

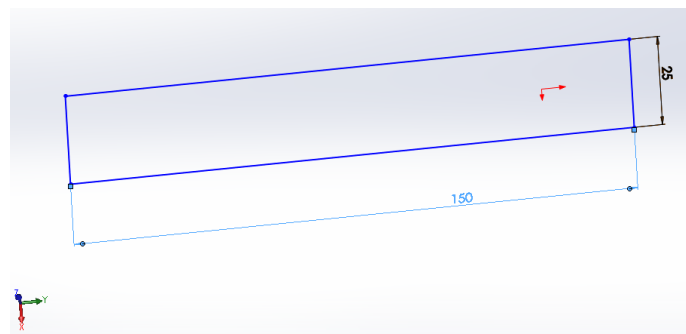
Defining and Optimising Composites

During this exercise we will set up a 3 point bending test with a simple flat plate in Solidworks. We will then export this to GENESIS and alter the material properties to a composite plate. We will then optimise the layer thicknesses and the angles of the fibres within those layers to achieve a certain stiffness whilst minimising the mass.

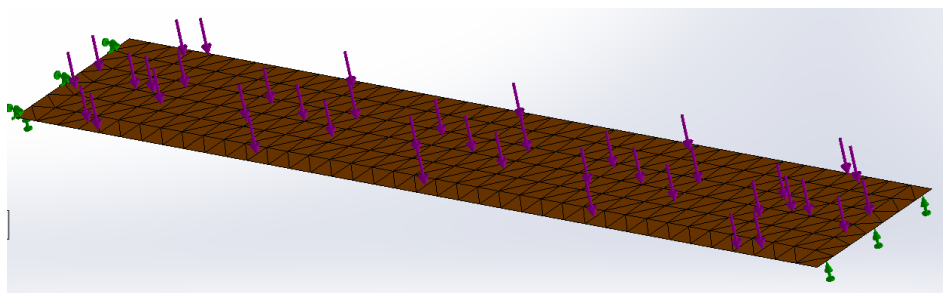
SETTING UP THE BASELINE 3 POINT BEND TEST IN SOLIDWORKS

Initially we will set up a 150mm x 25mm steel plate that is 3mm thick. We will put a total force of 500N as a udl across the plate. This will be the baseline product and the performance that we must match with the composite alternative.

Draw the following sketch in solidworks on the xy plane such that the long axis of the plate is in the y-direction.

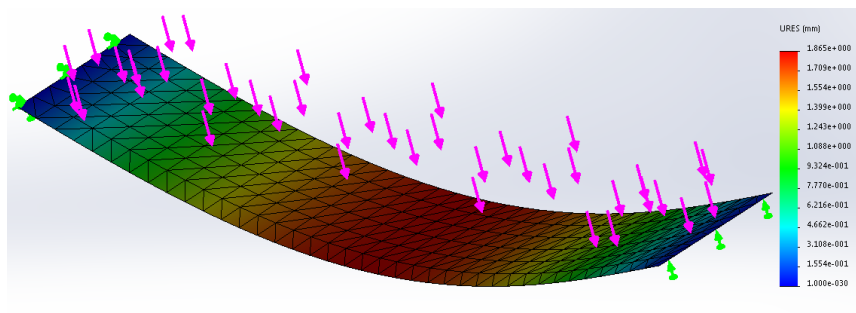


Now use INSERT>SURFACE>PLANAR to create your plane and then set up the following static model.

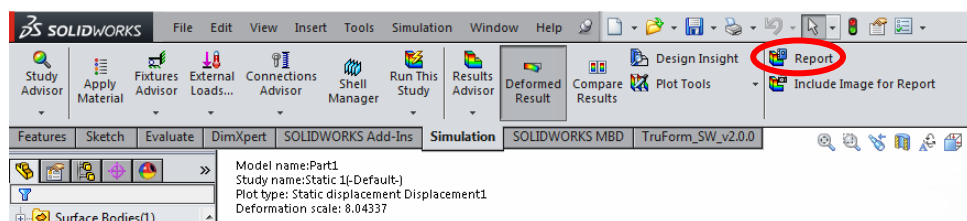


Create the above model with which models a 3mm steel plate with 4mm first order elements. The left hand end is free to rotate about the x axis only, the right hand end is free to rotate about the x axis and to translate in they axis. There is a total load of 500N applied to the face.

Your results should look as follows



As you can see the maximum displacement is 1.9mm. Now, generate a report to see the details of your model.



Accept the defaults and press PUBLISH. Check the model is as you are expecting. You should also see that the plate is 0.087kg or 87g. Thus we can calculate the stiffness to weight ratio of this part and summarise this as follows

Steel Plate

- Stiffness, $k = f/x = 500/1.9 = 263 \text{ N/mm}$
- Stiffness/weight = $k/(mg) = 263/(0.087 \times 9.81) = 308 \text{ N}^2/\text{mm}$

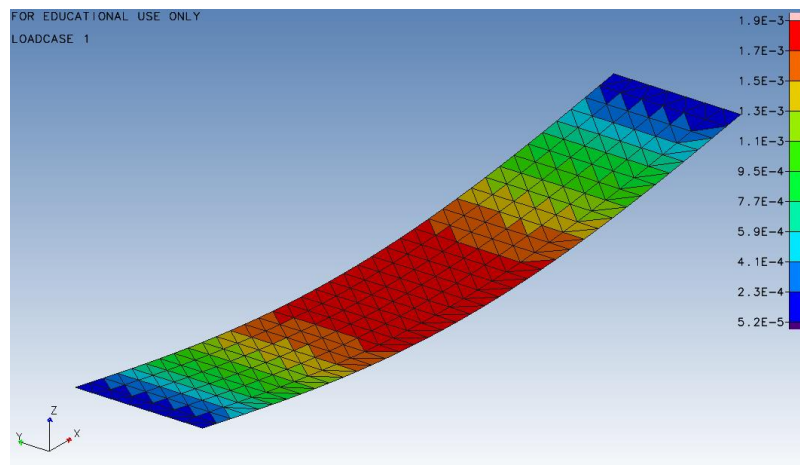
This will be our target to exceed with our optimised plate.

Create a folder called “baseline” and export this model as a Nastran file in SI units to this folder.

SIMULATION>EXPORT>set the file type to Nastran and call the file “Steel_plate.dat”

ASSESS THE STEEL PLATE IN DESIGN STUDIO

Now read this file into design studio and carryout a single analysis to check that you are getting the same results



You should see 1.9e-3m.

ASSESS THE BASELINE COMPOSITE PLATE

Now create a folder called “composite_baseline” and then save this model as “composite_baseline.dsg”

This part is to be made from a composite material consisting plys of a unidirectional composite. Initially this plate will be made up pre-preg in 5 orientations. Each orientation is to have 5 plys that are all 0.2mm thick and are orientated at -90, -45, 0, 45 and 90 degrees (zero degrees running down the length of the plate). The material properties of the unidirectional composite are shown below.

Young's Modulus in the longitudinal direction (E_1):	132 GPa
Young's Modulus in the lateral direction (E_2):	10.8 GPa
Poissons Ratio (Nu_12):	0.24
Inplane Shear Modulus (G_12):	5.65 GPa
Transverse Shear Modulus (1-Z plane)(G_1z):	5.65 GPa
Transverse Shear Modulus (2-Z plane)(G_2z):	5.65 GPa
Mass Density (Rho):	1540 kg/m ³
Allowable longitudinal stress in tension (Xt):	1513 MPa
Allowable longitudinal stress in compression (Xc):	1000 MPa
Allowable transverse stress in tension (Yt):	43.4 MPa
Allowable transverse stress in compression (Yc):	43.4 MPa
Allowable inplane shear stress (S):	43.4 MPa

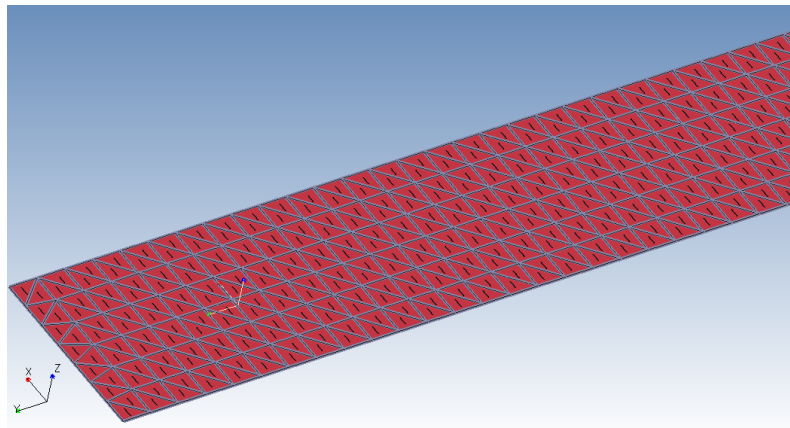
Define your material by ANALYSIS>MATERIALS>NEW>MAT8)call your material “unidirectional”>NEXT and enter the above values>FINISH

SAVE

Because our material has different material properties in different directions we need to orientate our elements relative to a vector which the material property uses.

ANALYSIS>ELEMENTS>SELECT ALL>GENERATE THETA VECTORS

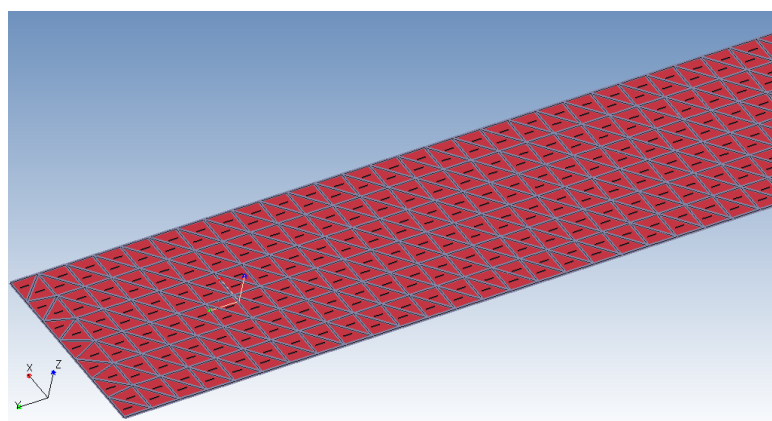
This shows the orientation of the E1 direction for each element as follows;



Note that by default the vectors for the elements align in the global x direction, this corresponds to 0 degrees. However, we have assumed that 0 degrees is along the y axis, therefore we must align the elements in the y direction.

ELEMENTS>SELECT ALL>MODIFY>CHANGE 2D ELEMENT'S THETA>NEXT>choose "use vector">pick two grids off of the screen to define the vector that you want to be 0 degrees for these elements (i.e. the y-axis)>NEXT>FINISH.

Now view the element vectors and you should see the following



we are now going to make up our laminate which consists of the unidirectional material that we have just defined.

ANALYSIS>GROUP PROPERTIES>NEW call you property 'laminate'

Choose PCOMP/PCOMPG as the type>Choose the colour you want your mesh to be>NEXT>Choose unidirectional as the material for layer 1>The next field is the thickness change this from 1 to 1e-3 (5 plies at 0.2mm)>The next field is the angle, change this to -90 degrees>Push the + button to add another layer.

Compile your composite as shown below;

Layer 1, Unidirectional, thickness 1mm, orientation -90 degrees

Layer 2, Unidirectional, thickness 1mm, orientation -45 degrees

Layer 3, Unidirectional, thickness 1mm, orientation 0 degrees

Layer 4, Unidirectional, thickness 1mm, orientation 45 degrees

Layer 5, Unidirectional, thickness 1mm, orientation 90 degrees

Our plate is thus made up of 25 plies (5 at -90, 5 at -45, 5 at 0, 5 at 45 and 5 at 90) with a total thickness on 5mm.

FINISH

SAVE

Now that our new PCOMP has been defined we need to move the elements from the PSHELL that we defined in Solidworks to the PCOMP that we have just defined.

ANALYSIS>ELEMENTS>SELECT ALL>MODIFY>CHANGE ELEMENTS GROUP>NEXT> select the PCOMP that you have just created>FINISH

We can now delete the empty group that we are no longer using.

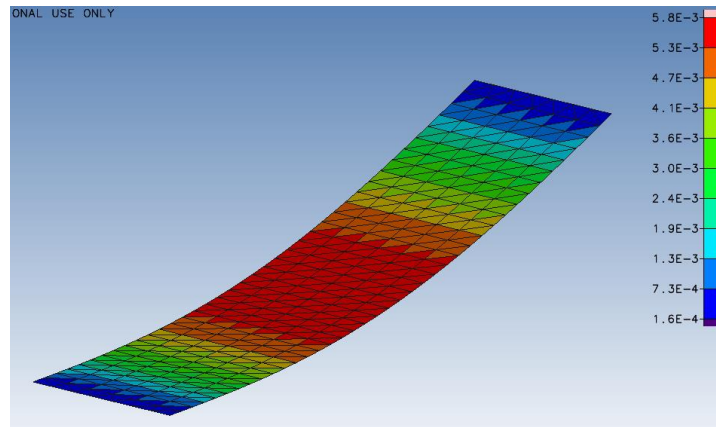
DISPLAY>MANAGE GROUPS>pick the empty PSHELL>DELETE

SAVE

We can now assess this composite structure and compare it to the original steel structure

GENESIS>SINGLE ANALYSIS

Look at your results you should see the following



As we can see displacement is 5.8mm. Now check the mass

ANALYSIS>ELEMENTS>SELECT ALL>CALCULATE MASS/VOLUME

You should see in the information window that the total mass is 0.029kg, i.e. 29g

So to summarise we now have

RUN	MATERIAL	THICKNESS	DEFLECTION	MASS	LOAD	STIFFNESS	STIFFNESS/weight
Steel baseline	Steel	3mm	1.9mm	97g	500N	263 N/mm	308 N ² /mm
Composite baseline	Composite	5mm	5.8mm	29g	500N	86 N/mm	297 N ² /mm

As we can see above in terms of stiffness/weight the composite part has NO advantage over the steel part.

We are now going to optimise the orientation of our composite layers and the thickness of these layers to achieve the same deflection as the steel structure at a reduced weight.

OPTIMISING YOUR COMPOSITE PART

Create another folder called OPTIMISED

Save your model as “optimised_composite.dsg” in this new folder

First we will set up our objective

DESIGN>OBJECTIVES>NEW>MASS>MIN>FINISH

Next we will set up our constraint

DESIGN>CONSTRAINTS>NEW>DISPLACEMENT>upper bound = 1.9e-3>NEXT>select all>TRANSLATION MAGITUDE>NEXT>highlight your loadcase>FINISH

SAVE

We are now going to set up our design variables.

We are going to set this up so that the thickness of each orientation can be between 0.2mm and 3mm.

DESIGN>DESIGN VARIABLES>NEW>name your variable L1THK as it will represent the variable for the thickness for plys in the first orientation>NEXT>INITIAL VALUE=1e-3>LOWER VALUE = 0.2e-3>UPPER VALUE = 3e-3>FINISH

Repeat the above step to define the following variables

Variable Name	Initial Value	Lower bound	Upper Bound	Description
L1THK	1mm	0.2mm	3mm	thickness of layer 1
L2THK	1mm	0.2mm	3mm	thickness of layer 2
L3THK	1mm	0.2mm	3mm	thickness of layer 3
L4THK	1mm	0.2mm	3mm	thickness of layer 4
L5THK	1mm	0.2mm	3mm	thickness of layer 5

DONE

We are now going to attach the above design variables to the PCOMP that we created

DESIGN>SIZING>highlight your PCOMP>modify

Point the thickness of layer one to L1THK

Point the other layers to their appropriate design variables

FINISH

SAVE

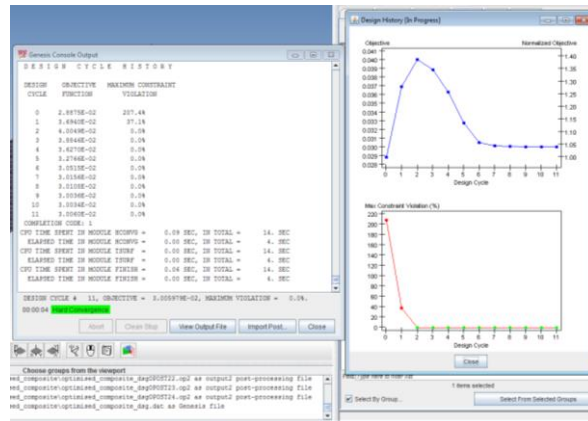
Before we optimise this we are going to increase the maximum number of cycles allowed to 30 so that Design Studio converges on a solution.

GENESIS>OPTIONS>DESIGN CONTROL> set the maximum design cycles to 30

Now optimise your structure

GENESIS>OPTIMISE

You should see something like the following



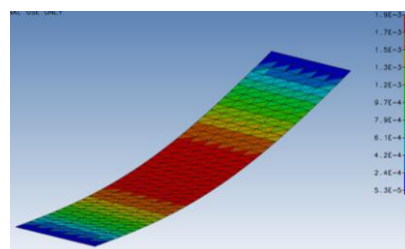
As you can see in design cycle 0 (our baseline) we are violating our constraint by $(5.8/1.9 - 1) \times 100 = 205\%$. By design cycle 11 this is 0 – i.e. we are meeting our displacement constraint of a maximum of 1.9mm

In terms of our objective you can see that this was initially 29g at cycle 0 and by cycle 11 it has gone up to about 30g.

IMPORT POST>hold down shift (note that there are now multiple files as there are files for the stress/displacement of every design cycle and information regarding the variable for every design cycle).

First confirm the displacement for the last design cycle.

POST>DEFORM/COLOR MESH> deform your mesh using the results from the last cycle and colour your mesh using the results from the last cycle. You should see something like the following;

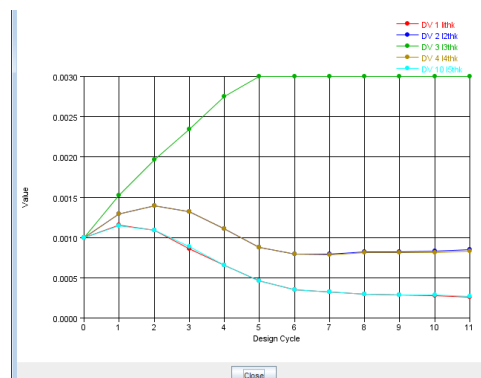


So genesis has added thickness to each orientation to meet the displacement target and minimise the mass. In order to see how genesis has achieved this we can view how the variables have changed.

FILE>IMPORT>DESIGN HISTORY RESULTS – read the file in.

Now view this

POST>DESIGN HISTORY PLOT>NEW>click on the plus sign to add a curve>NEXT>choose your file>NEXT>hold down shift to pick all of your thickness variables>NEXT>FINISH



As you can see all the variables started at 1mm (i.e. 5 plies). The plies at 0 degrees have increased to 3mm (the maximum) as you would expect, the plies at -90 and -45 have reduced to a minimum and the plies at +/-45 degrees are approximately 0.9mm to achieve the stiffness target.

The results file is a text file called “optimised_composite_dsg.HIS” which we can simply open in a text editor to get more details.

Open this file in wordpad and you should see the following for design cycle 0 (i.e. our baseline). We can see that the objective (mass) is 29g and the violation is 207%. We can also see that all of our variables are set to 1mm.

```
DESIGN CYCLE #    0, OBJECTIVE = 2.887500E-02, MAXIMUM VIOLATION = 207.4%.
DESIGN VARIABLE #      1 = 1.000000E-03    lithk
DESIGN VARIABLE #      2 = 1.000000E-03    12thk
DESIGN VARIABLE #      3 = 1.000000E-03    13thk
DESIGN VARIABLE #      4 = 1.000000E-03    14thk
DESIGN VARIABLE #     10 = 1.000000E-03    15thk
```

Now look at the last design cycle. We can see that the objective has increased to 30g and the constraint is now longer being violated. We can also see the design variables have changed to achieve this.

```
DESIGN CYCLE #   11, OBJECTIVE = 3.005979E-02, MAXIMUM VIOLATION = 0.0%.
DESIGN VARIABLE #      1 = 2.622527E-04    lithk
DESIGN VARIABLE #      2 = 8.508001E-04    12thk
DESIGN VARIABLE #      3 = 2.996601E-03    13thk
DESIGN VARIABLE #      4 = 8.289273E-04    14thk
DESIGN VARIABLE #     10 = 2.665779E-04    15thk
```

Our total thickness is thus $0.26+0.85+3+0.83+0.27 = 5.2\text{mm}$

To summarise

RUN	MATERIAL	THICKNESS	DEFLECTION	MASS	LOAD	STIFFNESS	STIFFNESS/weight
Steel baseline	Steel	3mm	1.9mm	97g	500N	263 N/mm	308 N ² /mm
Composite baseline	Composite	5mm	5.8mm	29g	500N	86 N/mm	297 N ² /mm
Optimised composite	Composite	5.2mm	1.9mm	30g	500N	263 N/mm	894 N ² /mm

So in terms of our stiffness to weight ratio our part is now roughly three times as efficient as the original part.

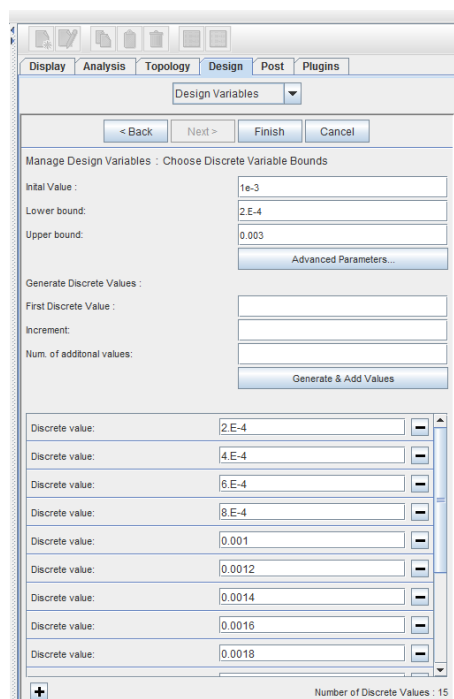
HOWEVER, we must now take into account manufacturing this part, we have now got the following lay-up if each ply is 0.2mm

DESIGN VARIABLE # 1 = 2.622527E-04 lithk = 1.3 plys at -90
 DESIGN VARIABLE # 2 = 8.508001E-04 l2thk = 4.25 plys at -45
 DESIGN VARIABLE # 3 = 2.996601E-03 l3thk = 15 plys at 0
 DESIGN VARIABLE # 4 = 8.289273E-04 l4thk = 4.15 plys at 45
 DESIGN VARIABLE # 10 = 2.665779E-04 l5thk = 1.35 plys at 90

Obviously we can not have 1.3 plys etc so instead of our variables being any value between 0.2mm and 3mm we must make them discrete values. i.e. we can choose 0.2, 0.4, 0.6, 0.8, 1.0, 1.2, 1.4, 1.6 1.8, 2.0, 2.2, 2.4, 2.6, 2.8 or 3mm.

Go into your design variables and choose modify and then discrete design variables> NEXT

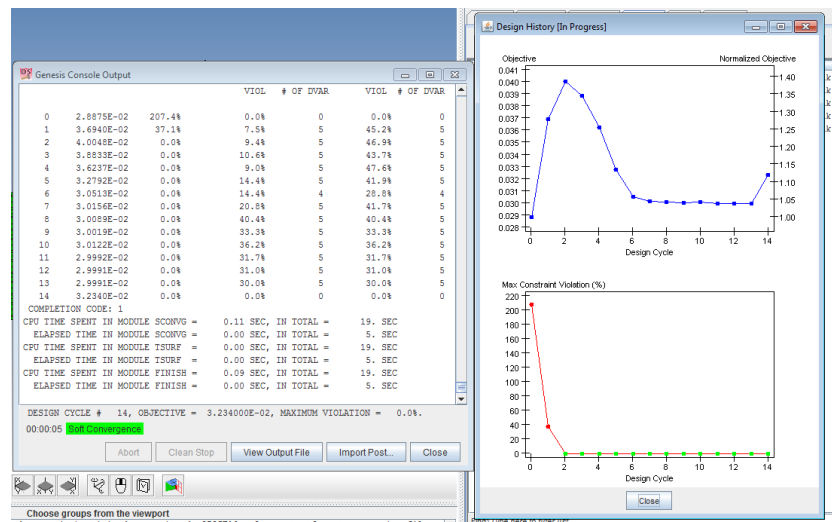
Set your first discrete value to 2e-4, your increment is 2e-4 and you need to add 14 additional values. GENERATE AND ADD VALUES. Note that you must remove the first discrete value that design studio has generated. You should now see the following;



FINISH

Repeat the above for all of your design variables and then re-optimize this.

We now see the following



View the displacement. Note that this is now 1.7mm. Still meeting our constraint of a maximum of 1.9mm but the part is now actually slightly stiffer as discrete plies must be used.

By looking in the history file you should see the following

```
DESIGN CYCLE #    14, OBJECTIVE =  3.234000E-02, MAXIMUM VIOLATION =  0.0%.
DESIGN VARIABLE #      1 =  4.000000E-04    lithk
DESIGN VARIABLE #     10 =  1.000000E-03    12thk
DESIGN VARIABLE #     11 =  3.000000E-03    13thk
DESIGN VARIABLE #     12 =  8.000000E-04    14thk
DESIGN VARIABLE #     13 =  4.000000E-04    15thk
```

The objective has increased to 32g however the suggested ply lay-up is now as follows

-90 degrees = 0.4mm = 2 plies
 -45 degrees = 1mm = 5 plies
 0 degrees = 3mm = 15 plies
 45 degrees = 0.8mm = 4 plies
 90 degrees = 0.4mm = 2 plies
TOTAL = 28 plies = 5.6mm

To summarise

RUN	MATERIAL	THICKNESS	DEFLECTION	MASS	LOAD	STIFFNESS	STIFFNESS/weight
Steel baseline	Steel	3mm	1.9mm	97g	500N	263 N/mm	308 N ² /mm
Composite baseline	Composite	5mm	5.8mm	29g	500N	86 N/mm	297 N ² /mm
Optimised composite	Composite	5.2mm	1.9mm	30g	500N	263 N/mm	894 N ² /mm
Optimised composite (discrete)	Composite	5.6mm	1.9mm	32g	500N	263 N/mm	838 N ² /mm

Hence, the final part is not as efficient as the theoretical part, however we can manufacture it and it is still nearly 3 times as efficient in terms of stiffness to weight ratio when compared to the original steel part.