

See discussions, stats, and author profiles for this publication at: <https://www.researchgate.net/publication/332383984>

Parametric Study of Honeycomb Composite Structure Using Open Source Finite Element Software

Conference Paper · April 2019

CITATIONS

0

READS

90

5 authors, including:



Ahmed H. Abdulaziz

Ain Shams University

5 PUBLICATIONS 1 CITATION

SEE PROFILE



Mohammed Hedaya

Ain Shams University

1 PUBLICATION 0 CITATIONS

SEE PROFILE



John Mccrory

Cardiff University

16 PUBLICATIONS 122 CITATIONS

SEE PROFILE



Karen M Holford

Cardiff University

135 PUBLICATIONS 1,372 CITATIONS

SEE PROFILE

Some of the authors of this publication are also working on these related projects:



Damage in composite structures [View project](#)



aeroelastic analysis of plate wings with piezoelectric sheets [View project](#)

PARAMETRIC STUDY OF HONEYCOMB COMPOSITE STRUCTURE USING OPEN SOURCE FINITE ELEMENT SOFTWARE

*Ahmed H. Abdulaziz^{1,2}, Mohammed Hedaya¹, John P. McCrory², Karen M. Holford², Adel Elsabbagh¹

¹Design and Production Engineering Department, Faculty of Engineering, Ain Shams University, Abbaseya, Cairo, Egypt, 11517

²School of Engineering, Cardiff University, Wales, United Kingdom, CF24 3AA

* ahesham@eng.asu.edu.eg

Summary

This paper deals with the parametric optimisation of a simply supported sandwich panel made of honeycomb composite structure using sequential quadratic programming SQP. The panel consists of aluminum honeycomb sandwiched between two orthotropic fiberglass faces. The parameters studied are fibreglass thickness, t_f , honeycomb height, h , and honeycomb wall thickness, t_c . The objective was to minimise weight to bending stiffness ratio by using the nonlinear MATLAB function *fmincon*, considering the maximum central displacement and intercellular buckling as the constraints. Following this, a static structural analysis was conducted on the optimised structure using the open source finite element solver CalculiX and Salome Platform software for preprocessing. The utmost displacement of the honeycomb panel was desirably less than the serviceability limit. Preliminary results show that composite honeycomb structures can be optimised yielding low weight to bending stiffness ratio using SQP method and CalculiX for design evaluation.

Keywords: *Parametric optimisation; Sequential quadratic programming; CalculiX; Salome Platform; Honeycomb composites*

Introduction

In the wind turbines industry, blade materials must have a low weight to bending stiffness ratio for optimal aerodynamics. Aluminum honeycomb can play a vital role in manufacturing longer wind turbine blades with fibre glass as outer skin due to the cost benefit when compared with using carbon fibre composites alone. This study proposes a parametric optimisation using Sequential Quadratic Programming algorithm SQP, which can be used in MATLAB's *fmincon* function to minimise weight to bending stiffness. This function deals with nonlinear constrained convex objective functions with linear/nonlinear equalities and inequalities. To evaluate the optimum values, structural analysis using finite element method was carried out using the open source finite element solver CalculiX. CalculiX has many interesting features such as wide compatibility with open source CAD modelling and mesh generating softwares such as Salome platform, FreeCad and GMSH. Further, it is extremely versatile as it can be modified using Python programming on Linux platform. For instance, recently Genao et al [1] have proposed a framework to merge Calculix FE solver with NASA's Micromechanics Analysis Code MAC to promote multiscale analysis of the composite materials effectively. Galehdari et al [2] have optimised honeycomb structural parameters using SQP and genetic algorithm for minimising the weight to absorbed energy ratio to increase the crashworthiness. Park et al [3], have conducted structural linear analysis on a cantilever model and sliding contact analysis using CalculiX and Code_Aster comparing the results favourably with ANSYS commercial FE software. Therefore, in this paper, Salome platform was used for meshing the honeycomb composite structure and CalculiX FE solver was used for conducting structural analysis.

Methodology

The common failure modes of sandwich structures may happen due to severe shear force, intercellular buckling, core crushing, delamination in case of orthotropic composite facets,

shear crimping and punching shear [4]. The sandwich panel dimensions width, b , and length, l , are $0.2\text{m} \times 0.2\text{m}$ respectively, and the honeycomb top/bottom faces are fibre glass with thickness, t_f , honeycomb height, h , and with wall thickness, t_c . The sandwich panel is treated as a shell structure considering the length/width are significantly larger than the height [5]. The top/bottom faces consist of three laminates with a combined thickness of t_f mm, and a $[0^\circ/90^\circ/0^\circ]$ layup meaning that the in-plane/bending deformations are decoupled. The fibre glass composite micromechanics properties are computed using the Halpin-Tsai empirical approach. Moreover, the research methodology workflow is presented in Figure 1.

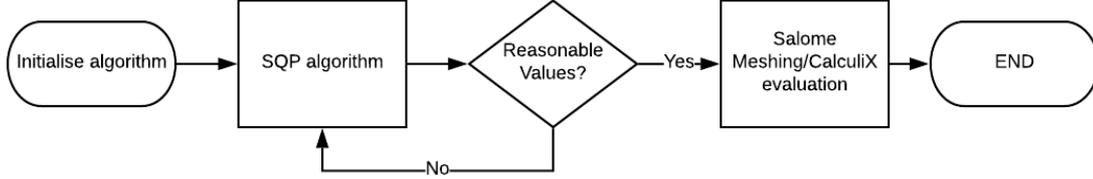


Figure 1: Block diagram of the research methodology

According to Bitzer [6] the equivalent bending stiffness of honeycomb sandwich panel D_{eq} can be computed using Eq. (1).

$$D_{eq} = \frac{E_1 t h^2}{2\lambda} + \frac{E_1 t^3}{6\lambda} + \frac{E_c t_c^3}{12\lambda_c} \quad (1)$$

where; E_1 is longitudinal Young's modulus of faces and E_c is the honeycomb Young's modulus, λ is equal to $(1 - \nu_x \nu_y)$ and ν_x, ν_y are Poisson's ratio in longitudinal and transverse directions of composite layer, λ_c is $(1 - \nu_s)$, ν_s is Poisson's ratio of aluminum. The weight is $(g * b * l * (2 * \rho_f * t_f + \rho_{core} * h))$ where, g is gravity acceleration, b is breadth, l is length and ρ_{core} is honeycomb density and ρ_f is top/bottom face material density. *Fmincon* function in MATLAB utilises sequential quadratic programming SQP algorithm to obtain the optimum minimum value. Therefore, the objective function is to minimise weight to bending stiffness which is formulated as in Eq. (2)

$$\text{Minimise } f(X) = \frac{\text{Weight}}{\text{bending stiffness, } D_{eq}} \quad (2)$$

The nonlinear constraints of the design are displacement δ and intercellular buckling P . The displacement at the panel centre should not exceed $span/100$. The displacement of the panel at the centre can be computed by Lèvy's single series as shown in Eq. (3)

$$\delta = \sum_{m=1}^{\infty} Y_m \sin \frac{m\pi x}{a} \quad (3)$$

Since the panel is square of side a , Eq. (3) can be re-written as given in Eq. (4) [7], and the intercellular buckling load must be less than critical value as computed in Eq. (5).

$$\delta = 0.00406 \frac{p a^4}{D_{eq}} \leq (\text{span}/100 = 2\text{mm}) \quad (4)$$

$$P < (P_c = K E_s (t c)^3 / ((1 - \nu_s^2) * s)) \quad (5)$$

where k is 5.75, E_s is the honeycomb Young's modulus, The upper/lower limits of the design variables are tabulated in Table 1. SQP function tolerance is 10^{-6} .

Table 1: Optimisation design variables with upper/lower bounds

Design Variable	$x(1)$	$x(2)$	$x(3)$
Geometric Parameter	t_f	h	t_c
Upper bound	0.002	0.010	0.001
Lower bound	0.0015	0.001	0.0001

As a side note, the positive definite Hessian matrix is a measure of function convexity over the domain [8]. Therefore, the eigenvalues of the Hessian have been computed and they are positive. After computations, a local minimum that satisfies the nonlinear constraints has been detected. Further, the variations of weight to bending stiffness ratio according to different honeycomb height and faces thickness are plotted in Figure 2. The eigen values of the Hessian matrix are computed. The iterations stopped as the objective function is non-decreasing in the feasible region. The buckling load factor (BLF) has been computed (i.e. $P_{critical}/P_{acting}$) and it is larger than 1 which indicates safety of the honeycomb cell wall under buckling.

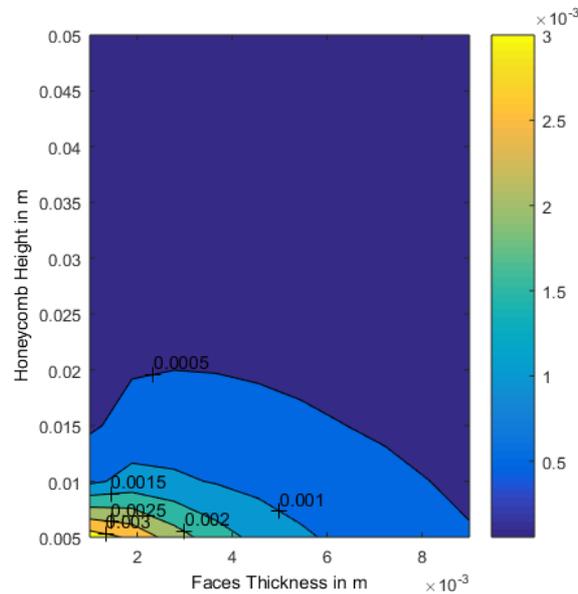


Figure 2: Weight to bending stiffness ratio for different height and face thickness

The optimum values are tabulated in Table 2. Furthermore, the MATLAB script used to obtain this result is provided at the end of this paper.

Table 2: Honeycomb optimum values

t_f (m)	H (m)	t_c (m)	W (N)	D_{eq} (N.m)	W/D_{eq} (N/N.m)	Intercellular Critical Buckling (N)	Buckling Load Factor
0.0020	0.0100	0.0003	3.31564	5.8898e+03	5.6295e-04	1599.4	1.6

Finite Element Model

To evaluate the optimisation results, a honeycomb composite panel of zero thickness is processed in Salome Platform and meshed with “S6” and “S8R” shell elements [9] using Netgen 1D-2D option with maximum length 3 mm and minimum length 1.5 mm. Further, for better accuracy in solution, second order approximation for the meshing process is followed. However, care must be taken in meshing process as unlike commercial softwares, node-to-node connectivity is not assured for multiple surfaces automatically. Therefore, the sandwich panel must be partitioned into multiple shells and edges to assure the nodal connectivity. Yet, after partitioning it, the honeycomb core and top/bottom faces must be grouped as well the two edges at the bottom face to form the elemental and nodal groups which will be used later for

materials definitions and boundary conditions in CalculiX. The mesh is saved as “.unv” file to obtain a Python code of elements data. Afterwards, *unical* mesh converter in CalculiX is used to convert the (.unv) mesh file into input deck for further finite element analysis. Figure 3a) presents a block diagram of the FE process and 3b) shows the meshed honeycomb panel. It consists of 18,309 quadrangle elements “S8R” and 2,863 triangular elements “S6”. These elements expand to 3D quadratic brick elements and 3D wedge elements in modelling the top/bottom composite faces .

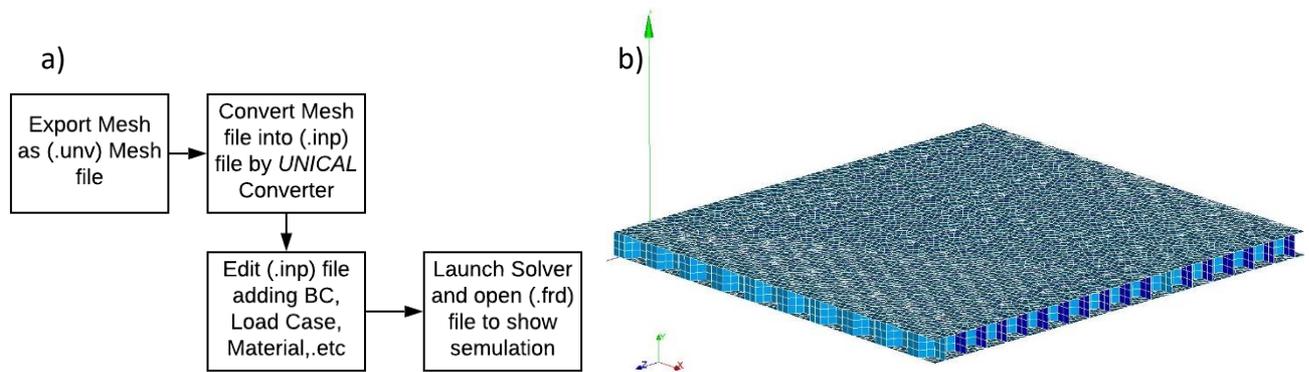


Figure 3: a) Finite element steps, b) Honeycomb Meshing in Salome Platform

The lateral concentrated force 5,000 N is positioned at the centre of the panel and structural static analysis is conducted. The maximum central deflection is 1.31 mm as shown in Figure 4a. Compared to serviceability limit (i.e. span/100) which has been utilised within the optimisation, the utmost deflection obtained by CalculiX for the panel is desirably less. It is noteworthy to mention that in CalculiX section definition, the shell elements after expanding to build the required thickness may intersect at the corners as shown in Figure 4b. This intersection is dependent on the shell offset value and its normal direction whether negative or positive.

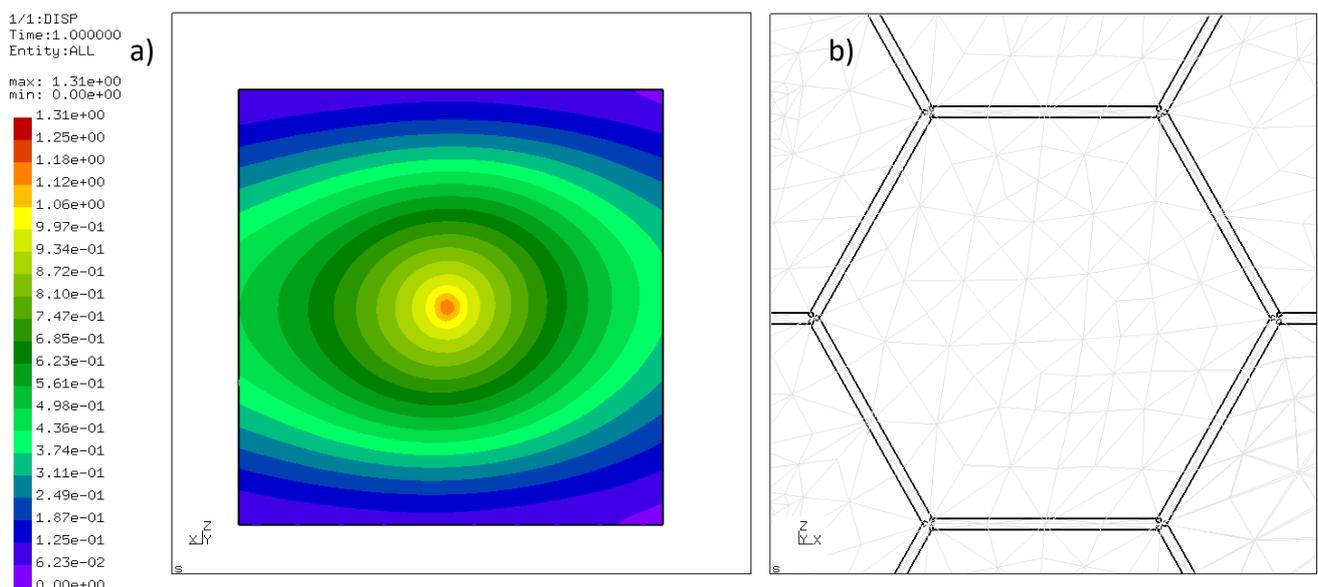


Figure 4: a) Maximum central displacement of the simply supported panel, b) the shell elements corner intersection after expanding

Conclusions

In conclusion, the honeycomb composite panel has been optimised using sequential quadratic programming. The ratio of weight to bending stiffness is minimised considering the intercellular buckling and lateral deflection as the main constraints functions. The optimum geometric parameters are the faces thickness, core height and core thickness. After the optimisation, a

numerical model is processed and meshed with S8R/S6 shell elements in Salome platform then a structural static analysis has been carried out in the open source finite element solver CalculiX. Overall, it is demonstrated that coding with CalculiX is flexible nevertheless care must be taken in the section definition. Mainly, the shell element offset and normal direction because the results are dependent on them. The maximum displacement retrieved from CalculiX was 1.31 mm which is less than the limiting value specified in SQP optimisation. Future research should be devoted to couple the SQP optimisation code within Salome Python code of the geometry/mesh to be processed after that in Calculix input deck. In addition, optimisation of honeycomb composite structure might prove an important area for future optimisation research so it is recommended that another optimisation technique such as Method of Moving Asymptotes MMA or Genetic algorithms is used and all optimisation results are examined.

Acknowledgements

This research is carried out within the PhD project entitled “*Investigation of Honeycomb Composite Structure for Wind Turbine Blades with Acoustics Emissions Damage Assessment*” funded by Newton-Mosharafa Fund in Egypt, *I.D: (NMJ 3/18)*. Many thanks to Prof Otto Ernst Bernhardt in Karlsruhe University of Applied Sciences, Germany, Hossem Elnachmie and Haoluan Li in Cardiff University for recommendations and discussions about some technical issues encountered in finite element modelling and optimisation.

References

- 1 F. A. Yapor Genao, E. J. Pineda, B. A. Bednarczyk and P. A. Gustafson, "Integration of MAC/GMC into CalculiX, an open source finite element code," in *AIAA SciTech Forum*, San Diego, California, 7-11 January 2019.
- 2 S. Galehdari, M. Kadkhodayan and S. Hadidi-Moud, "Analytical, experimental and numerical study of a graded honeycomb structure under in-plane impact load with low velocity," *International Journal of Crashworthiness*, vol. 20, no. 4, pp. 1754-2111, 2015.
- 3 S. K. Park, D.W. Seo, H. Jeong and M. Kim, "Performance evaluation of open-source structural analysis solver, CalculiX and Code_Aster, for linear static and contact problems," *ICIC Express Letters* , vol. 12 , no. 7, pp. 655-662, 2018.
- 4 G. Lubin, *Handbook of Composites*, Springer Science & Business Media , 2013.
- 5 T. Kubiak, *Static and Dynamic Buckling of Thin-Walled Plate Structures*, Lodz, Poland: Springer, 2013.
- 6 T. N. Bitzer, *Honeycomb Technology: Materials, Design, Manufacturing, Applications and Testing*, Springer Science & Business Media, 2012.
- 7 S. P. Timoshenko and S. Woinowsky-Krieger, *Theory of Plates and Shells*, Singapore: Mcgraw-Hill Inc., 1959.
- 8 A. Messac, *Optimization in Practice with MATLAB® for Engineering Students and Professionals*, Cambridge: Cambridge University Press, 2015.
- 9 E. J. Barbero, *Finite Element Analysis of Composite Materials using Abaqus*, Taylor and Franciscn Group, 2013.

MATLAB M-script

```
%This code is written to perform
parametric optimisation using SQP
algorithm-All dimensions are in SI-
units~~After reading left hand
column to its end, continue reading
ub= [0.002,0.010,0.001];
nonlcon = @Constrains;
x0 = [0.0018,0.002,0.0002] ;
%Initialisation point
```

```

from top of right column to its
end. ~~File 1 consists of 3
sections. File 2 presents
constraints functions
% File 1-Section 1: Halpin-Tsai
empirical approach for
micromechanics computations &
parameters definition
clc
rhof = 1800 ; % Faces density
rhos = 2700; % Aluminum density
% Material is Al honeycomb
L=0.2; % length
w=0.2; % breadth
g=9.81; % Gravity acceleration
E=60e9;
Ef = 73.1e9; %Fibre glass Young's
modulus
Em = 3.45e9;
vf=0.55; %volume fibre fraction
vm=0.45 ; % volume matrix fraction
uf = 0.22; % poisson ratio
fiberglass
um = 0.33; % poisson ratio Epoxy
Gf = 30e9; %Shear rigidity of fibre
Gm = 1.25e9 ; %Epoxy shear rigidity
v12 = uf*vf + um*vm ;
v21 = v12;
k = 0.9;%k:fibre misalignment
factor
E1 =k*(Ef*vf+Em*vm);%Longitudinal
Young's modulus
zeta = 2;
etae = ((Ef/Em)-1)/((Ef/Em)+zeta);
E2 = Em*(1+zeta*etae*vf)/(1-
etae*vf);%Longitudinal Young's
modulus
etaG= ((Gf/Gm)-1)/((Gf/Gm)+zeta);
G12 = Gm*(1+zeta*etaG*vf)/(1-
etaG*vf);
lambda = 1-v12*v21 ;
v_Al = 0.3 ; %Poisson ratio Al
lambdac = 1-v_Al;
s=0.0064;%Side length of cell
%File 1-Section 2:Optimisation
Formulation
f=@(x) 2*g*w*L*(rhof*x(1)+(x(2)*x(3)
*rhos/(s*(3^0.5))))/(((E1*x(1)*(x(2)
)+x(1))^2/(2*lambda))
+(E1*x(1)^3/(6*lambda))+ (E*x(3)^3/(
12*lambdac))));
A = [];
b = [];
Aeq = [];
beq = [];
lb =[0.0015,0.001,0.0001];

options =
optimoptions('fmincon','Algorithm','
sqp','Display','iter') ;
[x,fval,exitflag,output,lambda,grad,
hessian]=fmincon(f,x0,A,b,Aeq,beq,lb
,ub,nonlcon,options);
disp (hessian);
e = eig(hessian);
[~,r] = chol(hessian);
disp (e)
disp (x)
M=2*g*w*L*(rhof*x(1)+(x(2)*x(3)*rhos
/(s*(3^0.5))));
K = M/fval ;% stiffness at optimum
point
% File 1-Section 3: 3D plot of the
variables and corresponding
objective function
tf1 = linspace(0.001,0.009,10) ;
h1 = linspace(0.005,0.05,10) ;
tc1 = linspace(0.0001,0.009,10);
[XX,YY] = meshgrid(tf1,h1);
[VV] = meshgrid(tc1);
WW =
2*g*w*L.*(rhof*XX+(YY.*VV*rhos./(s*(
3^0.5)))); %Weight at optimum point
; %Weight
DD = (E1*XX.*(YY+VV).^2/ 2*0.9274)
+(E1*XX.^3/(6*0.9274))+(E*VV.^3/(12*
0.7));
Func = WW./DD ;
[FF] = meshgrid(Func);
%plot objective function vs design
variables core height&face thickness
figure
set(gcf, 'PaperPosition', [0 0 4
4]);
C = contourf(XX,YY,Func);
clabel(C,'FontSize',12)
xlabel('Faces Thickness in
m','FontSize',12,'Color','k');
ylabel('Honeycomb Height in
m','FontSize',12,'Color','k');

%File-2:Constraints.m File
function [c,ceq] = Constrains(x)
%Displacement at centre due to 5000
N concentrated force
c(1)=0.00406*5000*0.2^4/((3.758175e1
0*x(1)*(x(2)+x(1))^2/(2*0.9274))
+(3.758175e10*x(1)^3/(6*0.9274))+(6e
10*x(3)^3/(12*0.7)))-
0.002;%deflection
c(2) = 1000 -(5.75*60e9*x(3)^3/((1-
0.3^2)*0.0064)); %intercellular
buckling acting force on the side
ceq = [];
end

```