

COMPUTER AIDED DESIGN OF COMPOSITE STIFFENED PANELS

Kim Nielsen Martinez and Ever J. Barbero
West Virginia University
Morgantown WV 26506-6106

ABSTRACT

New software for the design of composite stiffened panels (CSP) is presented and integrated within the web application CADEC, mimicking the design process of CSP, including panel dimensions, skin and stiffener dimensions, flange and web laminates, all the way down to details such as lamina properties, which can be experimental values or calculated from fiber/matrix properties stored in the user's database. This approach ensures consistency of data and results calculated at different design stages by enforcing known relations between the different objects in the web application, which provides online access, cloud storage and computation, as well as real time deployment of software updates. The CSP module allows the analyst to obtain the buckling load and visualize the buckling mode interactively. This is facilitated by a fast solver specifically developed for this application. Comparison with experimental data from the literature is presented.

1. INTRODUCTION

Composite stiffened panels (CSP) design requires the optimization of several parameters such as panel dimensions, stiffener dimensions, flange and web laminates, etc. When one of these parameters is changed, all the calculations, finite element mesh, and boundary conditions must be redone, which is time consuming and error prone.

A new module called CSP is presented and integrated within the web application CADEC [1] in order to render an accessible and portable tool, using advanced features such as object-oriented programming (OOP), user authentication, database (DB) storage, and web visualization (WebGL). The tool generates the mesh and geometry from a set of predefined parameters that defines a given panel. Then, it computes the critical buckling load and provides instant visualization of the buckling mode. The application is validated by comparison with experimental results from the literature.

This paper is aimed to non computer science engineers with some background in programming. The value of the proposed software resides in the simplicity of it, which allows the design engineer to interactively define a stiffened panel in term of high level parameters and analyze it on the fly. A new integration scheme using full integration and relaxed intralaminar constraint is used to make the solution more robust; that is, eliminate zero-energy modes while avoiding locking. This allows the analysis of panels with arbitrary aspect ratio including very thin and very narrow ones. Furthermore, a novel eigensolver algorithm is proposed to allow computation of buckling loads/modes for shear loading and well as inplane loading.

Copyright 2014 by West Virginia University. Published by CAMX – The Composites and Advanced Materials Expo with permission.

CAMX Conference Proceedings. Orlando, FL, October 13-16, 2014. CAMX – The Composites and Advanced Materials Expo

2. USER INTERFACE

The CSP module is organized similarly to CADEC [1]. The user interacts with the application in two main areas: My Documents and Chapters. *My Documents* contains the pages that allow the user to define objects that will be later subject to analysis, or used by other objects. *Chapters* comprises the pages in charge of performing the analysis.

2.1 Problem Definition

First, the user accesses *My Documents* -> *My Stiffened Panels*. A list of the stiffened panels that are stored in the database is loaded. It is possible to click on one of those objects to edit its properties, or click on  to define a new panel. In both cases the user is redirected to the *Edit* page (Fig. 1) where several properties must be specified in order to define the problem.

CADEC : My Documents : My Stiffened Panels : New Stiffened Panel

General Properties

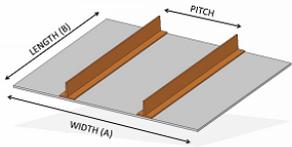
Name :

Geometric Properties

Width (A): [mm]

Length (B): [mm]

Add curvature to the panel



Stringer Properties

Stringer Profile:

Laminates

Skin:

Web:

Top Flange:

Bottom Flange:

Loads

BC and Load Type:

Side 3

N_r : [$\frac{N}{mm}$]

N_s : [$\frac{N}{mm}$]

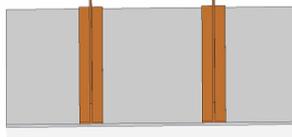
BC:

Side 4

N_r : [$\frac{N}{mm}$]

N_s : [$\frac{N}{mm}$]

BC:



Side 2

N_r : [$\frac{N}{mm}$]

N_s : [$\frac{N}{mm}$]

BC:

Side 1

N_r : [$\frac{N}{mm}$]

N_s : [$\frac{N}{mm}$]

BC:

[Terms and Conditions](#) | [Privacy Policy](#) | Hosted by [WVU Energy Materials Program](#)

Figure 1. Screen shot of the *Edit* page of the CSP tool.

First, a *name* must be given to the panel, so it can be stored in the database. Next, the geometry of the panel is introduced: width (A), length (B), and curvature if needed. Below that, the stiffener profile should be chosen, along with its characteristic dimensions, number of stiffeners, pitch, and offset (Fig. 2). Similar drawings can be seen in the application itself, online.

Stringer Properties	
Stringer Profile:	Omega ▾
Stringer Pitch:	100 [mm]
Stringers Offset:	0 [mm]
Number Of Stringers:	1
Stringer Height:	25 [mm]
Stringer Width:	35 [mm]
Top Flange Width:	10 [mm]
Bottom Flange Width:	10 [mm]

The diagram shows a cross-section of an Omega stiffener profile. It consists of a central web and two flanges. The top flange has a width labeled 'Top Flange Width'. The bottom flange has a width labeled 'Bottom Flange Width'. The height of the web is labeled 'Stringer Height'. The total width of the profile, including both flanges, is labeled 'Stringer Total Width'.

Figure 2. Parameters defining stiffener profiles in *Edit* page.

Seven profiles are included: T, J, Z, C, I, Hat (also called Omega), and JC. The latter is a combination of J and C profiles.

Next, laminates must be assigned to each part of the panel: skin, web, and flanges. These laminates should have been created previously in the corresponding module in *My Documents*. Laminates are defined by the laminate stacking sequence (LSS), which is comprised of laminas, thicknesses, and orientations. Any combination of laminas can be chosen, including diverse material systems as well as types of laminas, such as unidirectional, random, textile-reinforced, and experimental. Experimental laminas allow the introduction of as-produced lamina properties. The remaining lamina types have their properties automatically calculated by CADEC's micromechanics module in terms of user specified fiber and matrix properties, as well as fiber volume fraction [2].

Finally, boundary conditions and loads must be declared, which can be applied to either the skin only or to both skin and stiffeners. Four different types of boundary conditions are available: Simply Supported (SS), Clamped (CC), Symmetric (SYM) and Free. Either in-plane normal or shear loads can be applied to the panel.

At this point, the user should click on the  button. The *Visualize* button displays the geometry and mesh of the panel online, using WebGL technology [3].

2.2 Problem Solution

Once the problem has been defined in My Documents, it can be solved. To solve, the user goes to *Chapters -> Stiffened Panels*.

The layout of the page is very similar of the *Edit* page in *My Documents*. First the panel subject to analysis should be selected from the list, so that all properties are retrieved from the DB. It is possible to modify these parameters and re-run the analysis as many times as required. Before running the analysis, the user must specify either to apply normal or shear loading.

Then, the user should click on Calculate button, and a loading bar appears informing the user that the computations are being made in the server. After the analysis finishes, the user is redirected to Visualization page, where the lowest eigenvalues and buckling mode plot are shown (Fig. 3).

3. IMPLEMENTATION

The CSP module consists of five web pages. Each of these pages is created using ASP.NET and C# programming languages. ASP.NET allows the addition of controls to the page and defines its layout. These controls include table rows, buttons, textboxes, and dropdown lists. C# is used to define the code behind the ASP pages, i.e., implementing the actions to be performed on the controls.

In order to illustrate this, let us consider a page with two controls: a textbox where the user will introduce the length of the panel, and a button. In this case, the code-behind would include an event that will react to the action of clicking on the button, so that the value introduced by the user in the textbox is stored in the database. Additionally, a validator checks whether the value introduced by the user is a valid number.

Next, different concepts and approaches applied during the programming of the CSP module are explained.

3.1 Integration

An advantage of embedding CSP inside CADEC web application is the possibility of reusing previous extant modules to achieve a more powerful application. This approach allows the user to define the laminates stiffened panels using the corresponding modules in CADEC. This complete set of tools allows the definition of laminas either introducing directly experimental data or by defining fiber and matrix characteristics and computing the resultant properties of the lamina. After a lamina is defined, the laminate module is used to stack different plies, specifying their orientations and thicknesses to determine the final laminate properties.

Thanks to OOP, the communication between the stiffened panel class and previous classes (lamina, laminate, etc...) is easy, and the integration of the new module within the application is robust.

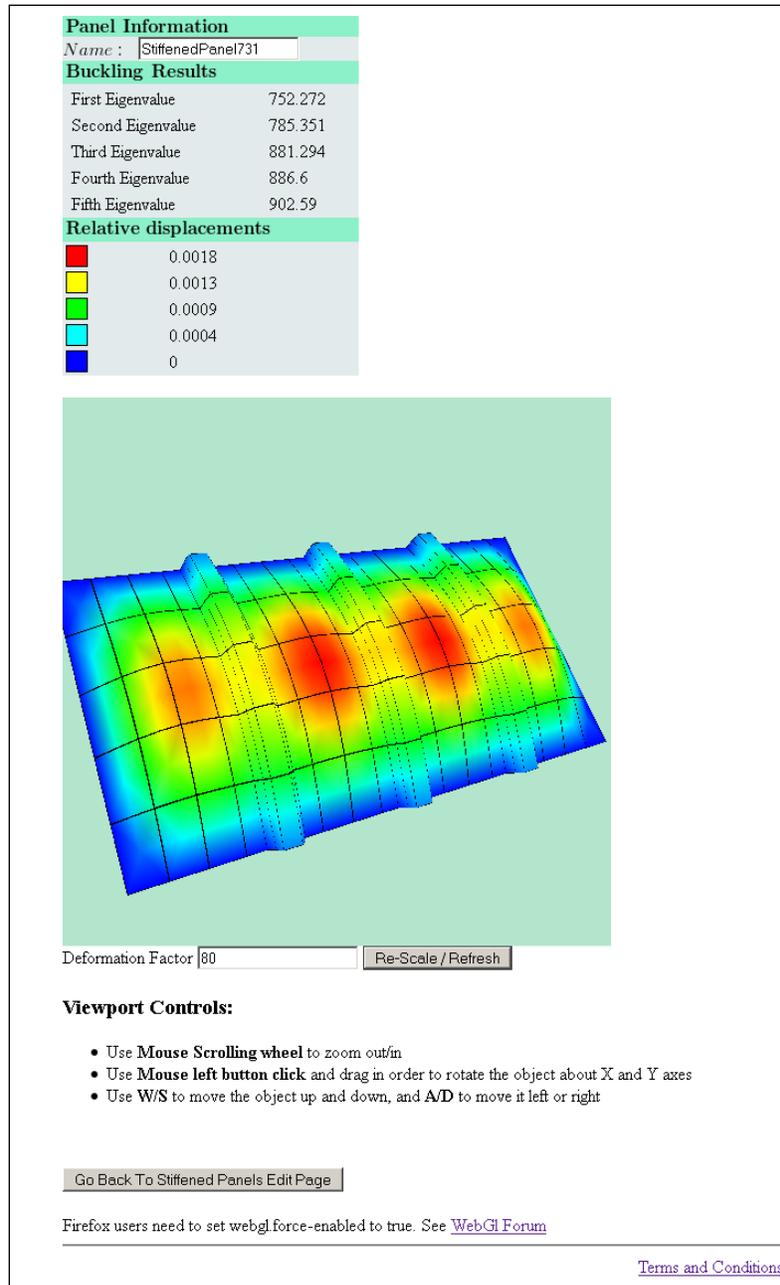


Figure 3. Visualization page of the tool CSP, in which buckling results are presented.

3.2 Code Reuse

One of the many advantages of OOP is code reusability. For instance, a “button” is a class that defines the properties and methods of that control in such a way that all buttons are similarly coded.

OOP classes were defined so that they can be reused in multiple pages. In order to illustrate this, let us consider a class to define a row inside a table. Notice that the *Edit* page in *My Documents* where stiffened panels are created consists of a large table with multiple rows. Each row

represents a property of an object. It is composed by different controls: one label (which includes the name of the property), a textbox where the user introduces the value, and a third column that contains the units of that property. The class *CADECproperty.cs* contains all these elements and specifies the layout, so that all the rows have a similar appearance. Two additional classes derived from this one to add code to specify whether the property is going to be a float, or an integer. Then, the general row classes contain the following properties:

- Property name: text that will appear in the label.
- Property units: text that will appear in the units column
- Error message: text shown if the user input is not valid.
- Tip message: text shown when the mouse hovers on the property label.
- Range error: text explaining a value-out-of-range error.
- Maximum value: maximum allowable value for the property.
- Minimum value: minimum allowable value for the property.

After defining these general-purpose classes, it is easier to create new property classes. For example, the class that defines the *panel width (A)* can be created by inheritance from the *DoubleProperty.cs* class, because the dimensions can be represented by a float number. Therefore, it is just necessary to assign the proper values to the previous properties, which for this example could be:

- Property name: “Panel Width (A)”.
- Property units: “mm”.
- Error message: “Panel width value is not a valid number”.
- Tip message: “Panel width dimension”.
- Range error: “Panel width should be a positive value”.
- Maximum value: 10.000.
- Minimum value: 0.

3.3 Mesh Generation and 3D Visualization

The Meshing tool acquires the geometric parameters introduced by the user and processes them to create the mesh of the panel.

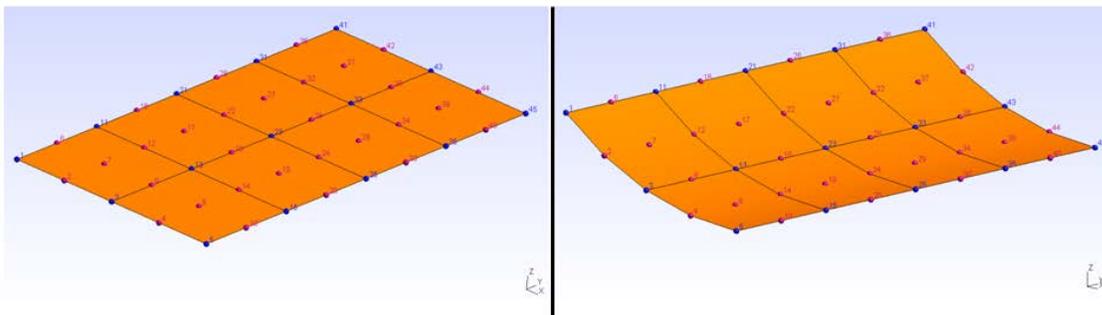


Figure 4. Meshed flat and curved subparts that can be used for skin, webs, and flanges.

Let us illustrate this process with an example. The geometry and mesh generation for a “JC” profile follows. First, the profile is divided into smaller parallelograms. Each of these parallelograms will be assembled in the order specified in Figure 5.

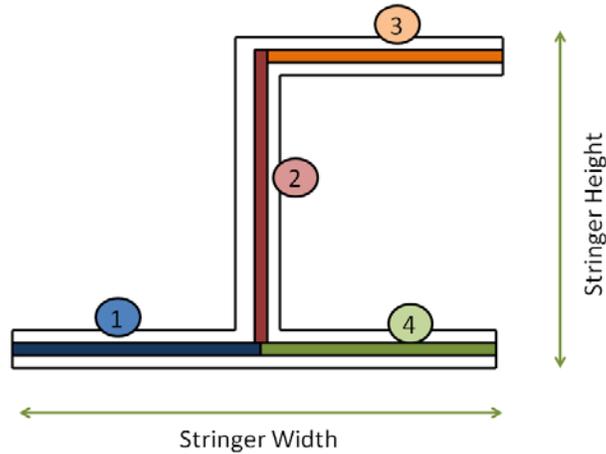


Figure 5. Sketch of the JC profile with subparts labelled in the order in which they are assembled.

Subpart #1 in Figure 5 corresponds to edge 1 in Figure 6. Then, it is extruded along the edge 4 direction to generate the panel length. After that, nodes are generated based on seeding parameters provided by *MeshParameters.cs*. Finally, elements are defined in the sequence shown in Figure 6.

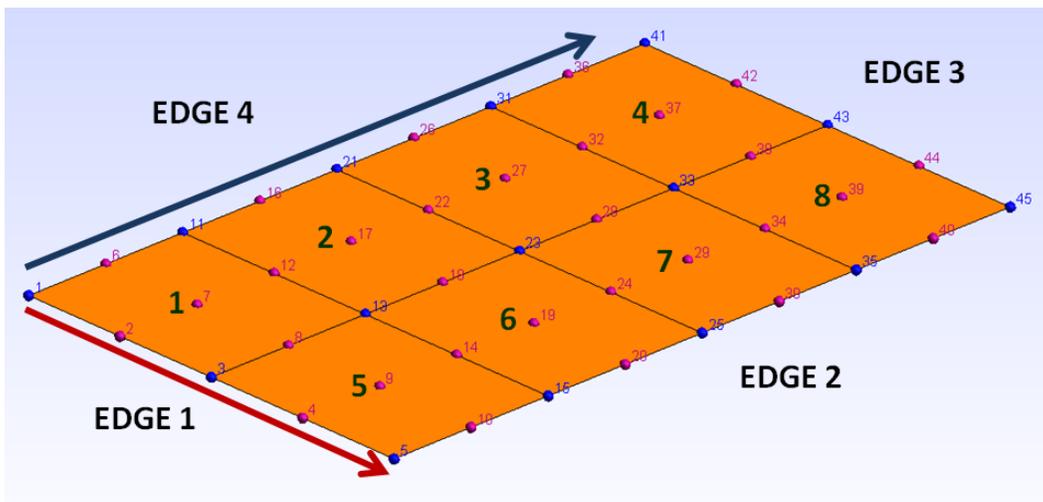


Figure 6. Nodes and elements definition in a flat subpart.

The procedure is repeated for all subparts. To ensure correct assembly of the subparts, nodes in edges 2 or 4 must be shared by two consecutive parallelograms. For example, nodes in edge 2 of the first subpart are the same as those in edges 4 of subparts 2 and 4.

3D visualization of model setup such as shown in Figures 4,6, as well as results (e.g, Figure 7) is accomplished using WebGL [3].

3.4 Eigenvalue Solver

The solver used is BMI3 [4]. It uses a 9-node isoparametric, fully integrated element with relaxation of the Kirchoff condition [5,6]. Relaxation is necessary in order to avoid locking, since fully integrated elements result in excessively stiffened elements.

On the other hand, full integration is used to avoid zero-energy modes, which appear if using reduced integration [7]. Implementation of full integration makes it possible to analyze panels with high aspect ratios (thickness to in-plane dimensions ratio) without any revelation of zero-energy modes.

Zero-energy modes are unrealistic modes of deformation, typically in the form of corrugated deformation of the panels. The corresponding buckling loads are also unrealistic. Unless the design engineer visualizes the deformation, he/she may be misled by erroneous results. On an interactive session, the user wants to concentrate on the design, not on checking the software for numerical errors such as zero-energy modes. Therefore, checking becomes a distracting burden on the designer. A fully integrated element eliminates this problem.

The drawback of fully integrated elements is that they may exhibit locking. As the panels become thinner, the intralaminar shear stress constraint, which is enforced exactly, may lead to zero deformation everywhere. This is called locking and it is the motivation for reduced integration, but reduced integration allows zero-energy modes. The solution is to relax the enforcement of the intralaminar shear stress constraint [5,6].

On an interactive design, the solution must be calculated in real time. This requires a fast eigensolver. To be fast, most eigensolvers are restricted to finding positive eigenvalues. This is fine when the eigenvalues represent vibrations frequencies or buckling loads, both of which must be positive, assuming the reference load is applied in compression. This assumption breaks down when trying to calculate the buckling load of a plate (or panel) under shear because the response of a plate under shear is indifferent to the sign of the shear stress/load. That means that the eigenvalue problem has pairs of eigenvalues with opposite signs. A standard eigensolver is incapable of solving this problem. A new, fast eigensolver was developed to address this problem.

The problem is to solve for the smallest positive eigenvalue e of the system

$$(\mathbf{K} - e \mathbf{M}) \mathbf{u} = 0 \quad [1]$$

where \mathbf{K} is positive definite and \mathbf{M} is not.

For a plane under shear load, there are both positive and negative eigenvalues. The problem is that fast eigensolver algorithms are best at finding the smallest/highest eigenvalues, which would be a very large negative/positive number, and not what the designer needs.

Lapack's SGBVX [11] can solve for both positive and negative eigenvalues but it is extremely slow because it solves for all eigenvalues. Bathe's subspace iteration [12] is fast, but all eigenvalues must be positive.

Therefore, we propose to transform the problem as follows. Multiply by $-1/e$ and rearrange as

$$(\mathbf{M} - e' \mathbf{K}) \mathbf{u} = 0 \quad [2]$$

Then, search for the highest e' , which is positive, to then compute the desired eigenvalue $e=1/e'$. The smallest positive eigenvalue represents the lowest buckling load multiplier, and it is what the designer wants.

To develop the algorithm, first rewrite Eq. (2) as

$$\mathbf{A} \mathbf{u} = e' \mathbf{u} \quad [3]$$

with $\mathbf{A}=\mathbf{K}^{-1} \mathbf{M}$. Now, the power method finds the largest eigenvalue (and its eigenvector) by iterating on

$$\mathbf{x}^k = \mathbf{A} \mathbf{x}^{k-1} \text{ for } k = 1, 2, \dots \quad [4]$$

But it is not necessary to invert \mathbf{K} . Instead, solve

$$\mathbf{LU} \mathbf{x}^k = \mathbf{M} \mathbf{x}^{k-1} \quad [5]$$

where \mathbf{LU} is the Cholesky factorization of \mathbf{K} . It is very fast.

4. RESULTS

This section addressed the accuracy of the results provided by the proposed tool. First, using Classic Plate Theory, some of the simplest cases whose approximate solution is known will be subject to analysis. Second, results obtained after analyzing a curved plate will be compared to those provided by commercial Finite Element software found in literature. Finally, in order to validate the application successfully, laboratory tests of stiffened panels subject to compression will be simulated with CADEC so results can be compared to experimental data found in the literature.

4.1 Composite Plates

A plate is a flat structural element having two dimensions much larger than the thickness. For most shell problems, the in-plane stress resultants N_x , N_y and N_{xy} can be obtained using only equilibrium equations. However, this is not the case for plates, in which moments M_x , M_y and M_{xy} cannot be found from the applied loads without specifying the material and the thickness. For this reason, in preliminary design some approximate methods which are valid only for particular cases are used. Normally this preliminary design is followed by refined Finite Element Analysis (FEA). Since the aim of our tool is to help in the preliminary design, it should obtain similar results than those derived from the theoretical approximated expressions.

The first objective is to verify our tool's performance for analyzing flat plates under compressive loading. Different plate dimensions as well as BC and loading layouts will be considered, and the results will be compared to those computed with Classical Plate Theory (CPT). The analytical expressions have been obtained from the literature [7]. Since there are no stringers, BC and loads will be applied just on the skin.

The solutions predicted by the expressions found in the literature [7] are based on the assumption that the laminates are symmetric and especially orthotropic with $B_{ij} = 0$ and $D_{16} = D_{26} = 0$. For balanced symmetric laminates with nonzero values for D_{16} and D_{26} , the equations may predict erroneous results.

The same laminate will be used for all these cases. It will be formed by glass/epoxy laminas with the following properties: $E_1=97.30$ GPa, $E_2=6.99$ GPa, $\nu_{12}=0.28$, $G_{12}=2.964$ GPa. The properties of the laminate are summarized as follows: Stacking sequence [90/0]_s and lamina thickness=0.1 mm.

For the material properties and LSS given, the following components of the bending stiffness matrix [D] are calculated. Dimensions are introduced in millimeters, and loads in N/mm. Therefore, all values of [D] are reported in MPa mm³. $D_{11}=98.040$, $D_{12}=10.504$, $D_{22}=461.328$, $D_{16}=D_{26}=0$, and $D_{66}=15.808$ MPa mm³.

4.1.1 Flat Plate with Simply Supported Boundary

The buckling load per unit length of a flat plate simply supported around the boundary and subject to one direction loading can be computed with Eq. 11.4 in [10]

$$N^{CR} = \frac{\pi^2 D_{22}}{b^2} \left[m^2 \frac{D_{11}}{D_{22}} \left(\frac{b}{a} \right)^2 + \frac{2(D_{12} + 2D_{66})}{D_{22}} + \frac{1}{m^2} \left(\frac{a}{b} \right)^2 \right] \quad [6]$$

where m is the number of waves of the buckled shape along the loading direction. This number can be computed as the nearest integer to the real number R_m

$$R_m = \left(\frac{a}{b} \right) \left(\frac{D_{22}}{D_{11}} \right)^{0.25} \quad [7]$$

Table 1. Loads, BC, and geometry for plate A

Geometry		Boundary conditions		Loads	
				Nr	Ns
Curvature	None	Side 1	SS	0	0
# of stringers	0	Side 2	SS	1	0
Width (A) [mm]	100	Side 3	SS	0	0
Length (B) [mm]	100	Side 4	SS	1	0

The input data for this case is given in Table 1, which results in $R_m=1.47$, so that $m=1$ (first mode). The solution predicted by the theoretical expressions is: $N^{CR}=0.635$ and the results

obtained with SPD is: $N^{CR}=0.636$ N/mm, with a difference of 0.12%. If the size of the plate is such that $a \gg b$, then the buckling load can be obtained by using Eq. 11.6 in [10]

$$N^{CR} = \frac{2\pi^2}{b^2} (\sqrt{D_{11}D_{22}} + D_{12} + 2D_{66}) \quad [8]$$

If $a=500$ and $b=100$, the solution predicted by the theoretical expressions is: $N^{CR}=0.503$ and the result obtained with CADEC is: $N^{CR}=0.523$ N/mm, with a difference of 3.99%.

4.1.2 Flat Plate with Clamped Boundary

The buckling load per unit length of a flat plate clamped around the boundary (with in-plane displacements enabled) and subject to one direction loading can be computed with Eq. 11.11 in [10], as long as the dimensions satisfy that $a/b > 4$.

$$N^{CR} = \frac{\pi^2}{b^2} [4.6 \sqrt{D_{11}D_{22}} + 2.67 D_{12} + 5.33 D_{66}] \quad [9]$$

Table 2. Loads, BC and geometry input for plate B.

Geometry		Boundary conditions		Loads	
				Nr	Ns
Curvature	None	Side 1	CC	0	0
# of stringers	0	Side 2	CC	1	0
Width (A) [mm]	500	Side 3	CC	0	0
Length (B) [mm]	100	Side 4	CC	1	0

The input data for this case is given in Table 2, which results in a theoretical critical buckling load: $N^{CR}=1.076$, and the result obtained with CADEC is: $N^{CR}=1.217$ N/mm, with a difference of 13.07%.

4.2 Stiffened Panels

Two flat stiffened panels manufactured with composites and subject to uniaxial compression are simulated and compared to experimental data [7,8]. One of the panels uses closed section hat stiffeners. The other uses open section JC stiffeners.

Both panels were manufactured with Hercules unidirectional IM7 carbon fiber pre-impregnated with Cytec's Rigidite 5250-4 resin (graphite/epoxy). The mechanical properties are shown in the Table 3. The laminate stacking sequences are shown in Table 4. The thickness of each ply is 0.125 mm.

Table 3. Ply properties for the Hat- and JC-stiffened panels.

Lamina stiffness properties		Lamina strength values	
E_{1T} (GPa)	179.5	F_{1T} (GPa)	3.068
E_{1C} (GPa)	138.4	F_{1C} (GPa)	1.684
E_{2T} (GPa)	10.6	F_{2T} (GPa)	0.0482
E_{2C} (GPa)	13.3	F_{2C} (GPa)	0.2576
G_{12} (GPa)	5.0	F_6 (GPa)	0.1382
ν_{12}	0.3		

Table 4. Laminate stacking sequence (LSS) for the Hat- and JC-stiffened panels

Skin SS (12 plies)	Bottom Flange SS (5 plies)	Top Flange SS (26 plies)	Web SS (12 plies)
$[\pm 45/0/\pm 45/90]_S$	$[\pm 45/90/\pm 45]$	$[\pm 45/0_4/90/0_3/\pm 45/0]_S$	$[\pm 45/90/0/\pm 45]_S$

4.2.1 Hat Stiffened Panel

The panel geometry consists of a flat skin with 3 hat stiffeners (Table 5).

Table 5. Dimensions of the Hat-stiffened panel.

Panel dimensions [mm]		Stiffener geometry	
Curvature	None	Profile	Hat (Hat)
Length(B)	560	Stiffener Height	39
Width (A)	554	Stiffener Width	102
Number of Stiffeners	3	Top Flange Width	41
Pitch	203	Bottom Flange Width	25

Once the material properties, LSS, and panel geometry are given, it is necessary to represent accurately the loading and BC so that they are as similar as possible to the laboratory test conditions. In the experiments, both panels were subject to a linearly increasing, distributed, compressive load on sides 1 and 3, until buckling was observed.

A unit distributed, compressive load is applied to the model so that the eigenvalue corresponds to the buckling load. Edges 2 and 4 (parallel to stiffener direction) are free, and sides 1 and 3 are clamped. Based on test image, we assume that the panel is loaded only on the skin.

The lowest eigenvalue is $\lambda_1=170.03$ (dimensionless). The buckling load is

$$P^{CR} = \lambda_1 N_r L \quad [10]$$

where

λ_1 : first eigenvalue.

N_r : load per unit length applied to the panel.

L : total length where N_r is applied. (in this case, length of sides 1 or 3)

resulting in

$$P^{CR} = 170.03 \cdot 1 \text{ N/mm} \cdot 554 \text{ mm} = 94.2 \text{ kN} \quad [11]$$

On the other hand, the experimental buckling load [5,6] is 94.3 kN, with a difference of only 0.1%. Moreover, it is noted in the experiment [5] that the skin bay (intermediate skins) buckle into five half-wave lengths. This coincides with the buckling mode predicted by our application (Fig. 7).

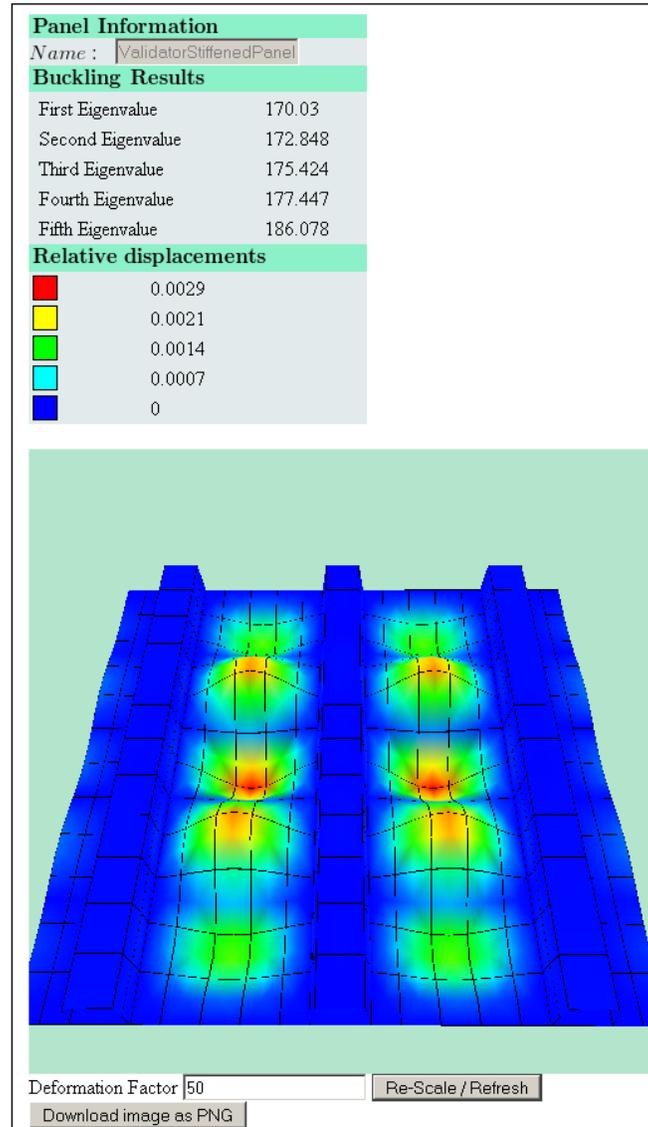


Figure 7. Buckling results of the stiffened panel with three hat stiffeners using CSP.

4.2.2 JC Stiffened Panel

The panel geometry consists of a flat skin with three JC stiffeners (Table 6) [6].

Table 6. Dimensions of the JC-stiffened panel.

Panel dimensions [mm]		Stiffener geometry	
Curvature	None	Profile	JC
Length(B)	480	Stiffener Height	30
Width (A)	450	Stiffener Width	50
Number of Stiffeners	3	Top Flange Width	20.4
Pitch	160	Bottom Flange Width	20.4

For simplicity, the top flange of the JC stiffener is approximated as having one-half of the stiffener width. The results are not affected significantly by this approximation. The lowest eigenvalue is $\lambda_1=132.39$ (dimensionless). The critical buckling load is calculated with equation (1) as follows

$$P^{CR} = 132.39 \cdot 1 \text{ N/mm} \cdot 450 \text{ mm} = 59.58 \text{ kN} \quad [12]$$

The experimental buckling load [6] is 58.1 kN, which results in a difference of only 2.54%.

5. CONCLUSIONS

The application generates the geometry and mesh for multiple configurations of stiffened panels from a few input parameters and performs FEA on the fly, providing buckling load and the deformed shape interactively, thus accelerating and simplifying the design process.

The application allows a wide range of possible panel configurations, including plates, skin-stiffened panels, and integrally-stiffened panels. Flat or slightly curved panels may be chosen.

One of the main problems regarding FEA is the influence of mesh density. Finer mesh provides more accurate results, but increases the time required to solve the problem. Since CADEC is a web application where the user is waiting for the solution, solution time must be short. After running hundreds of different cases, the mesh generator tool was configured to ensure accurate results as rapidly as possible. Depending on the geometry of the panel, the tool is capable of increasing or decreasing mesh density to stay below 60 seconds response time.

To speed up the design process, some of the parameters are simplified. For example, pitch is assumed to be constant, and in some profiles bottom and top flanges are assumed to have the same width. During the preliminary design stage, it is common to perform this kind of assumptions, while the actual geometry is analyzed more accurately after a satisfactory preliminary design is completed.

Since the problem is defined in terms of a few input parameters, it is possible to perform design modifications quickly without having to develop a new FE model for each modification. This is especially valuable during the preliminary design stage, when most of the geometrical changes take place.

The implementation of the application in a web environment allows the user to work on the design from any location and using any type of web-enabled device, while the database stores all data for a later use, thus increasing accessibility.

The visualization tool can zoom in or out, rotate, and move the panel to analyze every detail of the deformed structure. Color range highlights the areas where larger displacements take place. Having instant visualization on the web page presents information faster than standard post processing options.

6. REFERENCES

1. Barbero E. J., *CADEC (Computer Aided Design Environment for Composites)* [homepage on the Internet]. Morgantown: Mechanical and Aerospace Engineering Department of West Virginia University; 2010 [updated 1 March 2014, cited 12 March 2014]. Available from: www.cadec-online.com.
2. Cosso F. A., Barbero E. J., Computer Aided Design Environment for Composites. SAMPE 2012 Conference and Exhibition. 2012.
3. An introduction to WebGL [homepage on the Internet]. [updated 13 October 2011, accessed 28 January 2014]. Available from: <http://dev.opera.com/articles/view/an-introduction-to-webgl/>
4. Barbero E. J., Godoy L. A., Raftoyiannis I.G., *Finite elements for three-mode interaction in buckling analysis*. International Journal for Numerical Methods in Engineering. 1996; 39(3): 469-488.
5. Liu Y. J., Development of the Min-N Family of Triangular Anisoparametric Mindlin Plate elements [Ph.D. dissertation]. Manoa: University of Hawaii; 2002.
6. Liu Y. J., Riggs H. R., The MIN-N family of pure-displacement, triangular, Mindlin plate elements. Structural Engineering and Mechanics. 2005; 19(3): 297-320.
7. Barbero E. J., Finite Element Analysis of Composite Materials Using Abaqus. CRC Press; 2013. pp. 91-194.
8. Elaldi F., Buckling, Post-buckling and Failure Analysis of Hat Stiffened Composite Panel. Proceedings of 8th International Fracture Conference. 2007.
9. Elaldi F., Structural Efficiency and Post-buckling Strength of J- and Hat-stiffened Composite Panels. Journal of Reinforced Plastics and Composites. 2010; 29(10).
10. Barbero E. J., Introduction to Composite Materials Design. 2nd ed. CRC Press. 2010 pp. 385-393.
11. LAPACK User's Guide, 3rd Edition, 1999, <http://www.netlib.org/lapack/lug/>
12. Bathe, K-J, Finite Element Procedures in Engineering Analysis, Prentice Hall, 1982.