SPEIT-Shanghai Jiaotong University

ME6 - VA 2018

Project Report

Optimization of Composite Beam

Students:

CAO Anbo 操安博
CHEN Yujiong 陈煜炯
LI Tianyu 李天宇
YU Yue 俞越

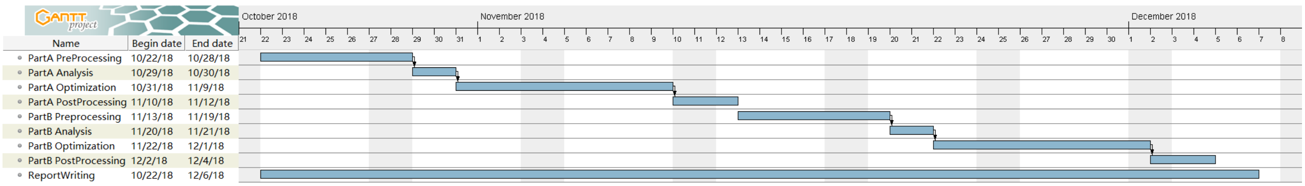
December 14, 2018

1 Introduction

A panel made of two stiff, strong skins separated by a lightweight core is called a sandwich panel. The core, which usually has relatively low density, serves to increase the moment of inertia. The load is mainly carried by the facings, also called the skins. By combining the core and the facings, this structure is able to carry load efficiently with a low weight. Such structure has been proved to be prominent in all kinds of applications.

In this project, two sandwich panels are studied. In both the structures, the size of the panels is 440mm*35mm *34mm. The core is made of polypropylene honeycomb and its thickness is 30mm. The thickness of the two skins is 2mm respectively. The sandwich plate is simply supported at the two ends and a 450N punctual load P is applied to the center of the panels. The optimization objective is to minimize the weight under several constraints. In part A, the skins are made of glass fibers (T800/M300)/polyester resin composite. The fibers are short, therefore the material is isotropic. In part B, the material of skins has the same component but the fibers are long. Consequently the material is anisotropic. The skin involves five layers which are arranged in symmetric orientation and thickness. The orientation is between -90 and +90, with the smallest interval equalling to 15.

The Gantt chart of our project is shown as below. We work on this project from 10/22 to 12/6. Yujiong is in charge of the static analysis, Anbo is responsible for the optimization of the structures, Tianyu solves the theoretic problem while Yue plays a major role in the redaction of this report.



Contents

2 Isotropic skin sandwich beam size optimization

2.1 Finite element analysis

2.1.1 Configuration

To make a finite element analysis of the given structure, we follow the steps below:

1) Enter X, Y, Z coordinates to create nodes. Link the nodes and create 4 lines to enclose a 440mm-long, 35mm-wide rectangle.

2) Create 2 `MAT1` type materials for the face sheet and the core layer with the following parameters:

Name	Value
Solver Keyword	MAT1
Name	skin
ID	1
Color	
Include	[Master Model]
Defined	☑
Card Image	MAT1
User Comments	Hide In Menu/Export
E	9162.0
G	2101.0
NU	0.3
RHO	1.44e-009
A	
TREF	
GE	
ST	321.0
SC	

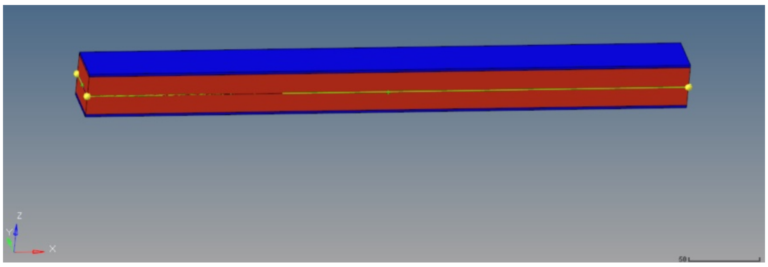
Name	Value
Solver Keyword	MAT1
Name	core
ID	2
Color	
Include	[Master Model]
Defined	☑
Card Image	MAT1
User Comments	Hide In Menu/Export
E	15.0
G	8.0
NU	0.4
RHO	8e-011
A	
TREF	
GE	
ST	
SC	1.3

Although the volume fraction of glass fiber in the skin material is known as 30%, we still need to calculate the density of this material according to that of glass fiber and polyester. However their density depends on many factors and it is not convincing to just choose one density for each and calculate. Finally we decide to choose the density of a 30% glass fiber reinforced PBT, a product of DuPont Company called Crastin LW9030 NC010 | (PBT+ASA)-GF30[1].

3) Create a new `PCOMPP` property with default parameters and assign it to the component.

4) Create a 2mm-thick skin ply, a 30mm-thick core ply and a 2mm-thick skin ply. The geometry is the rectangle created before, the material is skin, core and skin respectively. The orientation is 0.

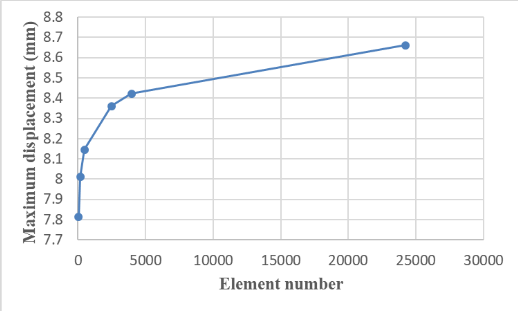
5) Create a laminate by selecting the 3 plies created. So far the structure should look like the figure below:

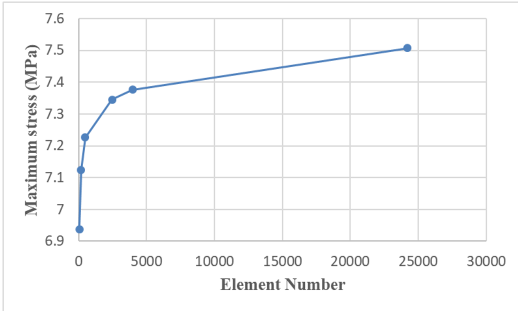


6) In order to verify the convergence of FEM calculation, several 2d meshes are created with different mesh sizes in the `skin` mesh window. The smaller the mesh size, the bigger the element number. The maximum displacement and the maximum stress are recorded for different element numbers to study the mesh convergence of the calculation. According to the results, the maximum displacement and the maximum stress tend to converge with the increase of element number. We can thus confirm the mesh convergence.

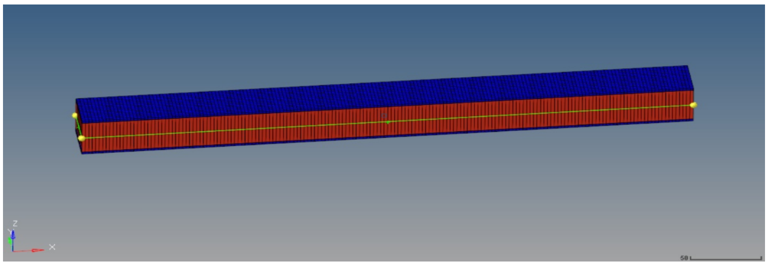
We finally choose 2.5mm as the mesh size to have a better trade-off between the calculation precision and the computing time.

[1]https://dupont.materialdatacenter.com/profiler/main/standard/main/ds/11754/4097

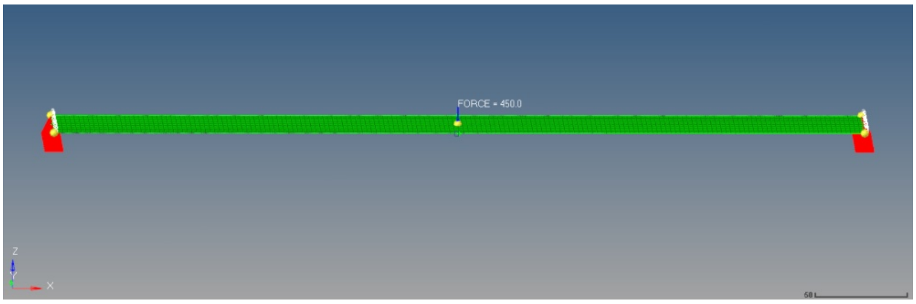




7) Realize the plies in order to apply the mesh to the composite geometry. The structure should look like the figure below.



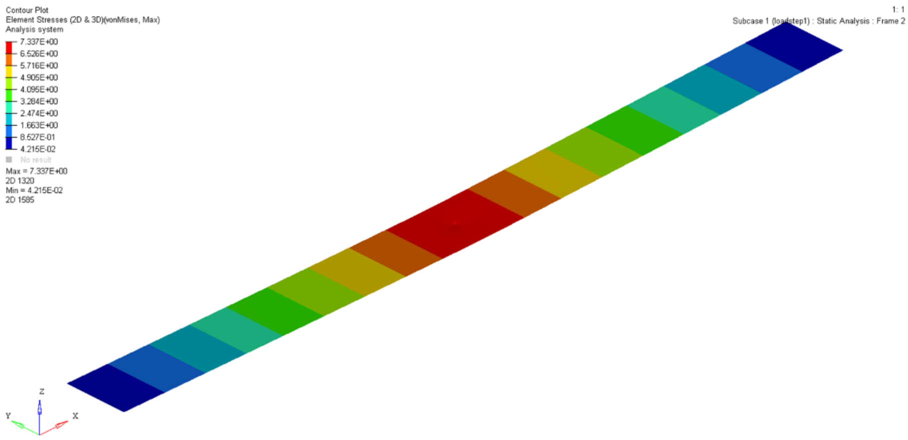
8) Apply a punctual force of 450N at the center of the component and constrain the 2 ends of the component in Z direction as the sandwich plate is simply supported.

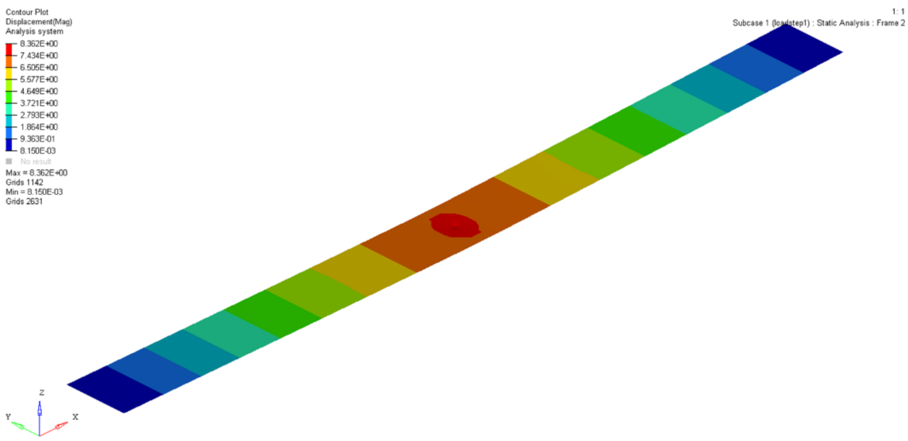


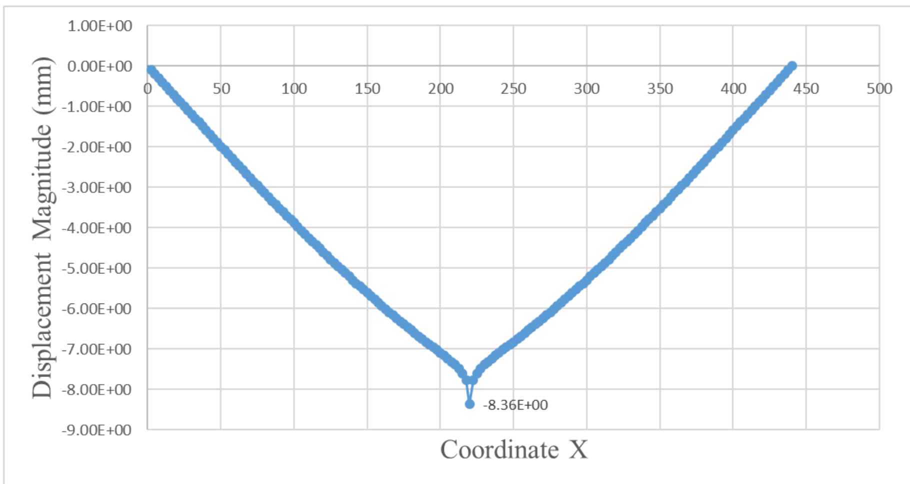
2.1.2 Results

The maximum displacement is 8.362mm at the center of the component, where we apply the punctual force. The maximum stress is 7.337MPa, also at the center of the component.

We then export the displacement of all nodes. By selecting the nodes with a Y coordinate close to 17.5, we get the the midline of the component. Draw the graph of displacement VS coordinate X for the midline. The maximum displacement is -8.36mm which corresponds to the previous plot of displacement. We observe that the displacement is almost linear along the midline. This is different from uniform beam whose displacement is more parabolic.







2.2 Theoretical deformation of the beam

2.2.1 Analysis from the first order beam theory

Since the short side length and the thickness of the structure are very close, we will use the assumption in the Euler-Bernoulli beam theory in the first place. The axial strain is assumed to vary linearly over the cross-section of the beam. Therefore, the axial stress in the sandwich beam is given by

$$\sigma_{xx}(x,z) = E(z)\varepsilon_{xx}(x,z) = -zE(z)\frac{\mathrm{d}^2 w^b}{\mathrm{d}x^2} \tag{1}$$

The bending moment in the beam is then given by

$$M(x) = \iint_{S(x)} z\ \sigma_{xx}\ \mathrm{d}z\,\mathrm{d}y = -\left(\iint_{S(x)} z^2 E(z)\ \mathrm{d}z\,\mathrm{d}y\right)\ \frac{\mathrm{d}^2 w^b}{\mathrm{d}x^2} := -D\ \frac{\mathrm{d}^2 w^b}{\mathrm{d}x^2} \tag{2}$$

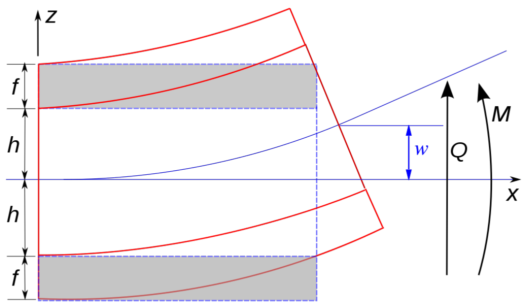


Figure 1: Bending of a sandwich beam without extra deformation due to core shear.

where D is called the flexural stiffness of the sandwich beam[2], which is equivalent to EI for a Bernoulli beam. In our case, its value is given by

$$D = E^f \int_b \int_{-h-f}^{-h} z^2 \,\mathrm{d}z \,\mathrm{d}y + E^c \int_b \int_{-h}^{h} z^2 \,\mathrm{d}z \,\mathrm{d}y + E^f \int_b \int_{h}^{h+f} z^2 \,\mathrm{d}z \,\mathrm{d}y \tag{3}$$

$$= \left(\frac{2}{3}E^f f^3 + \frac{2}{3}E^c h^3 + 2E^f fh(f+h)\right) b. \tag{4}$$

The two extremities being simply supported , i.e. $M(\pm\frac{L}{2}) = 0$, the profile of the bending moment under concentrated load at the center can be easily calculated from the balance of forces

$$Q_x = \frac{\mathrm{d}M}{\mathrm{d}x} \tag{5}$$

$$\frac{\mathrm{d}Q_x}{\mathrm{d}x} + P\delta(x) = 0 \tag{6}$$

where Q_x is the shear force. Therefore

$$M(x) = \frac{P}{2}(L/2 - |x|) \tag{7}$$

Then by integrating (2) twice we have

$$w^b(x) = \frac{PL^3}{48D}\left(2\left(\frac{x}{L}\right)^2\left(3 - 2\left|\frac{x}{L}\right|\right) - 1\right) \tag{8}$$

The maximum deflection is supposed to be

$$w^b_{\max} = \frac{PL^3}{48D} \tag{9}$$

which is a simple and classic result for a linear beam. However, since $G^c \ll E^f$, the **core shear** would be significant.

Nevertheless, we make the following assumptions in lieu of those in the Euler-Bernoulli theory:

- the transverse normal stiffness of the core is infinite, i.e., the core thickness in the z-direction does not change during bending
- the in-plane normal stiffness of the core is small compared to that of the facesheets, i.e., the core does not lengthen or compress in the x-direction
- the facesheets behave according to the Euler-Bernoulli assumptions, i.e., there is no xz-shear in the facesheets and the z-direction thickness of the facesheets does not change

The shear strain γ_{xz} on a core section is supposed to be constant. Therefore according to previous approximations, $\tau^c_{xz} = G^c\gamma^c_{xz}$ gives

$$(G^c A)_{eq}\frac{\mathrm{d}w^s}{\mathrm{d}x} = -Q_x \tag{10}$$

[2]http://www.mse.mtu.edu/ drjohn/my4150/sandwich/sp1.html

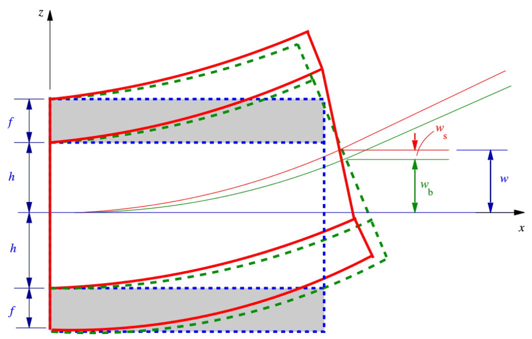


Figure 2: The total deflection is the sum of a bending part w^b and a shear part w^s

The integration of this relation gives us the profile of the deflection due to shear

$$w^s = \frac{P}{2(G^c A)_{eq}} \left(\frac{L}{2} - |x| \right) \tag{11}$$

where $(G^c A)_{eq} = b\dfrac{(2h+t)^2}{2h} G^c$.

Hence the maximum deflection is modified to

$$w_{\max} = \frac{PL^3}{48D} + \frac{PL}{4(G^c A)_{eq}} \tag{12}$$

2.2.2 Comparison

Using the previous relation and the given parameters, we can put the profile of deflection from the linear beam model alongside with the FEA results.

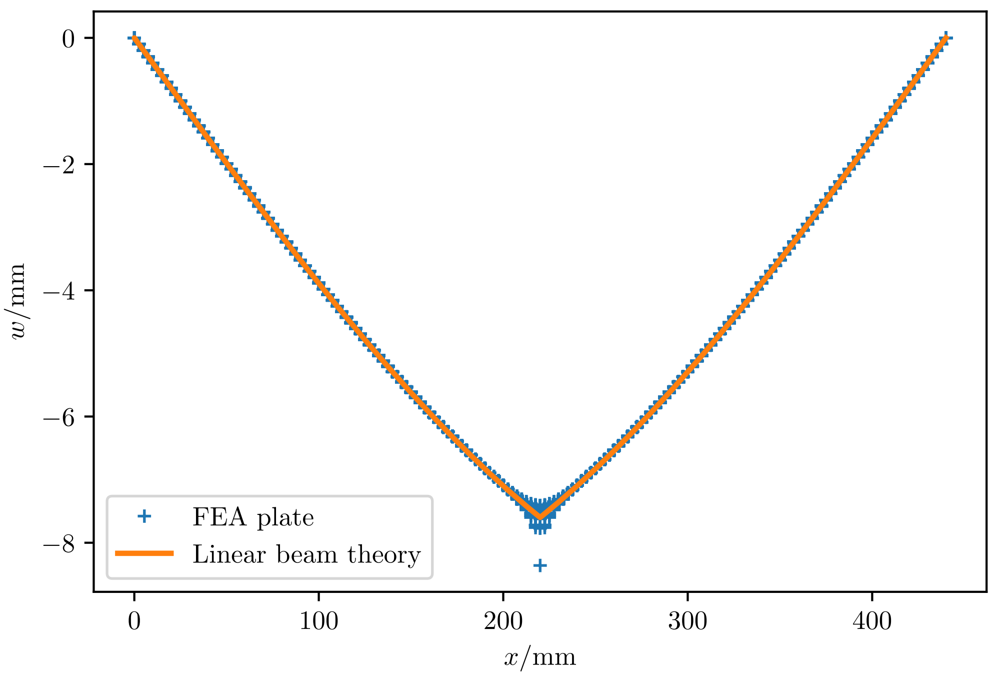


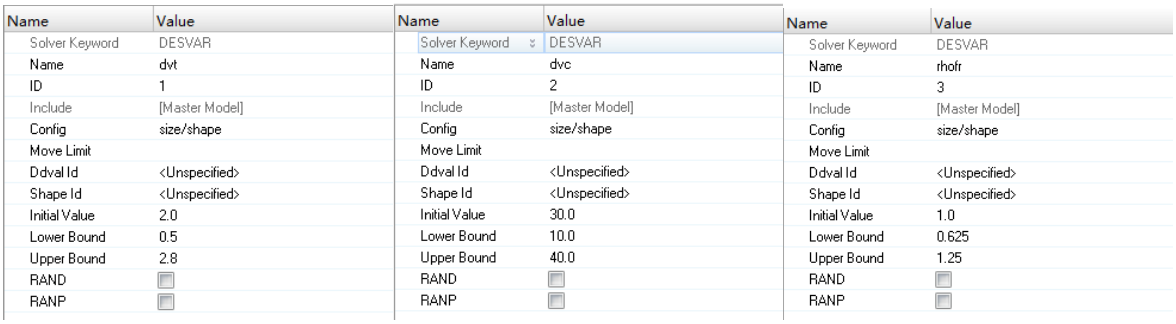
Figure 3: Comparison between the linear beam model and FEA result

The maximum deflection calculated from (12) is 7.60 mm, whereas the FEA gives 7.77 mm. Therefore this model gives the same approximation as the FEA. Furthermore, it shows that the deflection due to shear (5.18 mm) is much bigger than the deflection due to bending (2.42 mm).

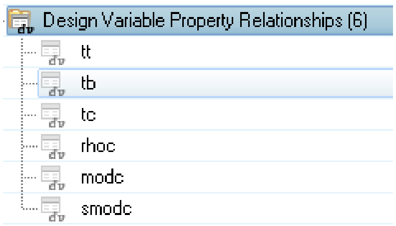
2.3 Optimization

In the previous step, we have created the material, property, ply, laminate. To do the optimization, we follow the steps below:

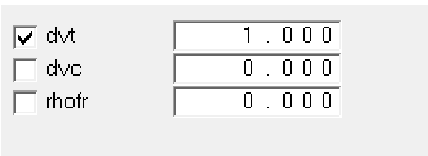
1) Create three design variables dvt (for the thickness of the facings), dvc (for the thickness of the core), and rhofr (for the mass density fraction of the core) with the initial value, upper bound and lower bound given in the demands.



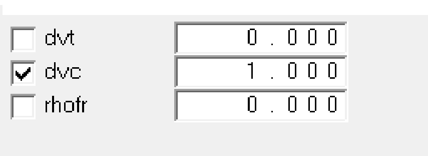
2) Create six properties that are linearly linked to the design variables.



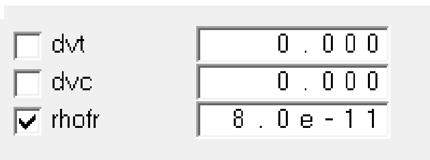
tt (thickness of the top face) and **tb** (thickness of the bottom face) are linked to **dvt**. **tc** (thickness of the core) is related to **dvc**. **rhoc** (mass density of core), **modc** (Young's Modulus of core) and **smodc** (Shear Modulus of core) are proportionate to **rhofr**. Their relationships are set with the following coefficient (c_0=0 for all of them):



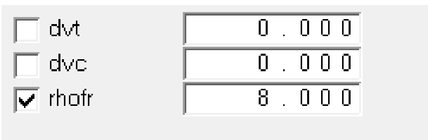
(a) Generic relationship for tt and tb



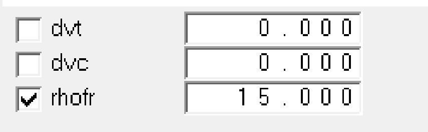
(b) Generic relationship for tc



(c) Generic relationship for rhoc



(d) Generic relationship for smodc



(e) Generic relationship for modc

3) Create three responses for stress, mass, and deflection. The response of mass is for the total structure. The response of the stress is for all the elements. And the response of the deflection is only for the node where the force is applied, which is also the node with the largest displacement. This is because the positive and negative displacement at other nodes may mislead the judgment on bound condition.

4) Create the constraints for the responses of stress and deflection. According to the criteria in the question, a maximum stress and a maximum displacement should be settled. We get from the question 1 that the maximum displacement is 8.362mm and the maximum stress is 7.337MPa. So the maximum displacement required here is 8.362*0.7=5.8527mm, and the maximum stress required is 5.1359MPa.

Name	Value
Solver Keyword	DRESP1
Name	stress
ID	1
Include	[Master Model]
Response Type	static stress
Property	ELEM
List Of Elements	2464 Elements
Region Identifier	
	von mises
	z1
Cluster	
COORD	☐
DREPORT	☐

Name	Value
Solver Keyword	DRESP1
Name	mass
ID	3
Include	[Master Model]
Response Type	mass
Property	PROP_TOTAL
Property Type	total
Region Identifier	
DREPORT	☐

Name	Value
Solver Keyword	DRESP1
Name	defle
ID	4
Include	[Master Model]
Response Type	static displacement
Property	PROP_TOTAL
List Of Nodes	1 Nodes
Region Identifier	
	total displacement
COORD	☐
DREPORT	☐

Name	Value
Solver Keyword	DCONSTR
Name	cs
ID	1
Include	[Master Model]
Lower Bound	
Upper Bound	5.1359
Response	(1) stress
List of Loadsteps	1 Loadsteps
PROB	

Name	Value
Solver Keyword	DCONSTR
Name	cd
ID	2
Include	[Master Model]
Lower Bound	
Upper Bound	5.8527
Response	(4) defle
List of Loadsteps	1 Loadsteps
PROB	

5) Create the objective of minimizing the mass.

6) Optimize the structure in OptiStruct window.

2.4 Comment

After the optimization, we can see the values of the responses and the values of the three design variables in the .out file. After the fourth iteration, the objective converges and the optimization process ends.

```
ITERATION   4
 Soft convergence criterion satisfied;
 the design did not change during the last iteration.
```

From the results after the fourth iteration, we get the following design results:

The thickness of the facings is 1.4mm, the thickness of the core is 40mm which attains the maximum value, and the density of the core is 100kg/m^3 which is also the upper bound. The mass of the optimal design is 1.237e-01kg, only 1.6% smaller than the initial mass 1.257e-01 kg.

The contour plot of the displacement is shown with a maximum displacement equals to 5.8mm located at the center. The maximum stress is 4.6MPa, also at the center. Both maximum values of the displacement and the stress are smaller than the required values, so the optimization constraints satisfied.

Then we draw the figure of displacement VS coordinate X for the midline again:

The contour plot of the stress in the core is shown:

And the contour plot of the stress in the skin is shown:

We can clearly see that the stress in the core is almost 0 and the maximum constraint is at the facing level.

Volume = 6.59159E+005 Mass = 1.23749E-004

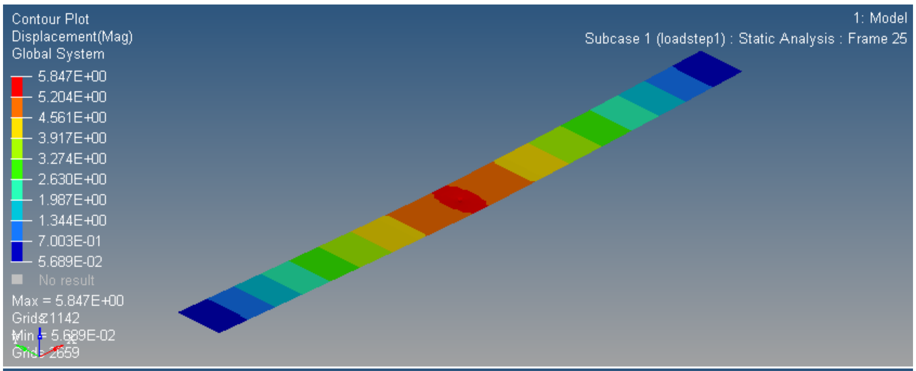
Design Variable ID	Design Variable Label	Lower Bound	Design Variable	Upper Bound
1	dvt	5.000E-01	1.402E+00	2.800E+00
2	dvc	1.000E+01	4.000E+01	4.000E+01
3	rhofr	6.250E-01	1.250E+00	1.250E+00

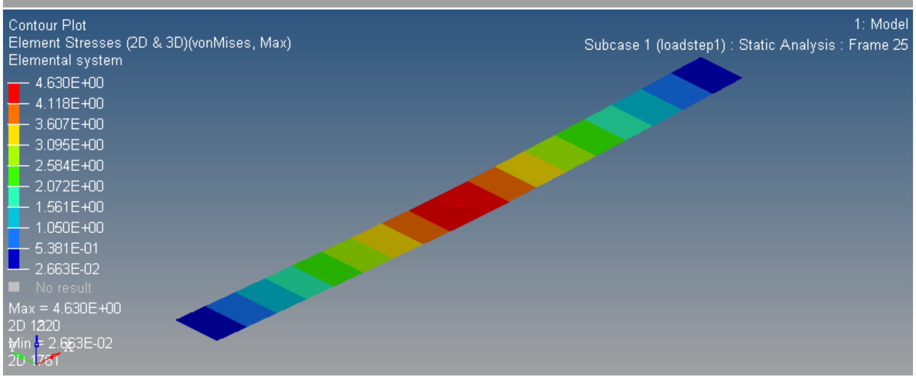
DESIGNED PROPERTY ITEMS TABLE

DVPREL1/2	USER-ID	PROP-TYPE	PROP-ID	ITEM-CODE	PROP-VALUE
DVPREL1	1	PLY	1	T	1.402E+00
DVPREL1	2	PLY	3	T	1.402E+00
DVPREL1	3	PLY	2	T	4.000E+01

DESIGNED MATERIAL ITEMS TABLE

DVMREL1/2	USER-ID	MAT-TYPE	MAT-ID	ITEM-CODE	MAT-VALUE
DVMREL1	4	MAT1	2	RHO	1.000E-10
DVMREL1	5	MAT1	2	E	1.875E+01
DVMREL1	6	MAT1	2	G	1.000E+01





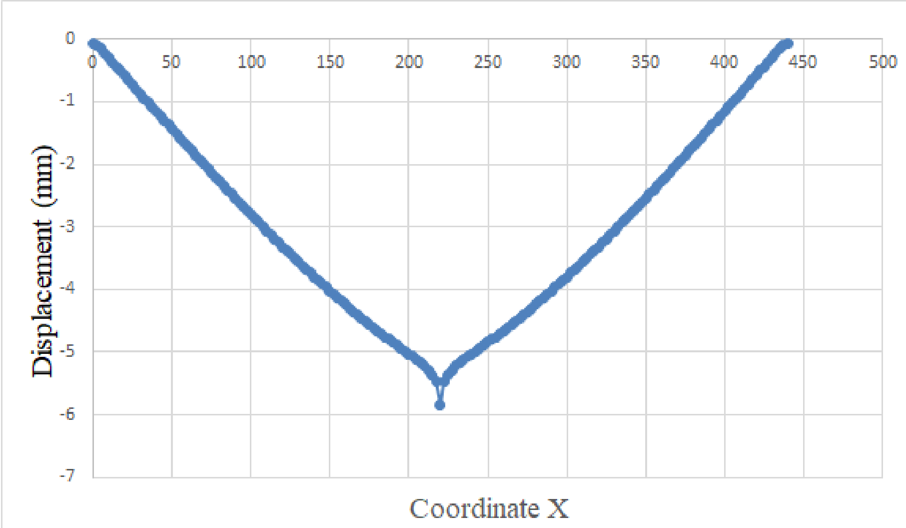
3 Anisotropic skin sandwich beam size optimization

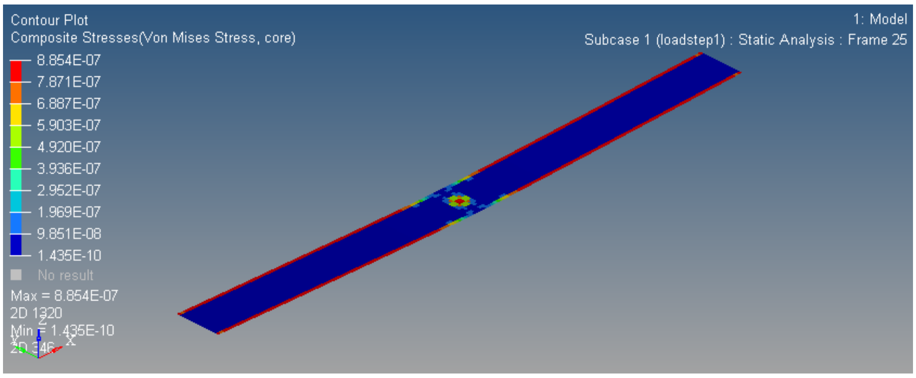
3.1 Finite element analysis

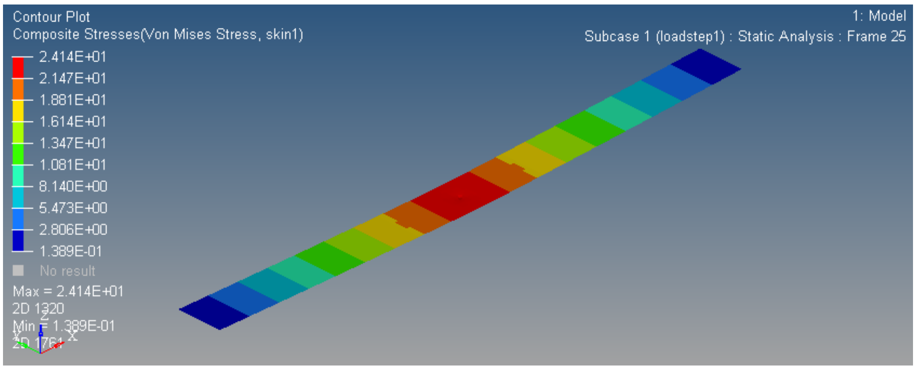
We use the same steps of the finite element analysis in part A.

1) Enter XYZ coordinates to create nodes. Link the nodes and create 4 lines to enclose a 440mm-long, 35mm-wide rectangle.

2) Create a MAT1 type material for the the core layer, the same as before. Then create a MAT8 material for the facing material. The density of this 30% glass fiber reinforced PBT is the same as part A, and for the







Young's Modulus in other directions we use the data collected on the website of DuPont Company, a PBT product called Crastin 6129 NC010 | PBT[3].

3) Create a new PCOMPP property with default parameters and assign it to the component.

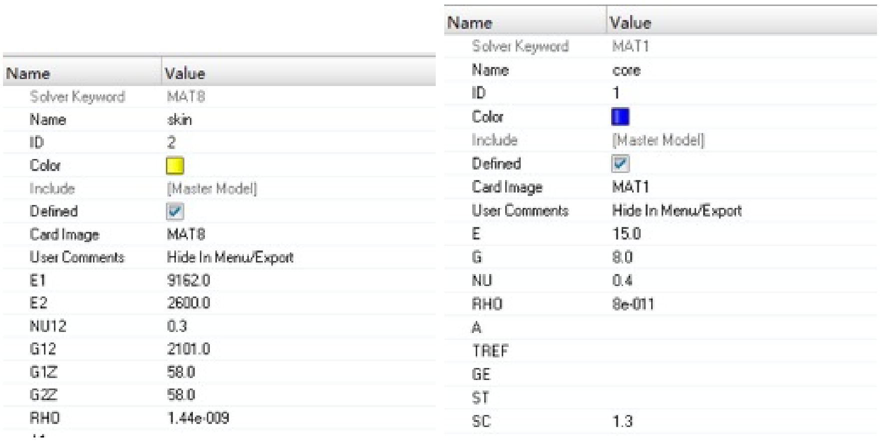
4) Use the geometric lines as boundaries to create separately 5 0.4mm-thick top facing sheet, a 30mm-thick core layer and 5 0.4mm-thick bottom facing sheet. Assign the orientation angle (-45, +45, 0, +45, -45, -45, +45, 0, +45, -45) to the plies 1-10.

5) Create a laminate by selecting the plies.

6) Mesh the geometry. This time we choose 2 as the mesh size, which is also acceptable according to the verification in part A.

7) Realize the plies in order to apply the mesh to the composite geometry.

[3]https://dupont.materialdatacenter.com/profiler/main/standard/main/ds/11731/4097



Name	Value
Solver Keyword	MAT8
Name	skin
ID	2
Color	
Include	[Master Model]
Defined	
Card Image	MAT8
User Comments	Hide In Menu/Export
E1	9162.0
E2	2600.0
NU12	0.3
G12	2101.0
G1Z	58.0
G2Z	58.0
RHO	1.44e-009

Name	Value
Solver Keyword	MAT1
Name	core
ID	1
Color	
Include	[Master Model]
Defined	
Card Image	MAT1
User Comments	Hide In Menu/Export
E	15.0
G	8.0
NU	0.4
RHO	8e-011
A	
TREF	
GE	
ST	
SC	1.3

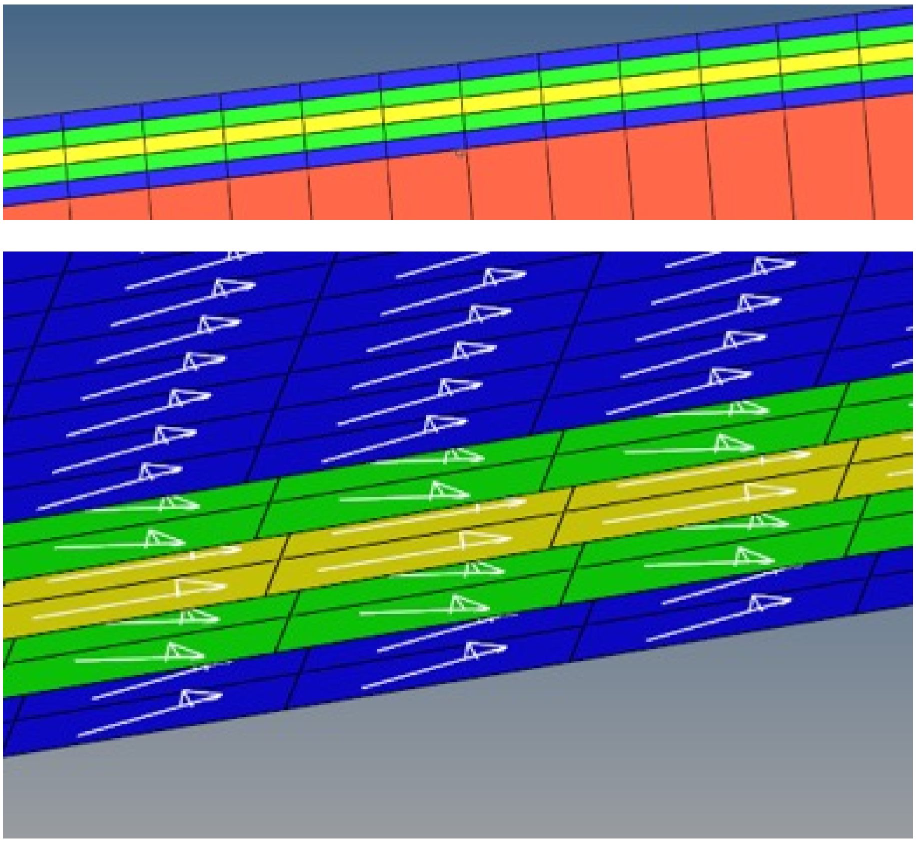


Figure 4: the skin assigned the orientation angle of (-45, +45, 0, +45, -45) for plies 1-5

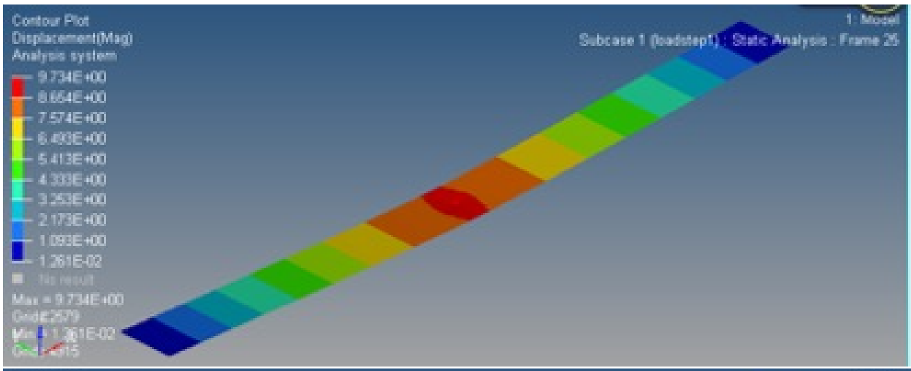
8) Apply a punctual force of 450N at the center of the component and constrain the 2 ends of the component in the Z direction as the sandwich plate is simply supported.

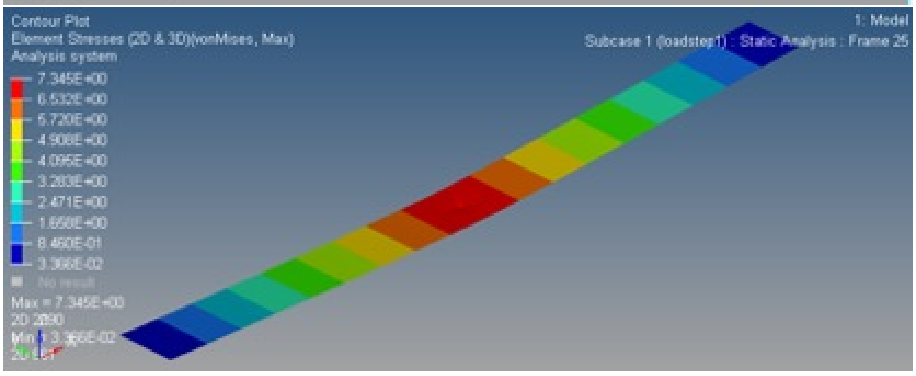
   After analyzed by OptiStruct, we get the contour plot of the displacement and the stress. We can see that the maximum displacement is 9.734mm and the maximum stress is 7.345MPa, and both of them are at the center of the component.

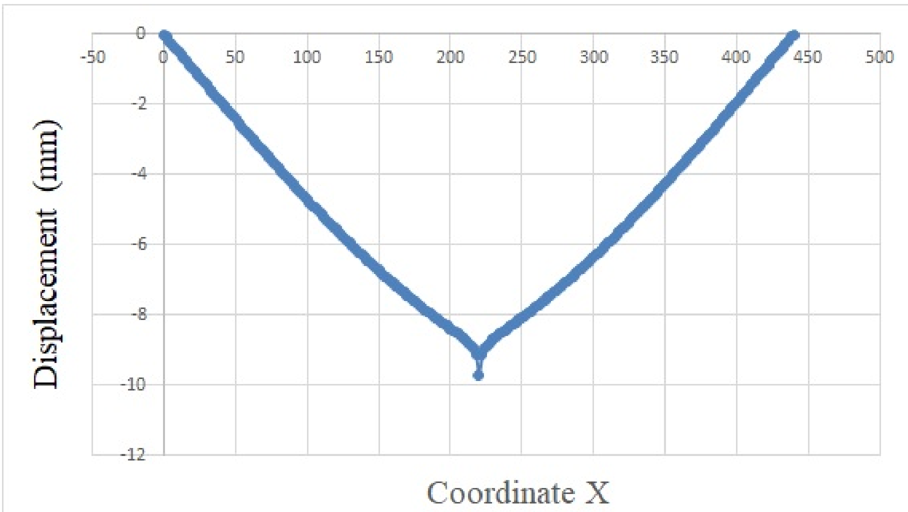
Using the same method as part A, we draw the graph of displacement magnitude VS coordinate X for the midline.

3.2 Optimization

For the optimization stage, we follow the same steps mentioned above in the part A with several differences:
- 8 design variables in total including thickness and orientation angle for each layer of the facing, thickness of the core, density fraction of the core.







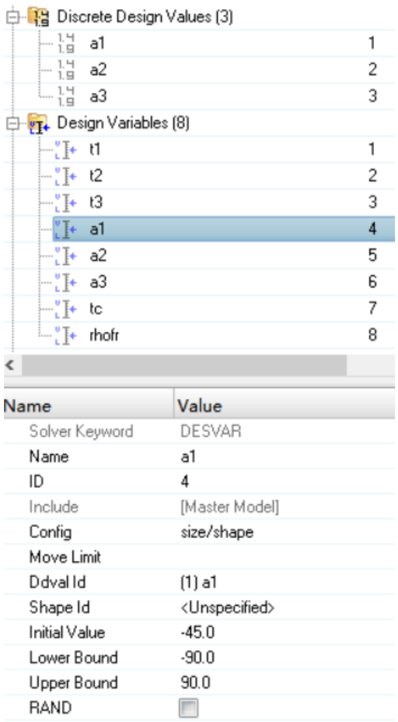
- Orientation angle of the layer can only vary in a discrete set (-90, -75, -60, -45, -30, -15, 0, 15, 30, 40, 45, 60, 75, 90). Discrete design values are created and assigned to the orientation angle design variables as below.

Optimization process is completed after 10 iterations. The optimal design given in the 10th iteration is listed below:

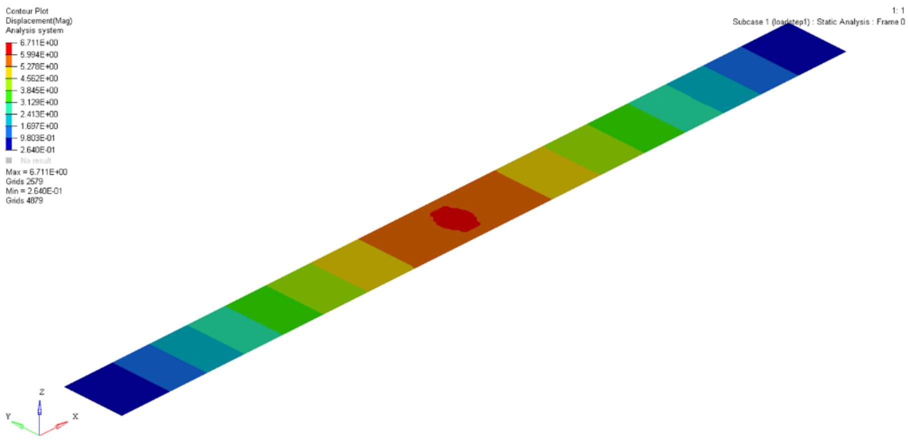
- Volume = 6.54496E-001 m^3
- Mass = 1.09784E-001 kg
- The optimal value for all orientation angles is 0 degree, which means each face sheet has only 1 layer with an orientation angle of 0 degree.
- The sum of thickness of all layers in each face sheet equals to 1.25mm.
- Thickness of the core = 40 mm
- Density of the core = 88.23 kg/m^3

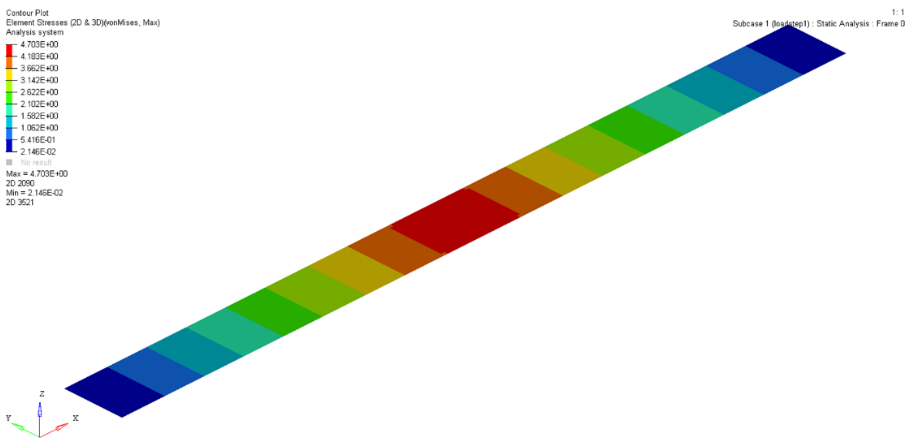
In conclusion, in the optimal design, there is only one 1.25mm thick-layer with an orientation angle of 0 degree in each face sheet. The optimal thickness of the core is 40 mm and the optimal density of the core is 88.23 kg/m^3. The mass of the optimal design is 1.1E-001 kg and is 15.4% smaller than the initial mass which is 1.3E-001 kg (Initial Mass = 1.25664E-004).

The maximum displacement in the component is 6.7 mm at the center, where the punctual force is applied.



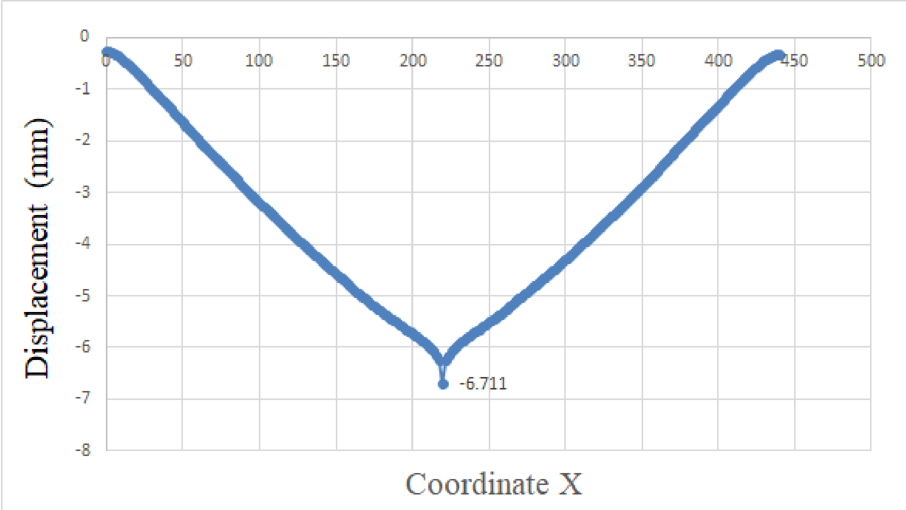
The maximum stress in the component is 4.7 MPa at the center.



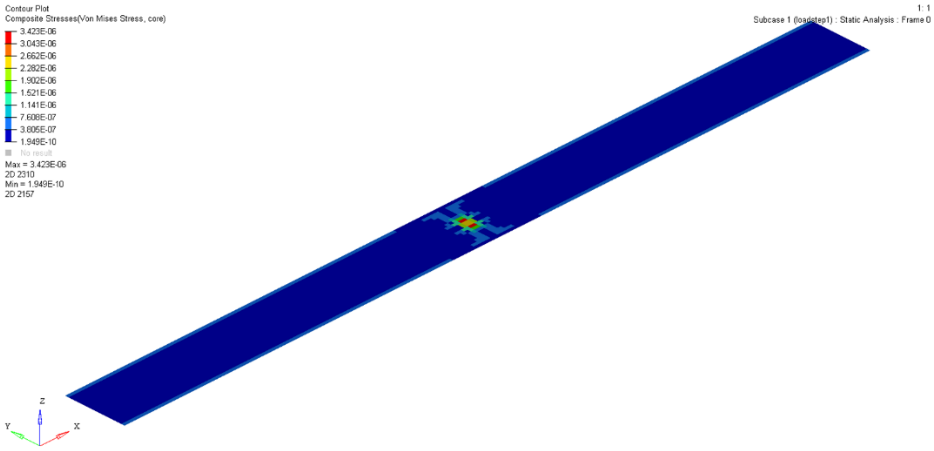


The maximum displacement and the maximum stress are smaller than 70% of the maximum calculated in the initial design. The criteria are satisfied.

The graph of displacement VS coordinate X for the midline of the component is shown below:



The contour plot of the stress in the core is shown below. The stress is almost 0 in the core.



The contour plot of the stress in one layer is shown below. The stress in the layers is much larger than in the core. We can thus conclude the stress is mainly taken by the skin and there is almost no stress taken by the core.