Graphical User Interface (GUI) Application to Perform Stability Analysis in ANSYS

Rafael V. Oliveira[1], Janine D. Vieira[1]

[1]*Dept. of Civil Engineering, Universidade Federal Fluminense*
Rua Passo da Pátria, 156, São Domingos, 24210-240, Niterói, Rio de Janeiro, Brasil
ravieira@id.uff.br, janinedv@id.uff.br

Abstract. There is few commercial software, based on finite element method, that are able to perform buckling analysis. One of the most popular commercial software based on FEM that can perform this kind of analysis is the ANSYS, and is considerably complex. Due to a big range of possibilities and resources that the program offer, sometimes the interest in some specific analysis, like stability, can be hard to perform because of the difficulty in the input data. Also, sometimes it is necessary to run a set of analysis with several profile dimension, so this task can be extremely difficult with a not friendly GUI of drawing. Then, the aim of this work is to give a GUI application that offer to the users an easy and friendly way to run buckling analysis in profiles using ANSYS. The application was developed using Python Language and the framework Electron and can perform linear and non-linear analysis. Moreover, was carefully developed to be easy to use to facilitate access and speed up numerical analysis of buckling for researchers and interested people. At the end, are presented the application use in a set of analysis of flexural stability of pultruded glass fiber reinforced polymer I-sections.

Keywords: GUI, Python, Stability, ANSYS, GFRP.

1 Introduction

Pultruded glass fiber reinforced polymer (GFRP) profiles are widely applied at industrial plants, especially on those subject to aggressive environmental such as oil and gas exploration. Considered as open thin-walled sections, GFRP beams and columns are susceptible to instability problems, which is one of the criteria control design of GFRP members and structures (Barbero and Tomblin 1994 [1], Godoy et al 1995 [2]). Therefore, the computer has been a very important aid in solving this kind of engineering problems. Nowadays there are several software that can be used to run instability analysis of which Ansys detaches in numeric analysis area. This program allows to solve static, dynamic, instability and other problems. However, its interface is not very friendly and can be difficult to run a set of analyses with different parameters, like different spans or geometries. So, in the face of this, the aim of this work was to developed a graphical user interface (GUI) to facilitate running instability analysis in Ansys with focus in local and global buckling in profiles. In this way, the developed program allows a range of possibilities of section shapes without needing any draw by the user.

2 The InstabiliTool

The use of the InstabiliTool app (GUI) requires a legally licensed of ANSYS 17.0 or newer and the student license is supported (using student license, all the limitation of this license will be maintained). InstabiliTool is licensed under the MIT license created by Massachusetts Institute of Technology and this app makes no commercial claim over Ansys whatsoever. This tool extends the functionality of ANSYS MAPDL by adding a Graphical User Interface (GUI) without changing the core behavior or license of the original software. At first, there are only support for Windows system, but support for other platforms can be added easily in the future. The InstabiliTool home interface is shown in Figure 1.

The InstabiliTool is powered by Python and Electron. The Framework Electron is used in the interface. It was chosen because your rich features which include the implementation of automated update and the development of a modern GUI with web development technology. Some of the famous desktop apps built with Electron are

Facebook Messenger, Twitch and Microsoft Teams. The Python language is used to generate a Mechanical APDL instance in the background and run in it the model defined on the interface. To do this connection is used the Pyansys library created by Kaszynski [3]. Other libraries used can be checked at InstabiliTool webpage.

The app allows to perform linear and non-linear instability analysis. In the linear one is used the eigen values buckling analysis (default of ANSYS). In contrast, the non-linear analysis uses the static analysis of the software with a step-load sequence process (geometric nonlinearity). Also, it is possible to analyze isotropic, orthotropic and anisotropic material. However, only linear material is allowed. It is implemented in the app an easily way to inform the geometry parameters of the section. The common sections that can be used is: I-section, Tubular, C-section, stiffened C-section, Rack section, Angle and Plate.

The model can be discretized in triangular or rectangular meshes and is allowed both mapped and free methods. It is possible to apply boundary conditions to any node in the extension of the profile. The load can be axial or bending moment. To the axial load is allowed to be point load or a distributed load along the whole cross section. The point load can be added at the centroid or with a defined eccentricity. To the bending moment is allowed to be a direct moment or an indirect moment with four-point flexure or three-point flexure.

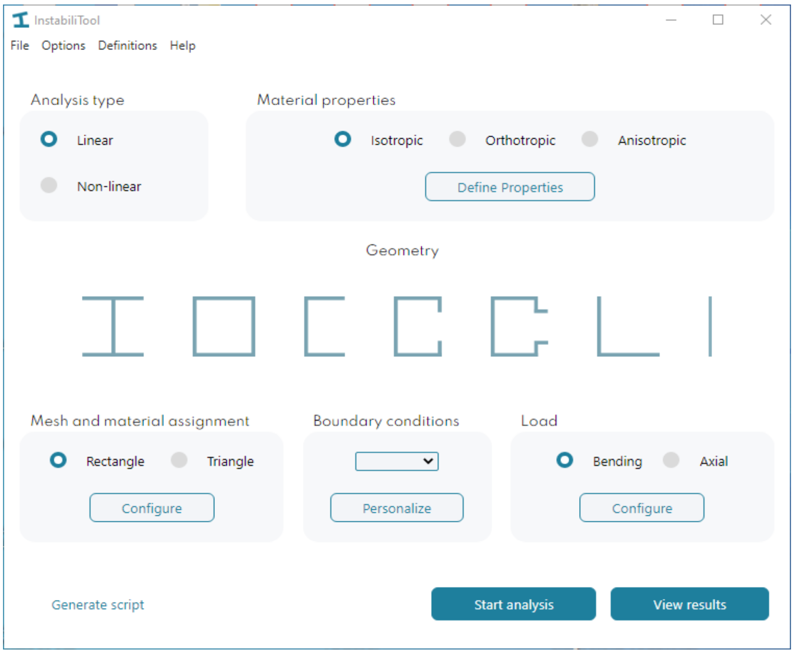

Figure 1. InstabiliTool Interface

3 App validation

3.1 Linear Analysis

For the validation of the app, two examples from the literature were analyzed. The first example, a column, was taken from Campos [4]. The second one was a plate from Hassan and Kurgan [5]. The material properties and the critical load results of the structures analyzed is presented in Table 1. The deformed shapes are present in Figure 2.

Table 1. Material properties and critical load results

Analysis	E (GPa)	v	Reference results	InstabiliTool results	Reference/InstabiliTool
RS-5 by [4]	21.2	0.3	66.74 kN	66.78kN	1.00
Full plate by [3]	70.0	0.3	2.199 N/mm	2.207 N/mm	1.00

The full plate has length 1.5m, width 1.0m, thickness 2mm and the load is applied on the shorter edges. The column has length 1397 mm and your section dimensions are presented in Figure 2 bellow.

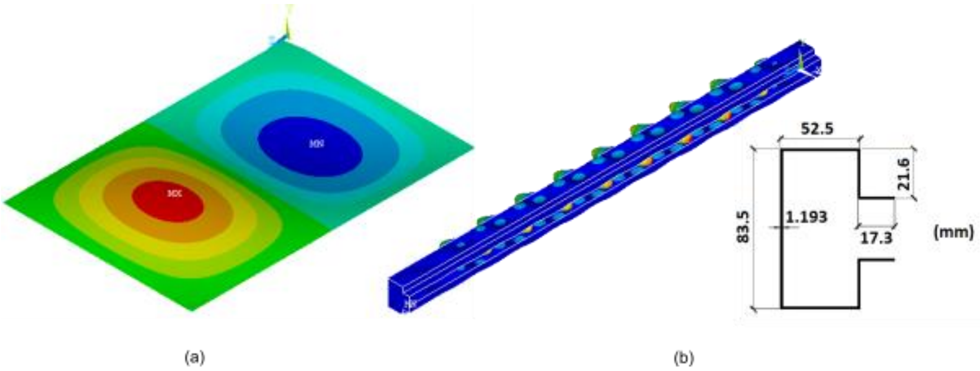

Figure 2. (a) Deformed shape of plate; (b) Deformed shape of RS-5 and its section dimensions

3.2 Nonlinear Analysis

In the nonlinear analysis validation, the plate of [5] were used. Then the displacement normal to surface (in the point of maximum displacement) are plotted against the axial load, as shown in Figure 3. The result is 1,03% lower than the value found by [5], which is 2200 N. Then, considering all the validation analyses, the results were considered satisfactory and the app were validated.

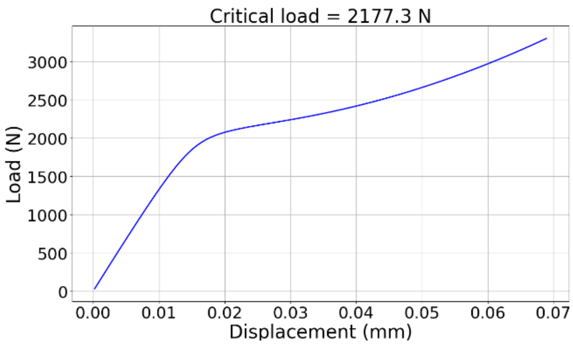

Figure 3. Displacement against axial load

4 GFRP I-Section stability analysis using InstabiliTool

Using the InstabiliTool app to perform instability analyses in pultruded glass fibre-reinforced polymer (GFRP) profiles subjected to flexure. Five different beams were studied. The dimensions and the geometries of I-Section profiles used are shown in Table 2. Were performed both linear and non-linear instability analyzes and the results were compared with the experimental results obtained by Vieira, Liu and Harries [6] (FLB1 to FLB4) and Vieira, Vieira and Cardoso [7] (FLB5) in their extensive experimental studies. The mechanical properties used in the model, based on the studies of [6], [7] and Liu, Harries and Guo [8], are presented in Table 4.

Table 2. I-Sections profiles dimensions used

Profile	Span (mm)	Shear span (mm)	Nominal dimensions
FLB1	2900	1000	152,4 x 152,4 x 6,35
FLB2	2600	900	152,4 x 152,4 x 6,35
FLB3	2200	800	152,4 x 152,4 x 6,35
FLB4	1800	700	152,4 x 152,4 x 6,35
FLB5	1100	310	102,0 x 102,0 x 6,00

To evaluated Flange Local Buckling (FLB) a simply-supported four-point flexure having 5 different spans were modelled. ANSYS Shell181 elements were used, and the mesh size was set to 10mm. To define the mesh size, it was performed a convergence analysis using the biggest beam, FLB1. The results, as shown in Table 3, indicate that 10mm is satisfactory. Boundary conditions to represent lateral support to prevent lateral torsional were provide at both reactions points and near the load points, as shown in Figure 4. In the experimental procedure conducted by [6], the load application occurs in the whole width of the top flange. Then, to the beams by [6] was forced the same displacement to all nodes at each line of load in the top flange.

Table 3. Mesh convergence analysis.

Mesh dimension (mm)	100	75	50	25	10	7.5	5
Critical load (kN.m)	10.8	11.6	11.0	10.8	10.8	11.0	11.3
Variation (%)	-	7.4	-5.2	-1.8	0.0	1.9	2.7

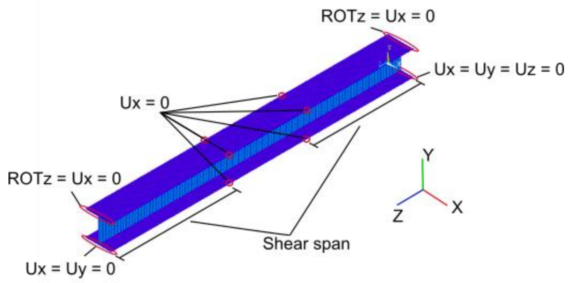

Figure 4. Boundary conditions used in the model

Table 4. Mechanical properties by [1] ,[2] and [5]

Mechanical properties	Values by [1] and [5] (MPa)		Values by [2] (MPa)	
	Flange	Web	Flange	Web
E_{LT}	24490	26470	32653	26350
E_{LC}	31219	31250	30401	28328
E_{LF}	18750*	18750*	19854	17425
E_{TT}	8289	5500	7867	7867
G_{LT}	2882	3100	2298	2375

* Longitudinal bending modulus by [5]

The properties of Table 4 were used. It was conducted three analyses of local buckling on the flange. In the first analysis was used the longitudinal modulus of elasticity in compression as recommended by guidance document EUR 27666 (2016) [9]. For the second one was taken the minimum modulus of elasticity in tension and in compression as recommended by ASCE Pre-standard (2010) [10]. In the end, the longitudinal modulus of elasticity in flexure was used because curvature arises in the top flange when the instability phenomenon occurs. Moreover, in nonlinear analyses to represent the imperfection were used the eigenvector resulting of bifurcation analysis. The maximum value of imperfection adopted was 10% of the profile thickness. In nonlinear analysis, the onset of bucking was defined when the displacement changes significantly more than the load applied.

4.1 Results

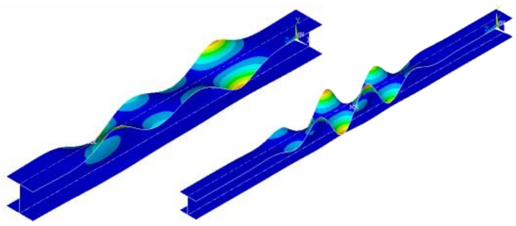

Figure 5. Deformed shape of FLB5 and FLB1 (left to right)

The results of analyses using modulus elasticity in compression (E_c), in flexure (E_f) and taken the minimum between the modulus value in compression and in tension (E_{min}) are shown in the Table 5. As it can see, the linear analyses conducted to prediction closer to experimental results in the case of E_f and E_{min}. For the nonlinear analysis the better results were reached to E_c. Furthermore, as expected, the non-linear results are lower than the linear ones for the whole set of analysis. Deformed shape of FLB5 and FLB1 are shown in Figure 5.

Table 5. Analyses results

Profile	Critical moment (kN.m)												
	Exp.	Linear analyses						Nonlinear analyses					
		E_c	Num. /Exp.	E_f	Num. /Exp.	E_{min}	Num./ Exp.	E_c	Num. /Exp	E_f	Num. /Exp	E_{min}	Num. /Exp.
FLB1	9.3	10.8	1.17	8.5	0.92	9.8	1.05	8.3	0.89	6.6	0.71	8.7	0.94
FLB2	9.3	10.9	1.17	8.7	0.93	10.0	1.07	9.7	1.04	8.3	0.89	6.6	0.71
FLB3	10.7	11.6	1.08	8.9	0.83	10.4	0.97	8.5	0.79	8.0	0.75	6.7	0.62
FLB4	10.1	12.1	1.19	9.1	0.90	10.7	1.06	10.7	1.06	7.9	0.78	5.6	0.55
FLB5	5.5	6.6	1.19	5.2	0.95	6.6	1.19	6.1	1.10	5.0	0.90	5.0	0.90
Aver.			1.16		0.91		1.07		0.98		0.81		0.74

5 Conclusions

Through the validation analysis, the software InstabiliTool was found to predict, with great accuracy, the results of instability analysis conducted directly on Mechanical APDL interface, as expected. As for the instability analysis of GFRP profiles, the both linear and nonlinear results were relatively close to the experimental results. In addition, the nonlinear analyses using modulus in compression detaches in the set of analysis with an average difference of 2% in relation to experimental results. In face of the discrepancy between the results of linear and nonlinear analyses, it can be concluded that geometric nonlinearity must be considered. Furthermore, using the flexural modulus, the results found were very conservative, especially for the nonlinear analyses.

For the more curious users, the InstabiliTool provides the Python and/or MAPDL script of the analysis created with the GUI. This can be useful for those who want to explore features that are not included in the app.

Acknowledgements. The first author thanks CNPq (Conselho Nacional de Desenvolvimento Científico e Tecnológico) for financial support through scholarship for this research.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] Barbero, E., & Tomblin, J. (1994). A phenomenological design equation for FRP columns with interaction between local and global buckling. Thin-Walled Structures, 18(2), 117-131
[2] Godoy, L. A., Barbero, E. J., & Raftoyiannis, I. (1995). Interactive buckling analysis of fiber-reinforced thin-walled columns. Journal of Composite Materials, 29(5), 591-613.
[3] Kaszynski, A. pyansys: Python Interface to MAPDL and Associated Binary and ASCII Files. Zenodo, 2020.
[4] Campos R. Análise da interação entre os modos de flambagem local e distorcional em paredes de perfis de aço formados a frio com seção rack sob compressão axial. Dissertação de mestrado, Universidade Federal do Rio de Janeiro, 2020.
[5] Hassan A, Kurgan N. "Modeling and Buckling Analysis of Rectangular Plates in ANSYS". International Journal of Engineering & Applied Sciences, vol. 11, Issue 1, 2019
[6] Vieira J, Liu T and Harries K. "Flexural stability of pultruded glass fibre-reinforced polymer I-sections". Proc. Inst. Civ. Eng. Struct. Build. 171 (11): 855–866, 2017.
[7] Vieira E, Vieira J, Cardoso D. "Local buckling of pultruded GFRP I-section subject to bending". 4th Brazilian Conference on Composite Materials, Rio de Janeiro, 2018.
[8] Liu T, Harries K and Guo Q. "Effects of fiber architecture on flexure properties of pultruded GFRP plates and sections". 9th International Conference on Fibre-Reinforced Polymer (FRP) Composites in Civil Engineering (CICE 2018), Paris, 2018.
[9] L. Ascione, J.-F. Caron, P. Godonou, K. van IJselmuijden, J. Knippers, T. Mottram, M. Oppe, M. Gantriis Sorensen, J. Taby, L. Tromp; Prospect for new guidance in the design of FRP; EUR 27666 EN; doi:10.2788/2230
[10] American Society of Civil Engineers. (ASCE). (2010). Pre-Standard for Load and Resistance Factor Design (LRFD) of Pultruded Fiber Reinforced Polymer (FRP) Structures.