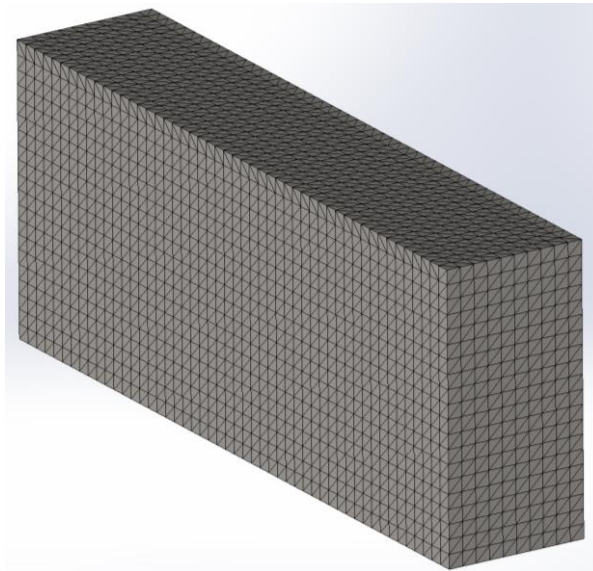


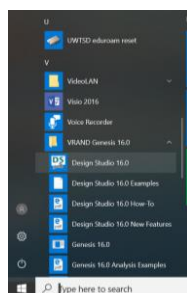
Introduction to Design Studio: Single Loadcase

In this exercise we will create a mesh in solidworks of a package space and then export this mesh to Design Studio. In Design Studio we will carry out a Topology Run to provide design direction as to what an efficient cantilever beam would look like when it is subjected to one load.

1. Creating your package space in Solidworks.
 - Input the file “cant_igs” and then mesh this with 10mm first order “Alloy Steel” tets. It does not matter what type of analysis you choose as this loadcase will be deleted in Design Studio – however you need to choose something so that you can create your mesh, so choose static analysis. Your part should look as follows;

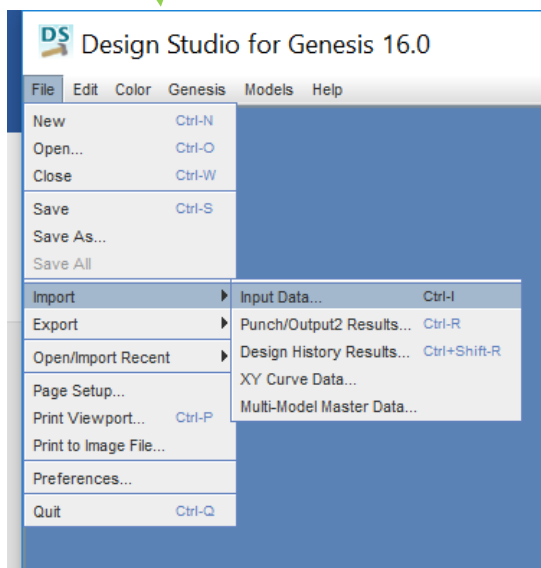
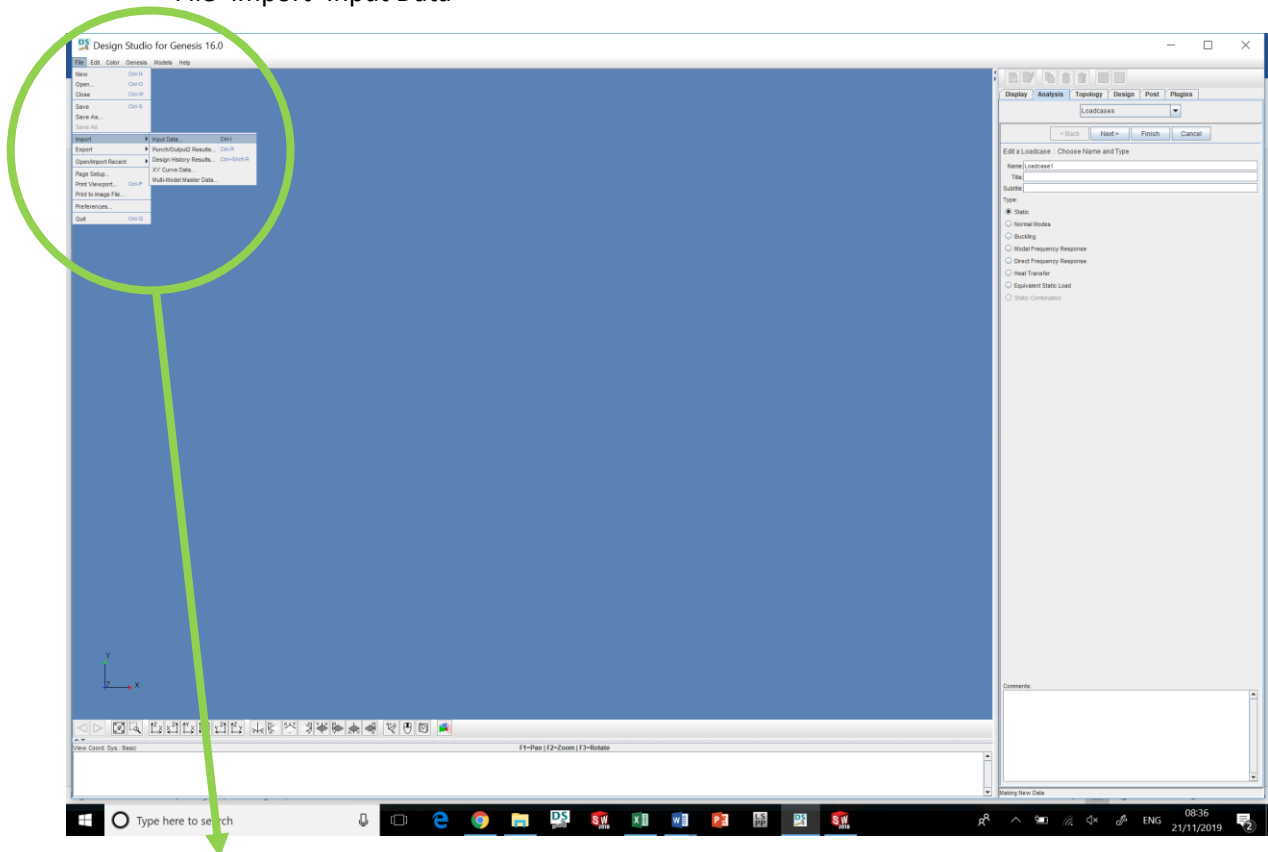


2. Exporting your Nastran deck.
 - Simulation>export
 - Save as type “NASTRAN”
 - Call your file cant.dat
3. Open and examine cant.dat in a text editor such as “wordpad”
4. Open Design Studio

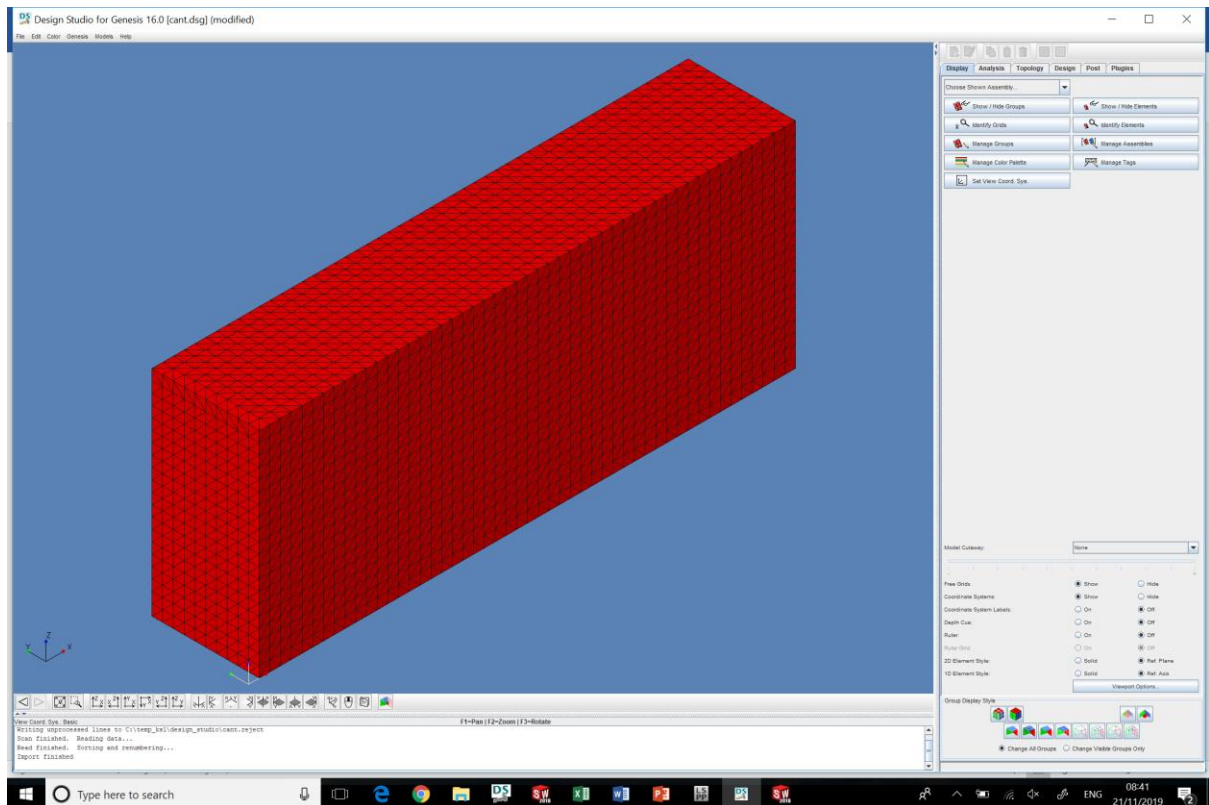


5. Import cant.dat into Design Studio

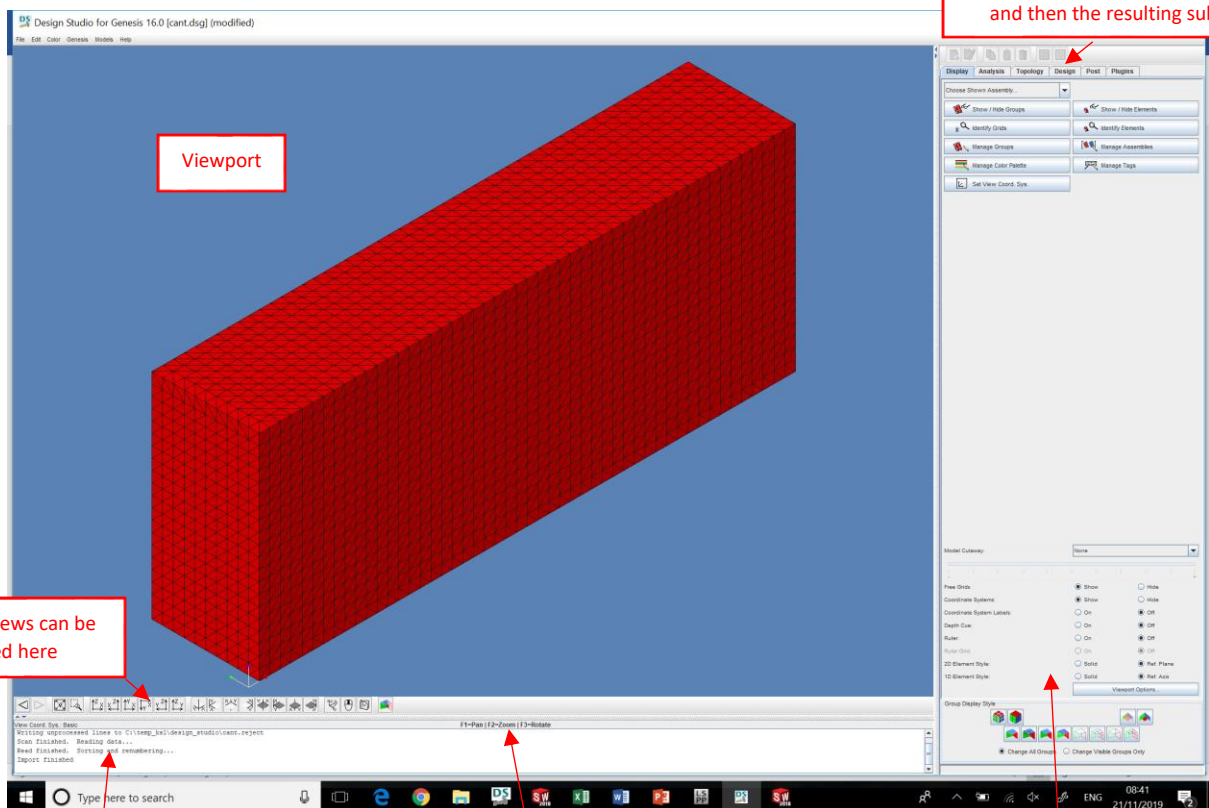
- “File>import>input Data”



- Find and import your Nastran deck. (note that Design Studio filters for “*.dat” and the file written by Solidworks is cant.DAT)
- Your part should look as follows;



- Note the following about the GUI to solidworks;



Commands are entered via these tabs and then the resulting sub menus

Viewport

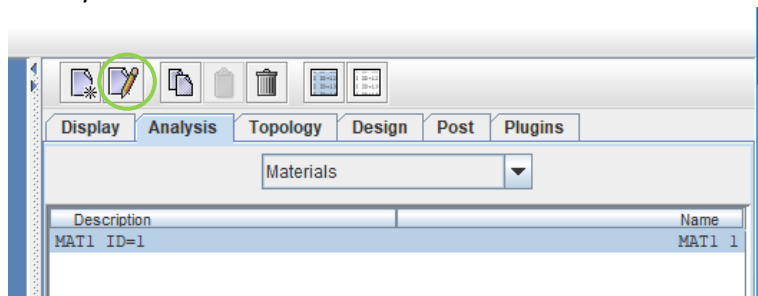
Standard views can be selected here

Messages appear here

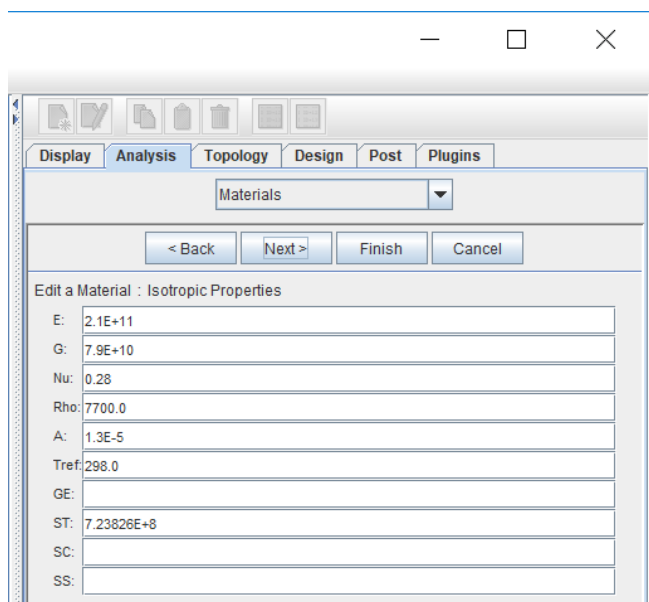
Use F1 to pan, F2 to zoom and F3 to Rotate whilst moving the mouse

Graphics can be changed here (in the "Display tab")

7. Examine the cant.dat in Design Studio
 - Examine this your material. (“Analysis>Materials”)
 - Highlight “MAT1”
 - Modify the material



- You should see that it is an Isotropic material
- “Next”
- You should see the following material properties that Solidworks has given the part

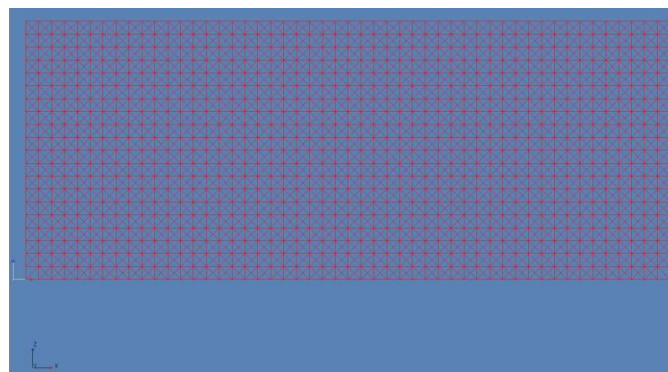


- “Finish”
- Examine your Physical Property. (Analysis>Group Properties)
- Highlight “PSOLID1”
- “Modify” the Group Property
- You can see that this points to MAT1 and has no other information. Since we are using solid elements the majority of the physical properties are defined by the geometry of the element. If we were using 2D elements, then the thickness of the shell would be defined here and if we were using 1D elements then the cross section would be defined here.
- “Finish”
- Examine your Loadcase. (“Analysis>Loadcases”)
- Highlight “STAT”
- “Modify” the Loadcase
- The first screen shows that it is a static loacase
- “Next”

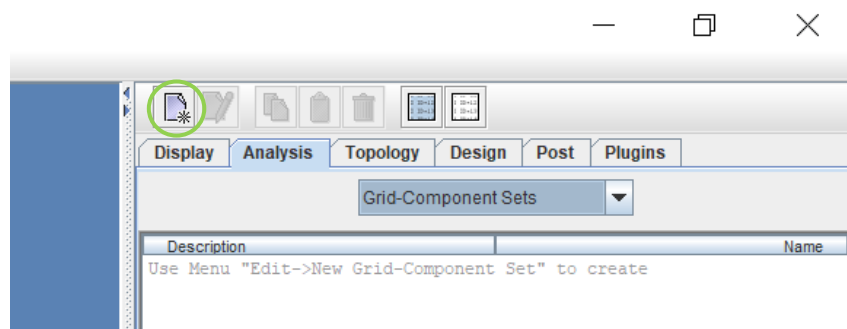
- The second screen points to any constraints/contacts. Note that these are all set to “None” as these were not defined in Solidworks
- “Next”
- The third screen points to any Loading. Note that these are also set to “None”
- “Next”
- The forth screen shows the output requests. Note that various requests have been set by Solidworks.
- “Finish”

8. Apply your constraints in Design Studio

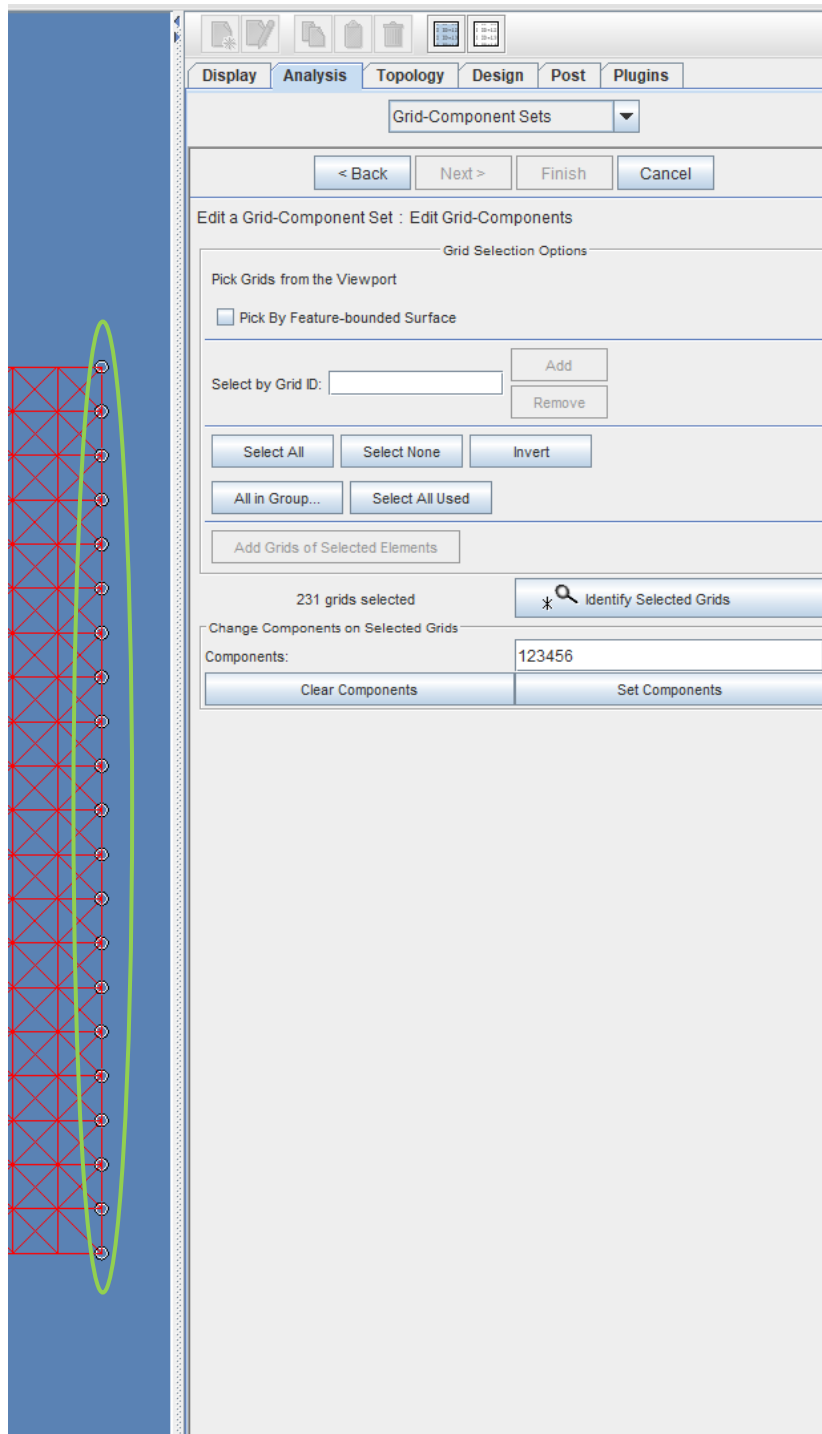
- Choose the ZX standard view (“Left”)
- Set the display to Wireframe (“Display>Wireframe”)
- Your part should now look as follows;



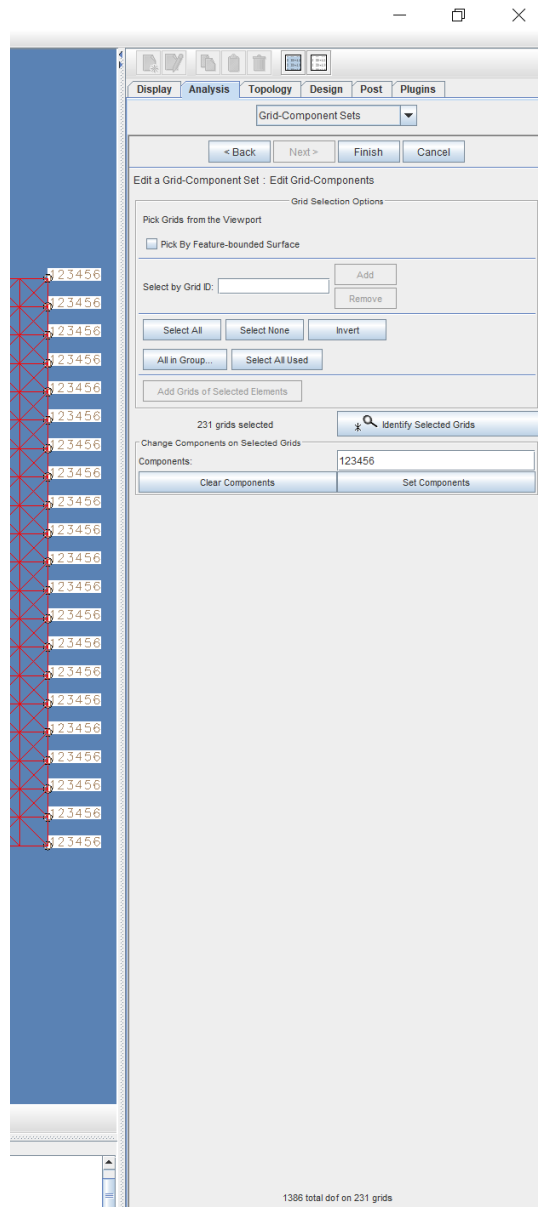
- Define your constraints (“Analysis>Grid-Component Sets”)
- “New”



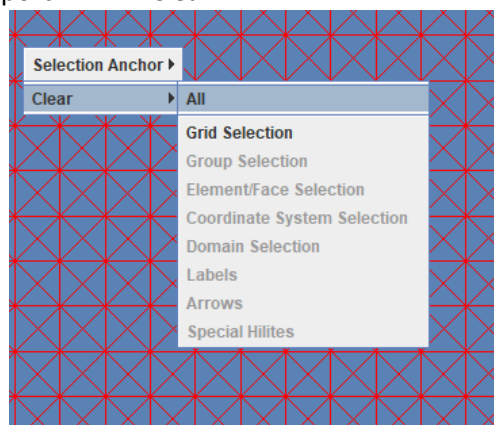
- Choose the radio button “Single-Point Constraints”
- Type name the name BCs for your constraints
- “Next”
- The next screen is asking you to pick the nodes from the viewport that you want to constrain
- Click and drag a box in the viewport around the nodes that we will constrain at on the right hand face;



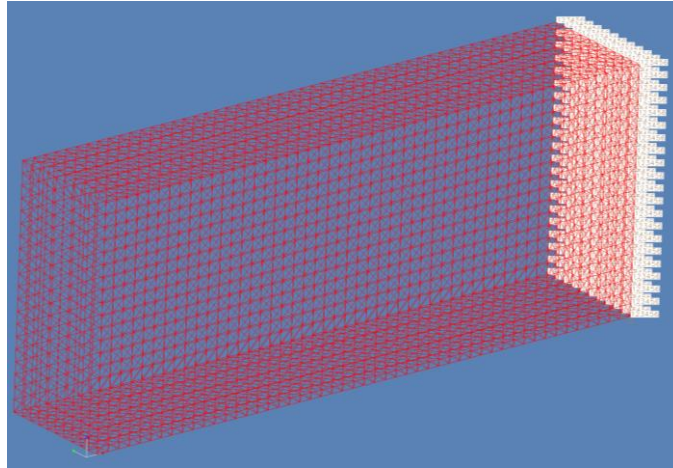
- You can see that we have selected 231 grids. We are going to fix components 1-6 for all of these grids
- “Set components”



- You can now see that all the grids are marked as being fixed in degrees of freedom 123456. You can also see that there are 1386 dofs on 231 grids ($6 \times 231 = 1386$)
- “Finish”
- Note that the grids are still selected in the viewport. You need to deselect these. Put your cursor in the viewport. “RMB>Clear>All”



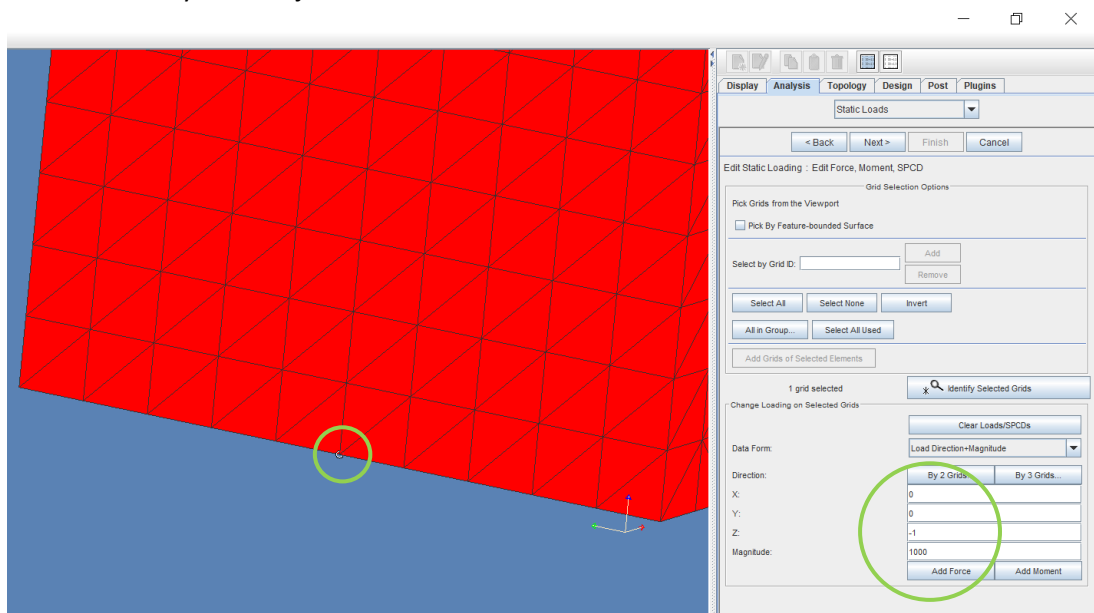
- Note that you have now defined one Grid component set called BCs. Try highlighting this and then displaying it on the viewport. (not that by default Design studio only shows 100 labels on the viewport. In order to change this do the following: “Display>Viewport Options” then increase the number of labels to draw. “Finish”)
- Your part should now look as follows;



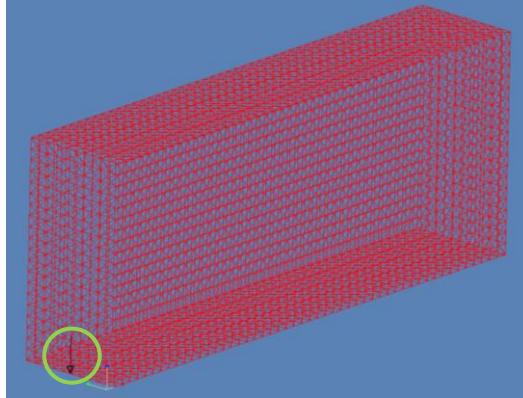
- We have now said that the grids on this back face can not move in any degrees of freedom.
- Save your work (“File>export>input data”)
- Call your work cant2.dat
- “Save”

9. Apply your loads in design studio

- “Analysis>Static Loads>New”
- Choose the radio button “Force, Moment, Pressure, SPCD”
- Type “1KN_minus_Z” in the name box
- “Next”
- Choose the single grid in the middle of the edge at the bottom of the face that is opposite the nodes that you have just constrained.



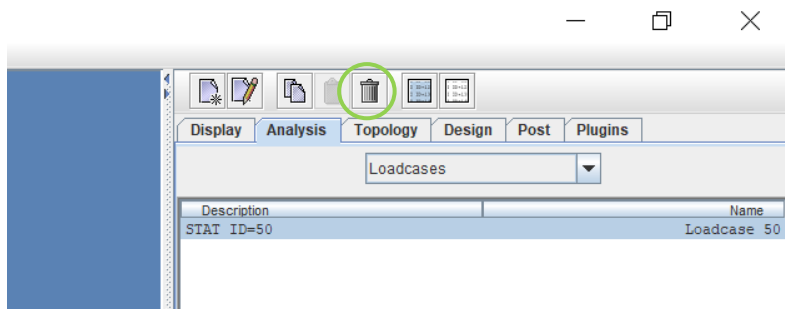
- Set the direction vector so that the force is applied in the negative z direction.
- Set the magnitude to 1000N
- “Add force”
- “Finish”
- Examine what you have just defined. It should look as follows;



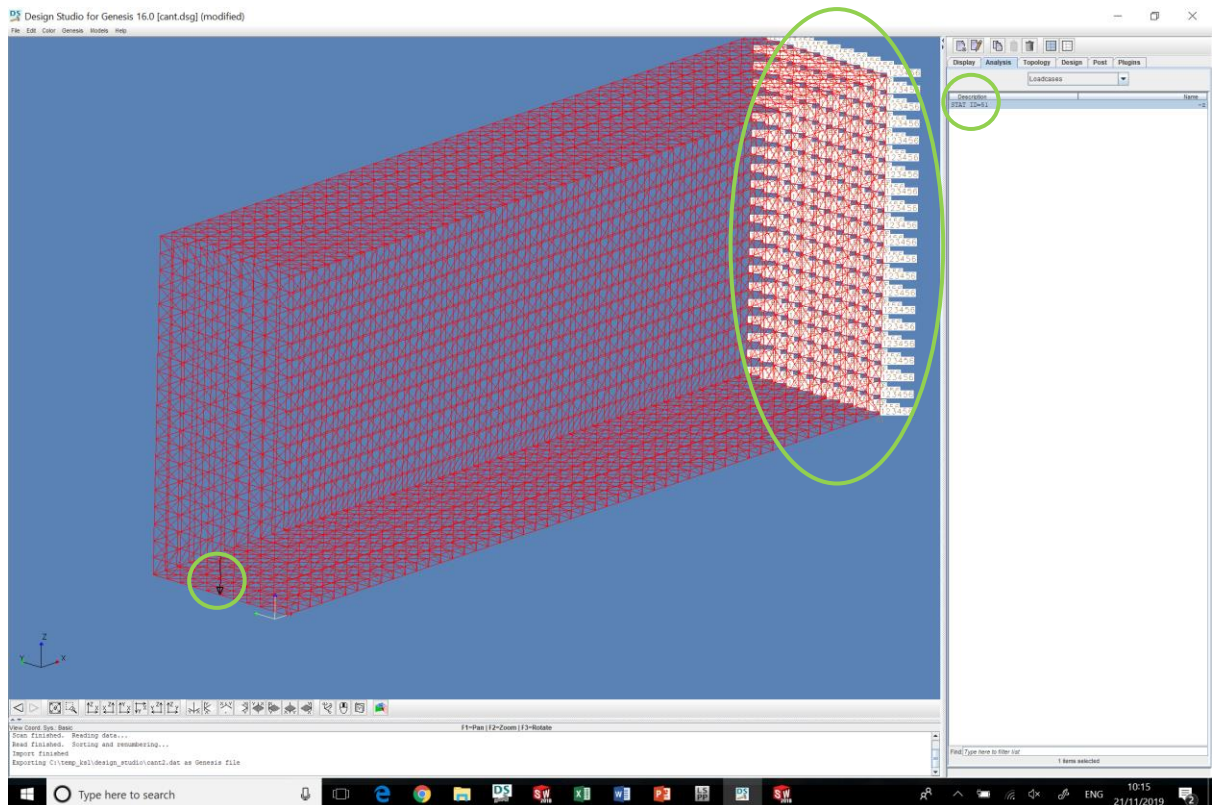
- Save your work (“File>export>input data”)
- Call your work cant3.dat
- “Save”

10. Define your loadcase in Design Studio

- Now that the constraints and the load has been defined, we need to create a loadcase that joins these together and then specifies what files we want written out.
- Delete the loadcase defined in Solidworks (“Analysis>Loadcase”)
- Highlight “STAT” and “Delete”



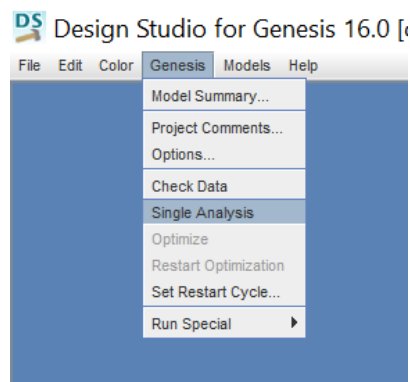
- “New”
- Click on the “static” radio button
- Type “-z” in the Name box
- “Next”
- Choose “BCs” from the drop down menu for the SPCs
- “Next”
- Choose “1KN_minus_z” from the drop down menu for the Load set
- “Next”
- Set the output requests to both for the Displacements, Element Stress and Element Strain
- “Finish”
- Examine what you have just defined. It should look as follows;



- You can see that we have now defined one loadcase which links the boundary conditions that we have set up with the load that we have defined.
- Save your work (“File>export>input data”)
- Call your work cant4.dat
- “Save”

11. Run your analysis

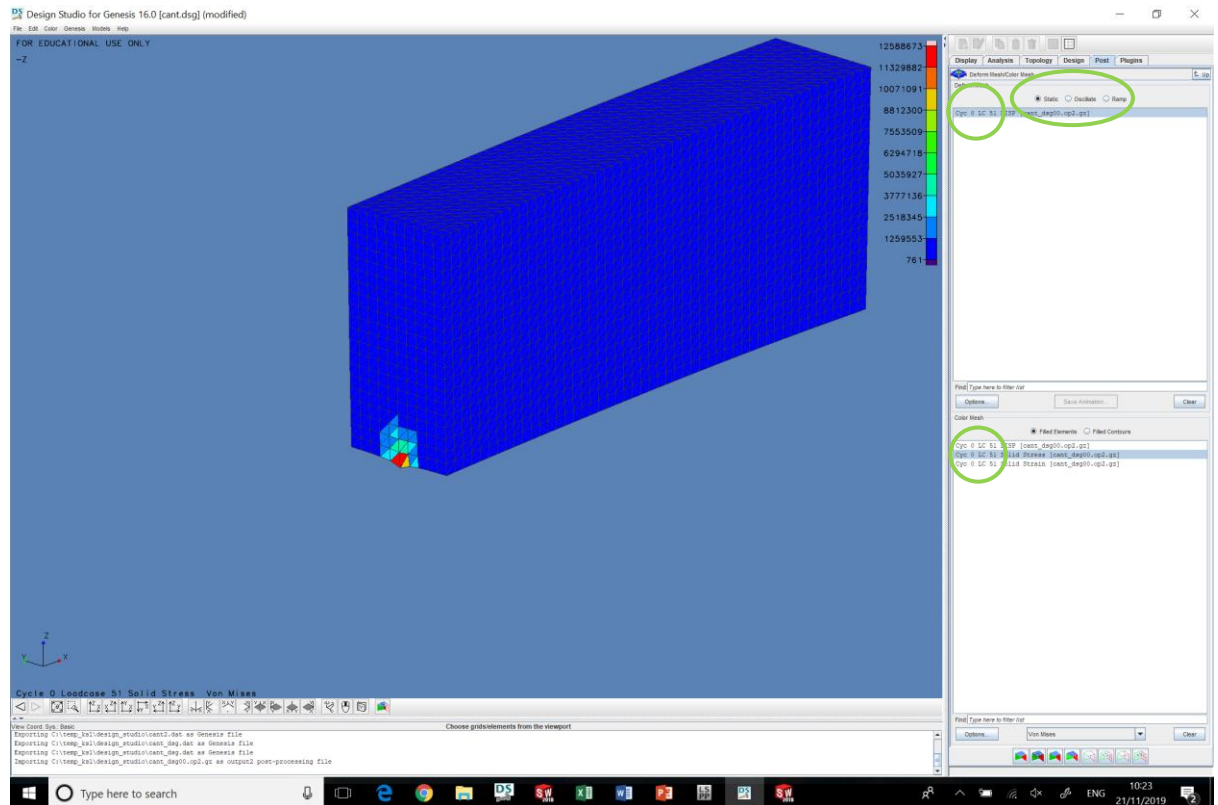
- “Genesis>Single Analysis”



- Read your results in (“Import post>choose your op2 file>Import>close”)

- “Post>Deform Mesh/Color Mesh”

- Click on the results in the deform mesh box to deform your mesh
- Click on the stress results in the color mesh box to show your stresses.
- Your results should look as follows;



- Note that although this is a static analysis, you can animate your results by clicking on the “oscillate” or “ramp” radio buttons
- Check that this is behaving as expected. You can see that we are getting a maximum stress of 12.6MPa.

Now that the static analysis has been set up on the package space, we can carry out the topology to define design direction for an efficient structure for this one loadcase.

13. Defining your variable

- Our variable is the available package space.
- “Topology>Topology Regions”
- Click on your one PSOLID
- “Modify”
- “Next”
- “Finish”
- Notice that the default that we have just accepted is that the initial mass fraction is 30%. That is to say that we will use 30% of the material in the package space.

14. Defining your objective

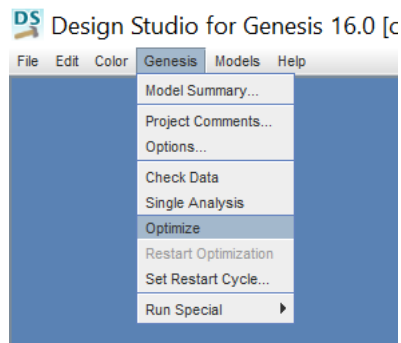
- Our objective is to minimise strain energy
- “Topology>Topology Objectives”
- “New”
- Click on the “strain energy” and the “min” radio buttons
- “Next”
- Highlight your loadcase
- “Finish”
-

15. Defining your constraint

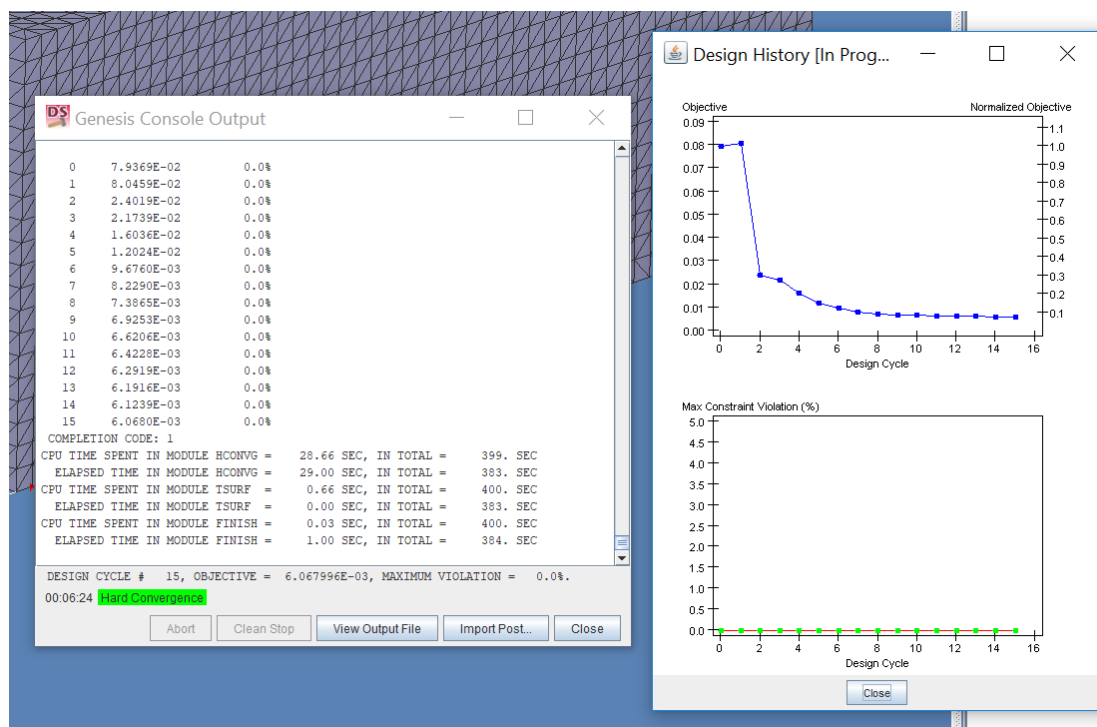
- Our objective is constraint is to only use 30% of the available material
- “Topology>Topology Constraints”
- Click on the “Mass fraction” radio button
- Set the upper bound to 0.3
- “Finish”
- Save your work (“File>export>input data”)
- Call your work cant5.dat
- “Save”

16. Optimise your model

1. "Genesis>Optimize"



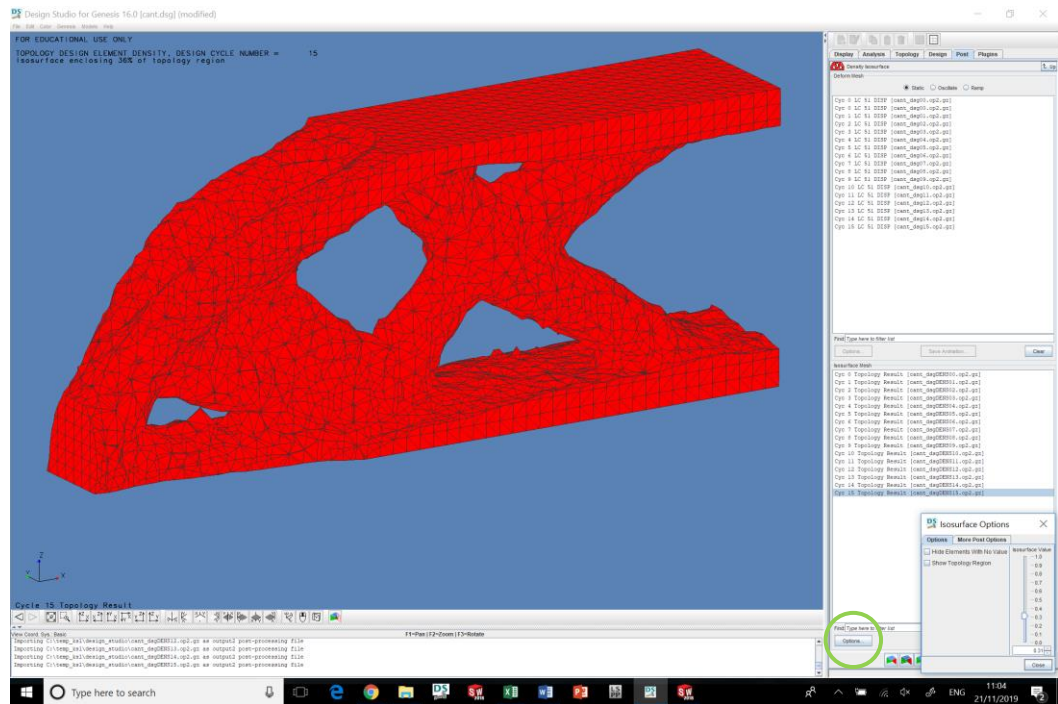
2. You should see something like the following;



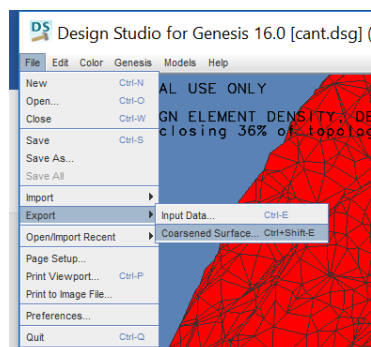
3. The above analysis has converged in 15 cycles
4. The normalised objective (strain energy has fallen from 1 to about 0.1)
5. The constraint has not been violated (we asked Design studio to start with 30% of the material and constrained it to use 30% of the material in all subsequent steps)
6. "import post"
7. Hold down shift and choose all of the results files (note that there are 32 files – stress results and density results for the elements for iterations 0-15)
8. "import"
9. Close both of the above windows

10. View your results

- “Post>Deform mesh color mesh”
- Note that there are deformations and displacement/stress/strain results for all 16 iterations. Examine these.
- “up”
- “Density isosurface”
- This shows the density of each element for each of the iterations. Examine these.
- Your final iteration will look something like the following;



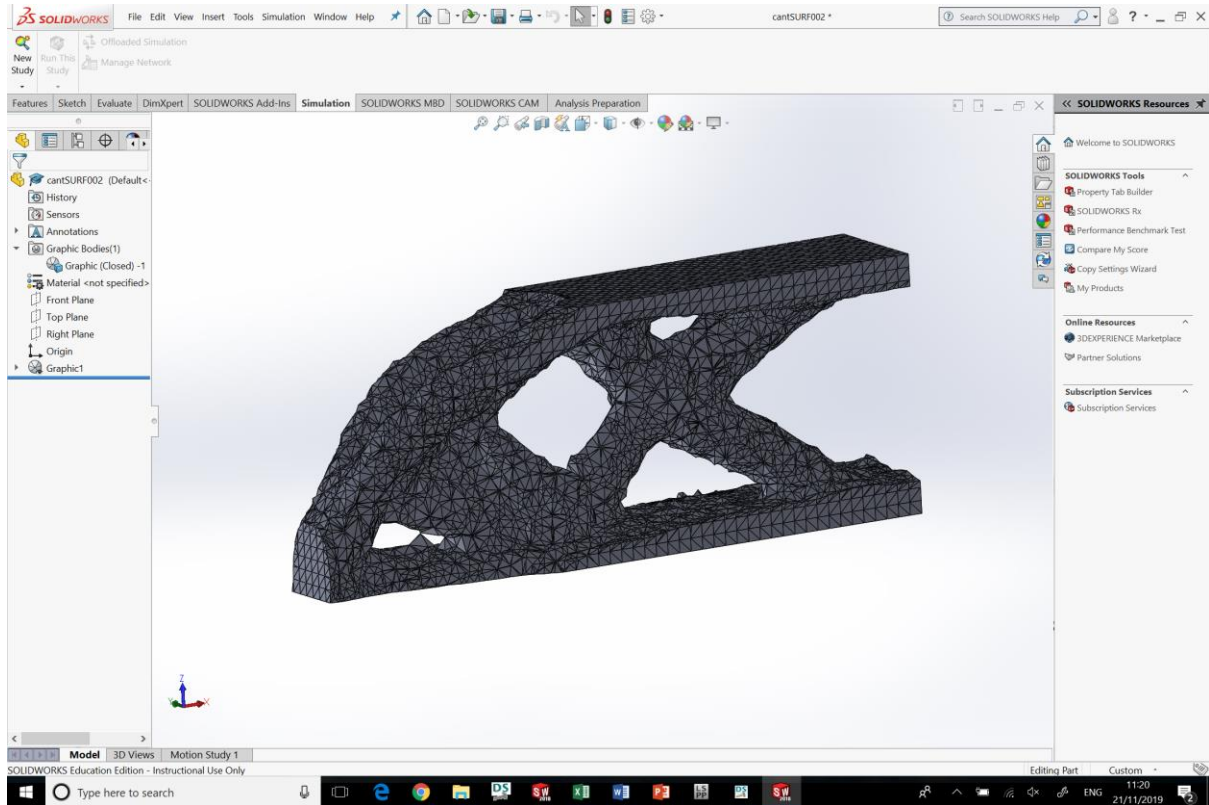
- You can change the percentage of the elements being displayed in the options button
- You can consider the elements with high percentages to be “important”. These display the main loadpaths.
- You can export an graphics file of this final iteration
- “File>export>coarsened surface”



- Move the slider on the pop-up window to fine and choose the STL radio button
- Save

11. Using your results to inform your design

- The STL file that was written out of Design Studio can then be read into a CAD package such as Solidworks as can be seen below.



This can then be overlaid on the original package space and then the original package space can be modified accordingly, resulting in an efficient structure as seen below.

