness not the distributed stiffness



Resolve your model - what frequency do you get? Why do you think that is this different? How long did this take? How many degrees of freedom did you have?

This approach demonstrates the methods that you may use to model the supporting structure and the associated pitfalls of such techniques.