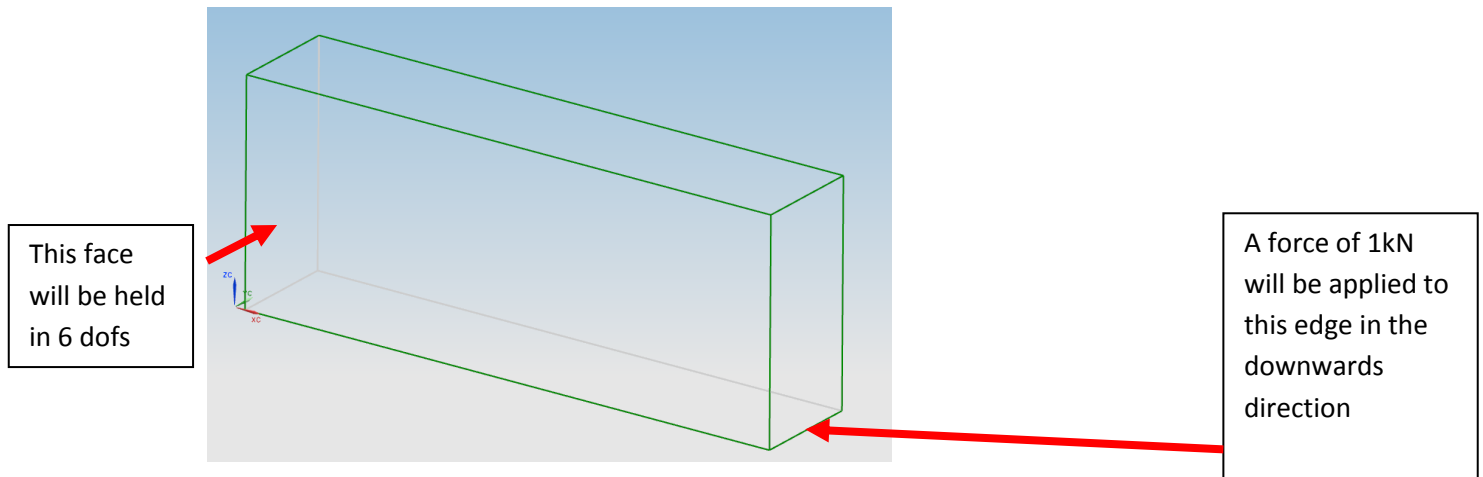


GENESIS – INTRODUCTION TO TOPOLOGY

First of all we need to generate our FE model from the CAD that we have. This model is a simple cantilever beam with a load on the end.

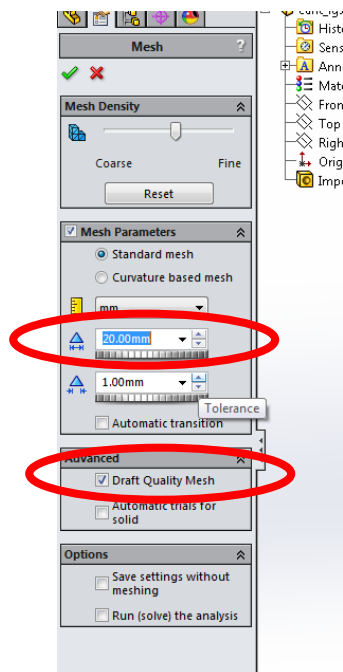
Start Solidworks then open “cant_igs.prt”. The below model is imported;



We will assume that the block is made out of steel and we are going to use TOPOLOGY in GENESIS to show us where the major loadpaths are so that we can ‘carve’ the most efficient shape out of the solid block that is available to us in order to minimise the strain energy whilst only using 30% of the available material to us.

1 – SET UP YOUR BASELINE

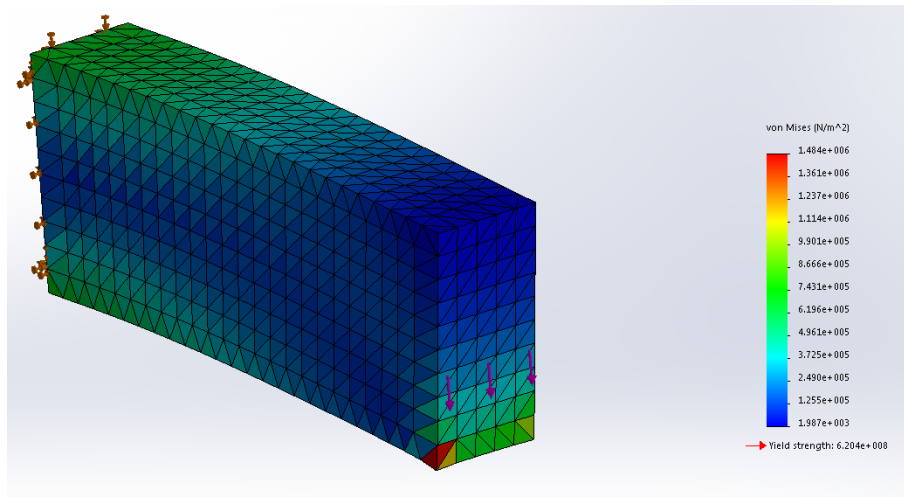
Mesh the part with FIRST ORDER tets made from steel of 20mm length.



Constrain your model

Apply your force

Solve your model and check that the results are as expected;

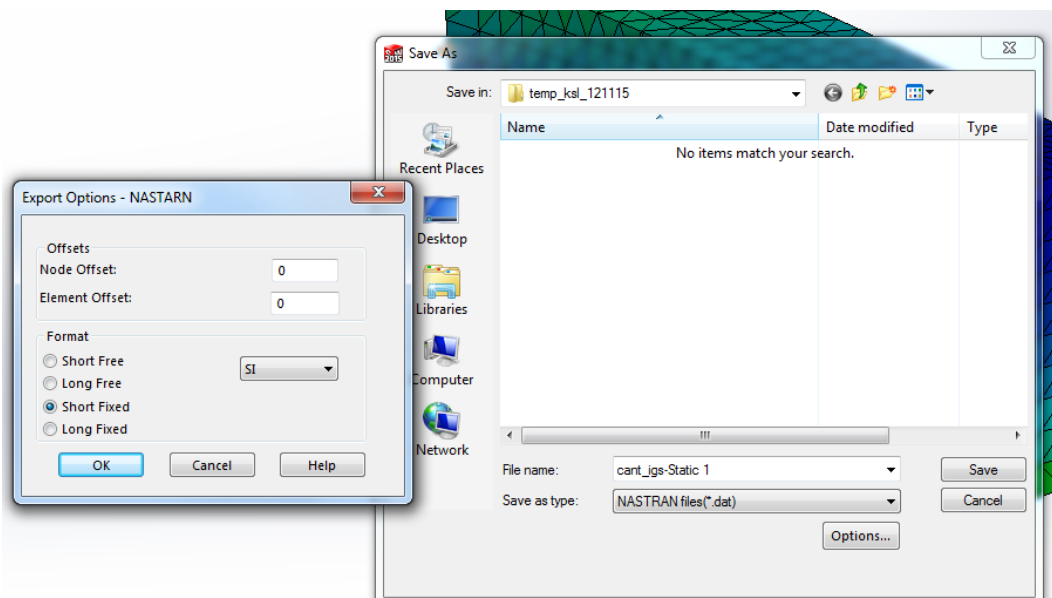


2- EXPORT YOUR DECK.

We are now going to export a NASTRAN deck of the simulation file (note that a nastran deck has the file extension *.blk or *.dat)

SIMULATION>EXPORT

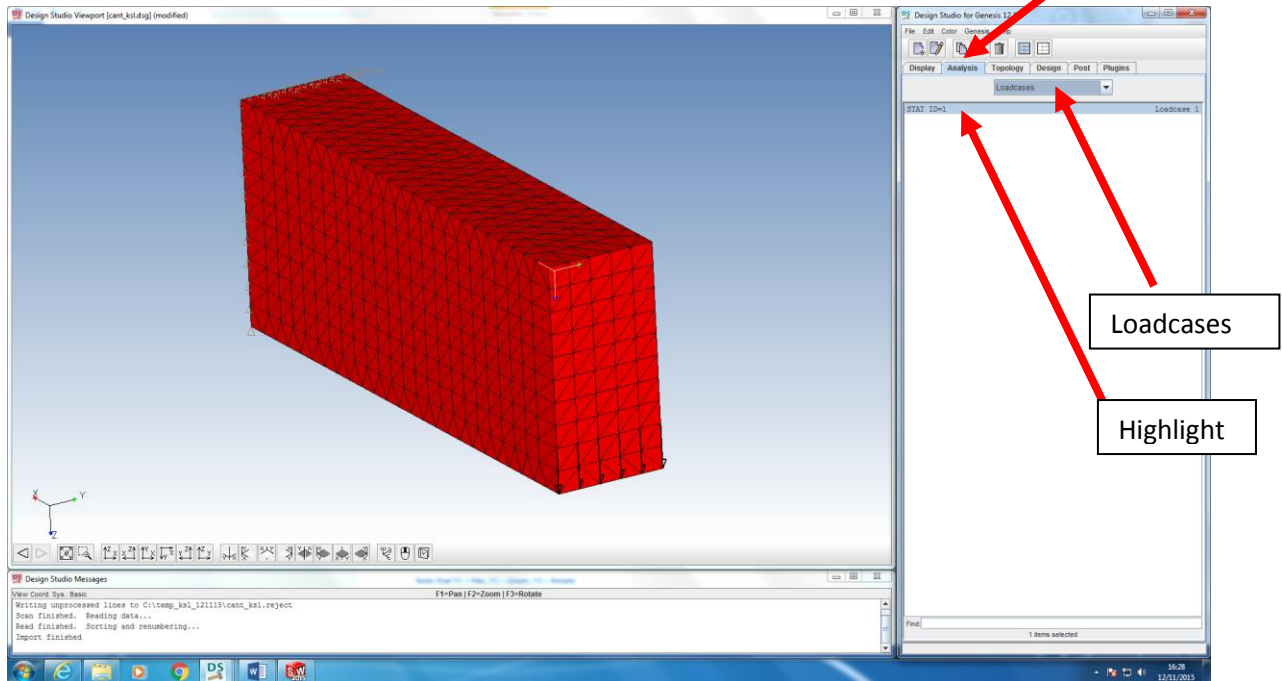
Choose to save as type NASTRAN and under options set this to SI units and short fixed format.



Now open and examine the file you have just written in “Wordpad”

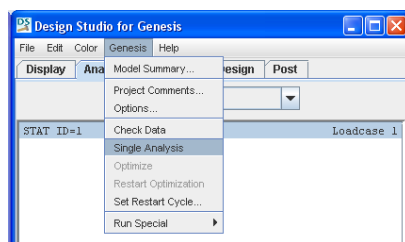
3 – IMPORT YOUR DECK INTO DESIGN STUDIO

FILE>IMPORT>IMPORT DATA, now check the loadcase

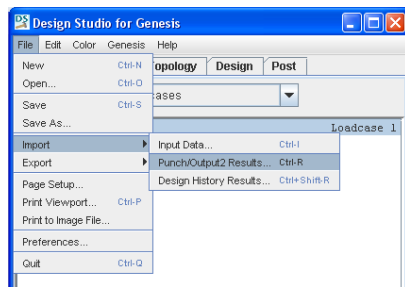


Note that F1 = Pan, F2 = Zoom, F3 = Rotate

4- RUN THE ORIGINAL MODEL IN GENESIS



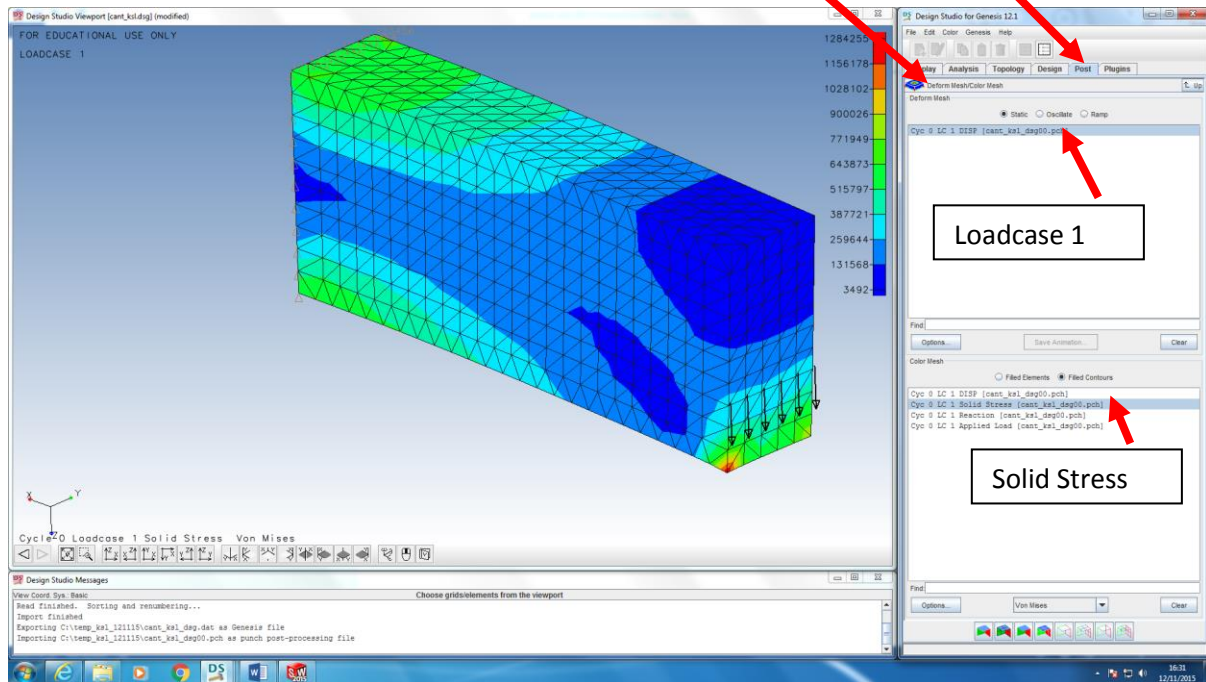
5 – IMPORT THE RESULTS FROM THE *.pch FILE



Deform mesh

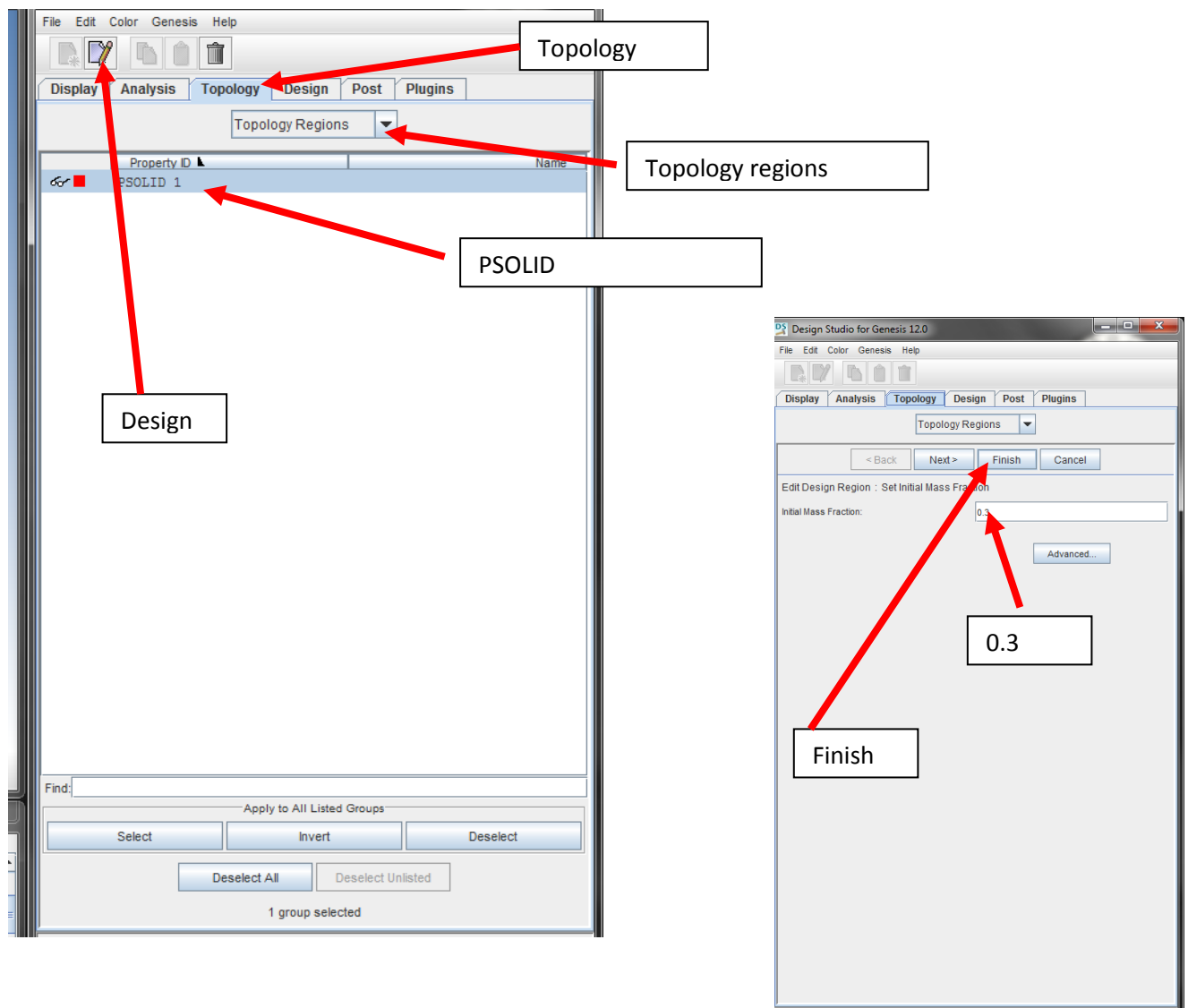
Post

6 – ACCESS THE RESULTS

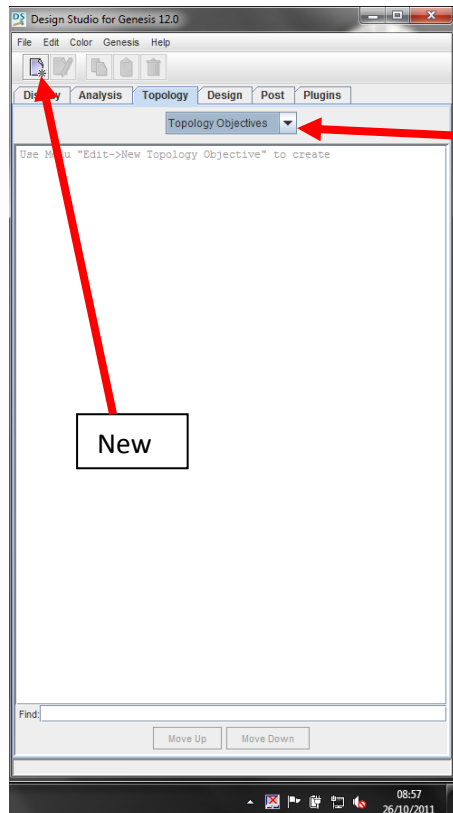


Now that we have assessed the baseline results we will do the topology run.

7 – DEFINE OUR DESIGNABLE MATERIAL

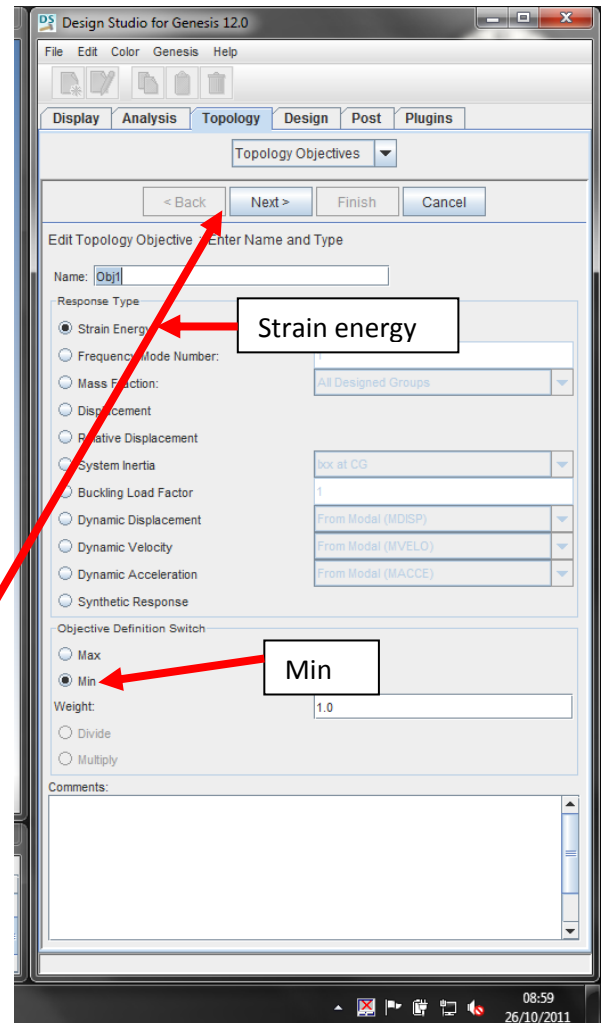


8 – DEFINE OUR OBJECTIVE – minimise strain energy



Topology objective

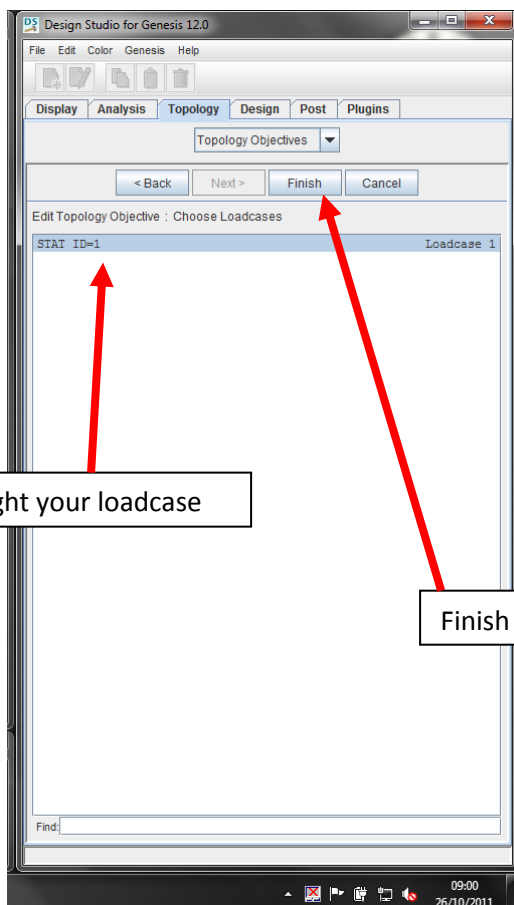
New



Strain energy

Min

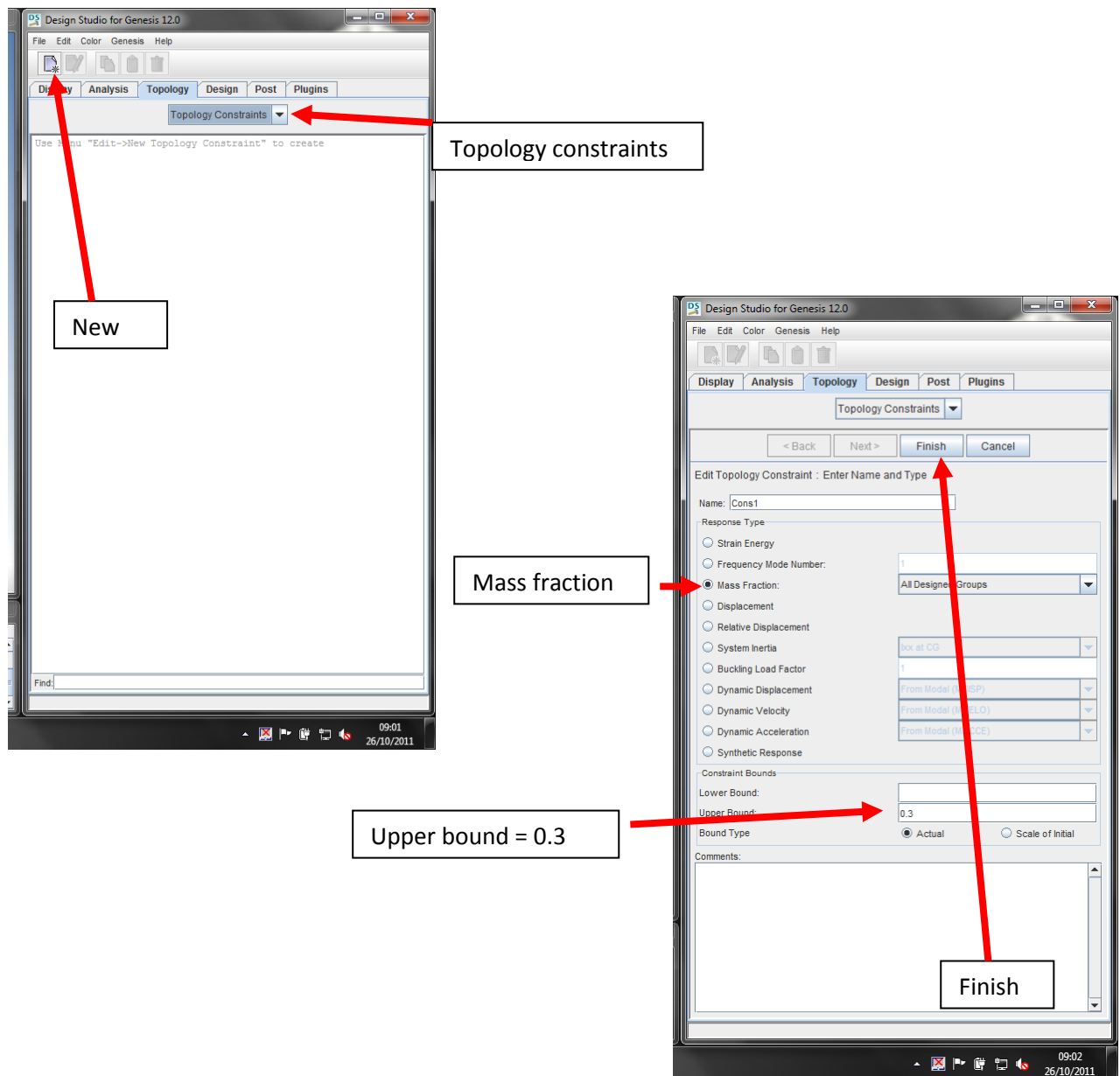
Next



Highlight your loadcase

Finish

9 – DEFINE OUR CONSTRAINT – we only want to use 30% of the material

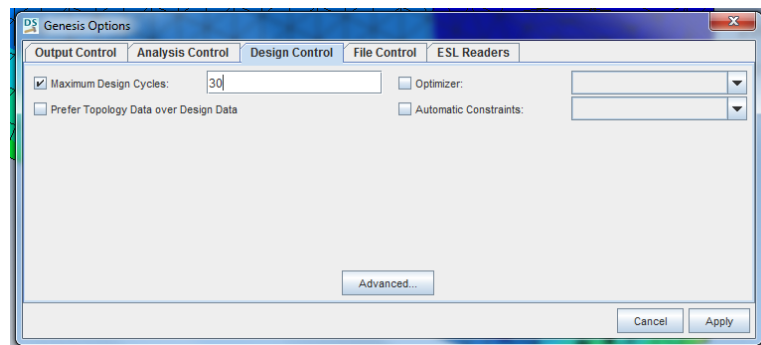


We have now defined the following

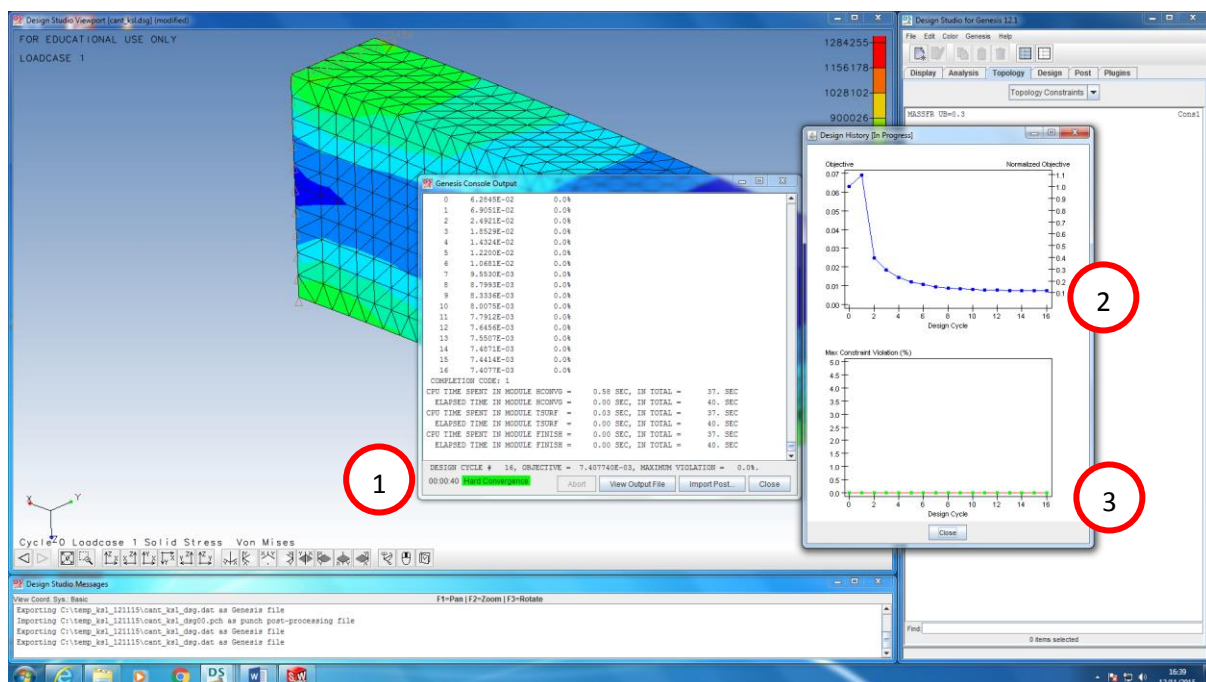
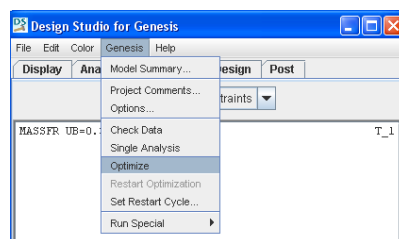
- We want to do a TOPOLOGY STUDY
- OUR DESIGNABLE REGION IS PSOLID
- INITIALLY WE WILL USE 30% OF THE AVAILABLE MATERIAL
- OUR OBJECTIVE IS TO MINIMISE STRAIN ENERGY FOR OUR LOADCASE
- OUR CONSTRAINT IS THAT WE MUST ONLY USE 30% OF THE AVAILABLE MASS

10 – before we optimise this structure we must increase the maximum number of iterations that GENESIS can do.

11 – GENESIS>OPTIONS>DESIGN CONTROL> set this to 30



12 – OPTIMISE THE STRUCTURE

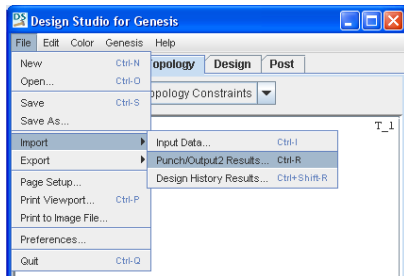


You should see something like the above. Note the following

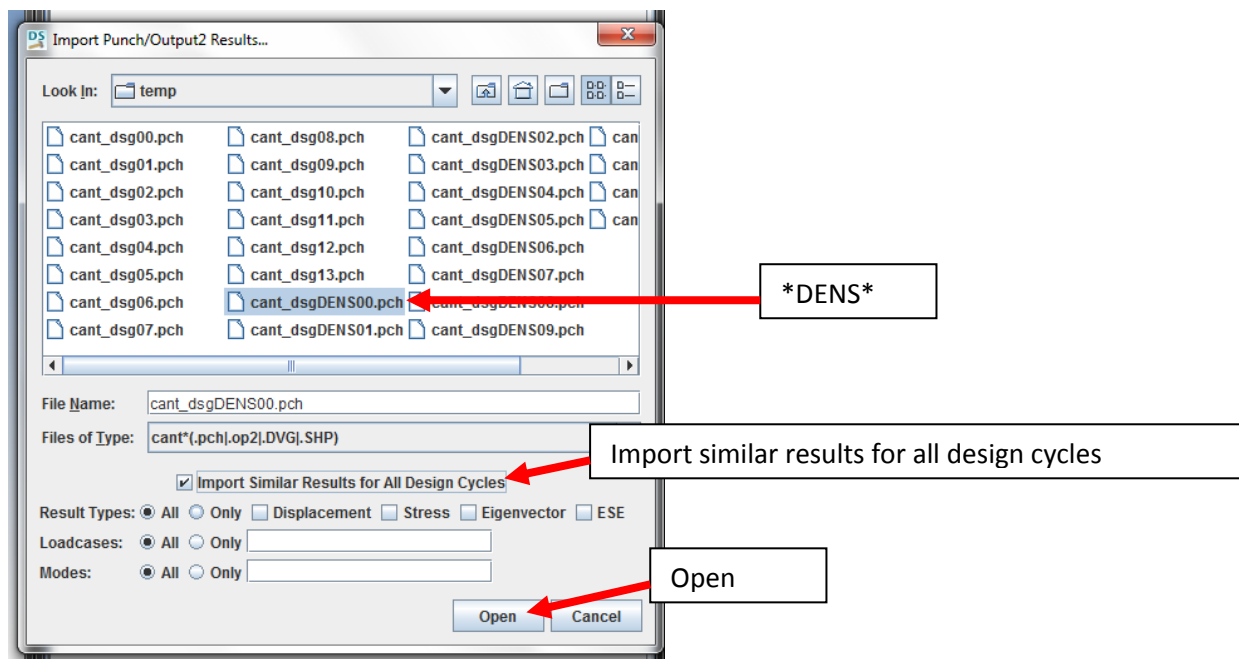
1. Genesis converged on a solution in 16 design cycles.
2. The normalised objective has fallen from 1 to about 0.1 – i.e. the structure with the mass distributed as per iteration 13 has about 10% of the strain energy compared to the baseline iteration when all elements had 30% of the material properties (the initial mass fraction)
3. The constraint is not being violated – i.e. we are only using 30% of that material as defined in the constraint.

CLOSE THE ABOVE WINDOWS

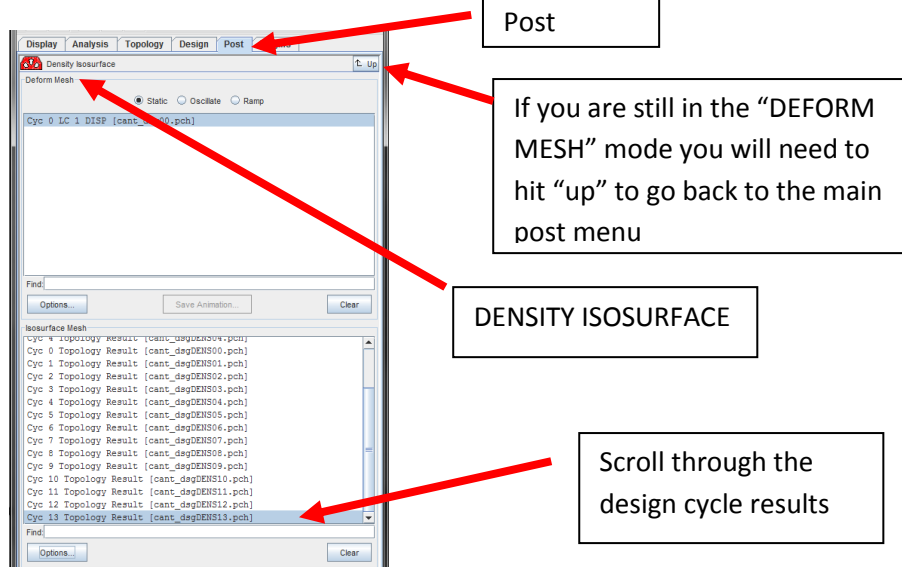
13 – IMPORT THE RESULTS



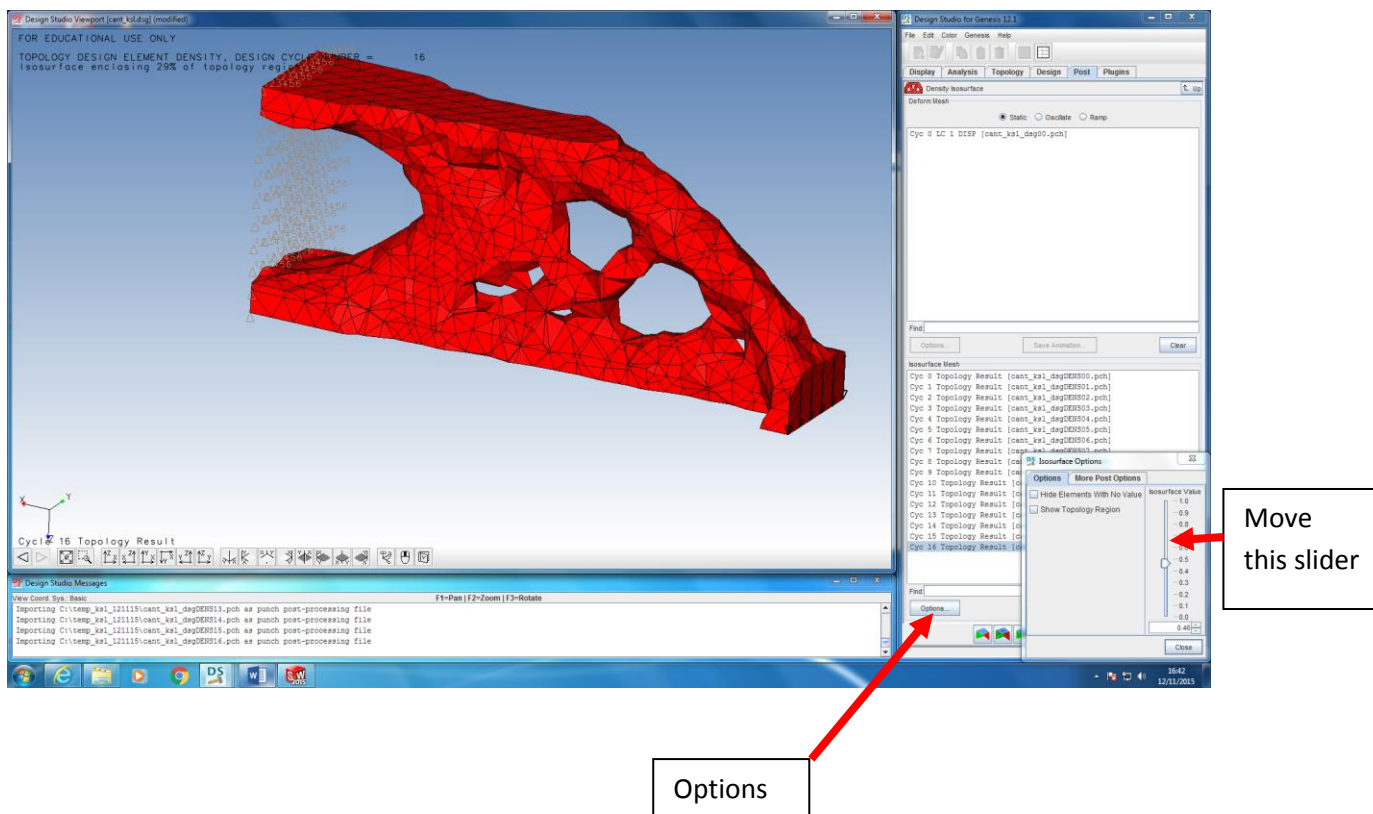
Notice that GENESIS has generated a *DENS* file for each design cycle which shows the distribution of the material. Import all of these;



14 – VIEW THE DENSITY ISOSURFACES

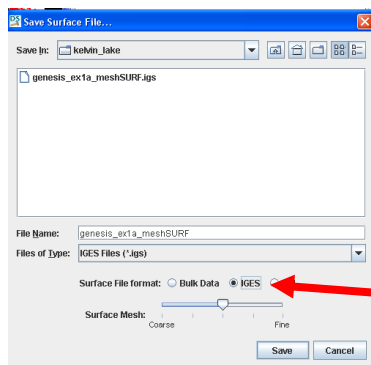
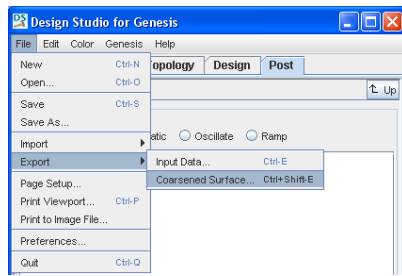


Go into options to see where the most important material is



You should now get some idea as to the most efficient structure for this component

15 – WE ARE NOW GOING TO OUTPUT A COARSENED SURFACE THAT WE COULD GIVE TO OUR DESIGNER TO DESCRIBE THE MAJOR LOADPATHS OF THIS COMPONENT OR WE COULD USE TO GENERATE ANOTHER FE MODEL FROM



IGES

This iges file can then be read into a cad package such as Solidworks and the design would then have to be INTREPRETED from the results.

