REVISTA Mundi
Engenharia, Tecnologia e Gestão
INSTITUTO FEDERAL
Paraná

FINITE ELEMENT ANALYSIS USING SHELL ELEMENTS ON RETICULATED STRUCTURES

ANÁLISE COM ELEMENTOS FINITOS DO TIPO SHELL EM ESTRUTURAS RETICULADAS

Alexandre Huberto Balbino Selhorst[1]

Rafael Elias Meyer[2]

Tatielen Pereira Costa[3]

Liércio André Isoldi[4]

Abstract: This paper studies the possibility of using SHELL elements on structures that are usually designed using BEAM or BAR elements. Using SHELL elements in such structures, named reticulated structures, opens the possibility to remove material from the structures' members by adding holes. Once the members get holes, the final weight of the structure and material consumption will drop. In order to develop the study, a total of three different structures were analyzed using the software ANSYS®. This software uses the Finite Element Method to solve the equations originated from the designed structure upon it is meshed and the boundary conditions are set. Several results between SHELL and SOLID elements for each one of the structures were compared. The comparison of SHELL model with a SOLID model is used as a verification process, for SOLID elements are known to return accurate values. All of the meshes were tested by the independence mesh study to check its convergence. It is shown that the results are in a very good acceptable range with differences no bigger than 0.1 mm for displacements, and the map of von Mises stresses are pretty similar. Von Mises stresses for Finite Element Analysis for the C-Shaped Truss are shown in a figure comparing the results between the two finite elements used. This figure shows that there are no major differences between the SHELL and SOLID analyses. The Finite Element Analyses results were compared to analytical solutions, also. In this case, a noticeable difference in one structure for von Mises Stress was found. This difference, however, is understandable and reasonable, given the works presented on this paper.

Keywords: Finite Element Analysis (FEA). Finite Element Method (FEM). SHELL elements. Reticulated structures.

---

[1] Graduando em Engenharia Mecânica Empresarial, Universidade Federal do Rio Grande, FURG, Brasil. E-mail: alexandre.selhorst@uol.com.br

[2] Graduando em Engenharia Mecânica Empresarial, Universidade Federal do Rio Grande, FURG, Brasil. E-mail: meyerafael@hotmail.com

[3] Mestranda em Engenharia Mecânica, Universidade Federal do Rio Grande, FURG, Brasil. E-mail: tatielen_costa@hotmail.com

[4] Doutor em Engenharia Mecânica, Universidade Federal do Rio Grande do Sul, UFGRS, Brasil. E-mail: liercioisoldi@furg.br

Resumo: Este trabalho estuda a possibilidade de utilizar elementos do tipo SHELL em estruturas que são normalmente projetadas utilizando elementos dos tipos BEAM ou BAR. O fato de utilizar elementos SHELL em tais estruturas, chamadas de estruturas reticuladas, abre a possibilidade de criar furos nos membros das estruturas. Uma vez que os membros são perfurados, o peso final da estrutura e o uso de material serão reduzidos. Para desenvolver este estudo, um total de três diferentes estruturas foram analisadas utilizando o software ANSYS®. Este programa usa o Método dos Elementos Finitos para resolver as equações originadas a partir da estrutura desenhada tão logo a malha tenha sido gerada sobre ela e as condições de contorno postas. Vários resultados entre elementos SHELL e SOLID para cada uma das estruturas foram comparados. A comparação de um modelo utilizando SHELL com um modelo utilizando SOLID é utilizada como um processo de verificação, pois elementos do tipo SOLID são conhecidos por retornar valores acurados. Todas as malhas foram testadas segundo um teste de independência de malha para checar sua convergência. Os resultados obtidos estão em um bom intervalo de aceitação, apresentando diferenças no deslocamento não maiores que 0.1 mm e os mapas para as tensões de von Mises são bastante idênticos. Tensões de Von Mises para as análises da treliça com seção transversal no formato C são mostradas em uma figura comparando os resultados entre as análises com elementos SHELL e SOLID. As análises por Elementos Finitos também foram comparadas com soluções analíticas. Nesse caso, foi encontrada uma diferença notável para a tensão de von Mises. Essa diferença, entretanto, é compreensível e razoável, mediante aos trabalhos apresentados neste artigo.

Palavras-chave: Análise por elementos finitos. Método dos elementos finitos. Elementos SHELL. Estruturas reticuladas.

1 INTRODUCTION

Finite Element Analysis (FEA) has been used in a wide variety of engineering problems since it was developed. FEA has changed the way engineers design projects, whether they are simple or complex projects (MACKERLE, 2003). This type of analysis is done using the Finite Element Method (FEM) to solve the system of equations that appear once the boundary conditions are set at the model.

Due to its relatively easy implementation, robustness, reliability, time and cost saving, FEA is nowadays used in a large number of engineering projects. Structural, thermal and fluid flow are examples of analyses that are performed by FEA. In all of these three areas, a problem that comes into play is to choose the most suitable type of element to do the analysis. Essentially, the element should return coherent and reliable results, and have a computational cost that is consistent with the problem that is being solved (LOGAN, 2007).

Bearing it in mind, it is common to find papers and designs using simple elements such as BEAM elements, or even simpler elements such as BAR elements, which are named as LINK on ANSYS®. Zheng (2012) used BEAM elements to design and simulate a dome subject to an impact load. Covill et al. (2014) simulated 82 models of road bicycles to understand their vertical compliance and lateral stiffness using BEAM elements. A progressive collapse analysis of a planar frame using BEAM elements was simulated by Kaewkulchai & Williamson (2004). Kim, Doh & Lee (2008), used BEAM and BAR elements to verify the behavior of a cable-tensioned and shaped Hypar space truss.

The problem that may arises from using BAR and BEAM elements is that one may not be able to spot stress concentration areas or perform any type of optimization, since these element types are unidimensional. It might be interesting in some cases to reduce the weight of a structure without reducing its mechanical strength, for instance, in the structure of an airplane.

On the other hand, when the problem is more complex, it is common to find solutions using SOLID elements, which is just fine (XUE, LIU & LI, 2015; KIM, YOON & KANG, 2007). SOLID elements will likely return very reliable values, however, its computational cost may be high when compared to SHELL elements. Conversely, it is important to know whether or not a SHELL element will give reliable results, as in Sadowski & Rotter (2013) that studied the buckling of tubes under different ratios between radius and thickness employing SHELL elements.

This paper intends to evaluate the applicability about the use of SHELL elements to solve problems that are normally solved using BAR and BEAM elements, for instance in reticulated structures. The use of SHELL elements makes changes possible at the section's geometry, for instance, to add holes on the surface of the model in order to reduce weight without compromising its resistance. Using SOLID elements is a way to verify the models, once SOLID elements return reliable results (KHAN &PARHI, 2013).

2 METODOLOGY

As stated before, the usage of SHELL elements in structures that are normally simulated using BEAM or BAR elements will be studied in this paper. The main idea is to evaluate the results of three different structures under different types of loads, and verify whether they return reliable results.

Each structure will be numerically simulated using SHELL and SOLID finite elements. It is well known that the SOLID type element is the most accurate, and returns results comparable with real tests, which make the SOLID elements the choice to serve as a reference result (KHAN & PARHI, 2013). By doing so, the results from SHELL and SOLID elements can be compared in each structure, which could catch eventual non desired behavior from SHELL elements. To avoid the case in which all of the two elements could be returning misleading results and to verify the models, a comparison with an analytical solution will be performed, also.

All of the analyses will be done on ANSYS® software. This software uses FEM, which is a mathematical method to solve differential equations. Initially, FEM divides the structure into smaller pieces that are named as finite elements. Each element is connected to another one by a node, and may have many nodes depending on its shape function and geometry. All the elements together form a mesh.

Every one of the created elements will have an equation. All of these equations are combined into a global equation in order to solve the problem:

$$\{F\}=[K]\{U\} \tag{1}$$

where: $\{F\}$ is the external forces vector, $[K]$ is the stiffness matrix and $\{U\}$ is the unknown nodal displacement vector (LOGAN, 2007).

It is important to note that an element will have a larger stiffness matrix depending on its shape function and geometry, which is increased proportionally to the number of degrees of freedom of the finite element. Regarding geometry, elements can be unidimensional (BAR and BEAM), bi-dimensional (PLATE and SHELL) or tri-dimensional (SOLID).

This paper will not get into the details of FEM. If clarifications are needed, one can consult either Logan (2007), or Bathe (2014). Simulations will be run using SHELL and SOLID elements. A mesh independence study will be performed on each type of element for each structure.

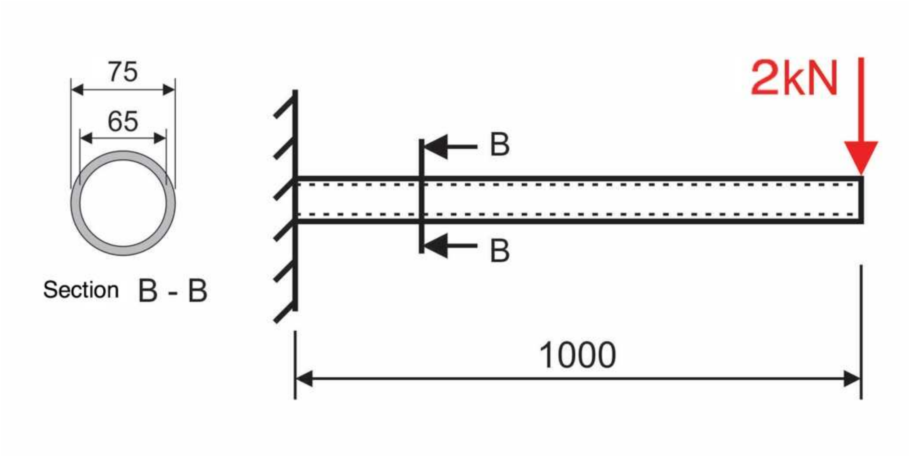
The three structures, their dimensions, boundary conditions and designs are presented in the next sub-sections.

2.1 Cantilever Beam

The first structure is a cantilever beam. This beam will be designed with a tubular section. SHELL and SOLID elements require different designs. The first can be meshed onto a surface, while the second needs a volume. The initial size of the mesh is proportional to the structure’s dimensions.

The boundary conditions and the dimensions for the cantilever beam are presented in Figure 1. This structure is made of a material that has a Young’s Modulus of 68.9GPa, and a Poisson’s ratio of 0.33. All the dimensions are in mm.

Figure 1 - Cantilever Beam with load condition

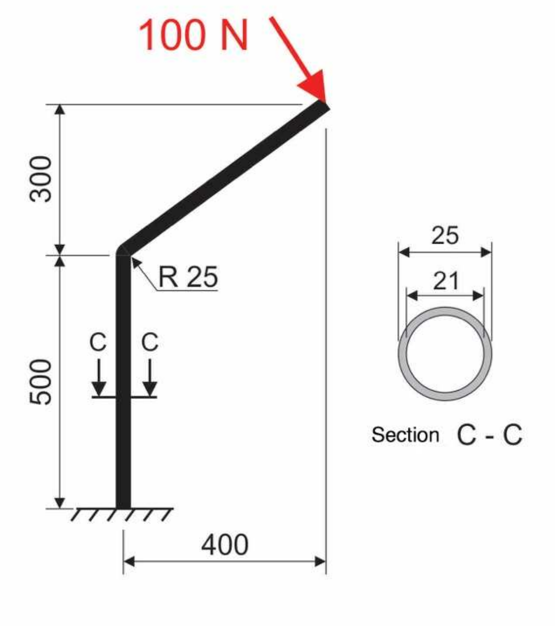


Source: Authors

2.2 Tubular Frame

On their study, Sadowski & Rotter (2013), analyzed only straight tubes. This paper intends to check a tubular frame that is curved. The frame is presented on Figure 2, and is made of a material that has a Young's Modulus of 124GPa and a Poisson's ratio of 0.34. All dimensions are in mm.

Figure 2 - Tubular frame with load condition



Source: Authors

2.3 C-Shaped Roof Truss

A truss is a structure formed by slender straight members usually considered to have pinned joints between members (HIBBELER, 2009). However, due to the high prices, and the friction that would eventually occur, pinned joints are rarely used nowadays (PFEIL & PFEIL, 2009).

The truss is made of a material with Young’s Modulus of 210GPa and a Poisson’s ratio of 0.3. Figure 3 shows the truss and its dimensions, which are given in mm. The members of this structure are assembled as in Figure 4.

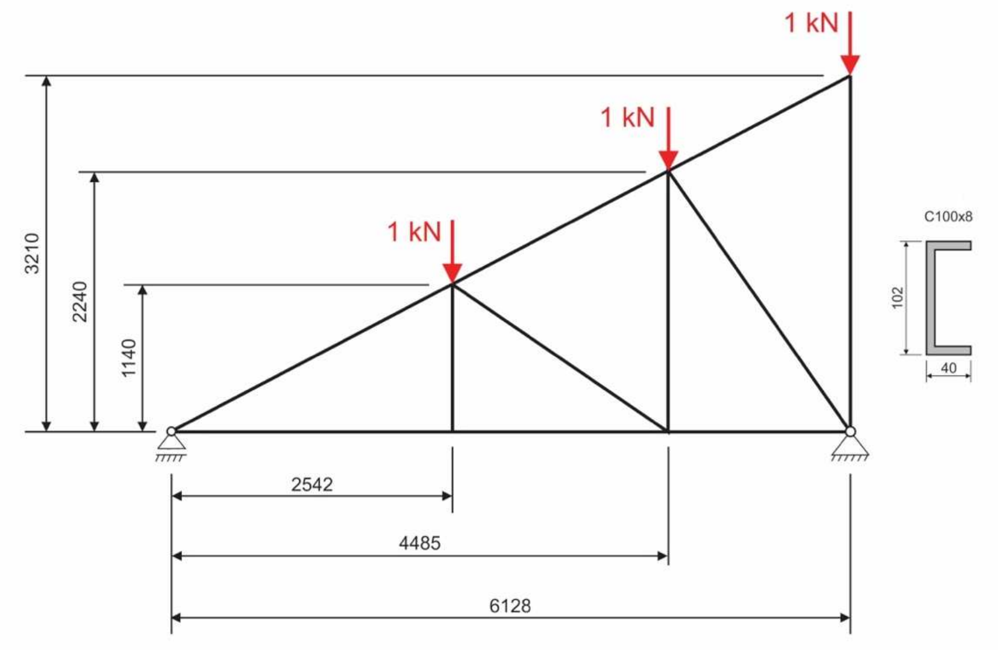


Figure 3 – Roof Truss with load conditions

Source: Authors

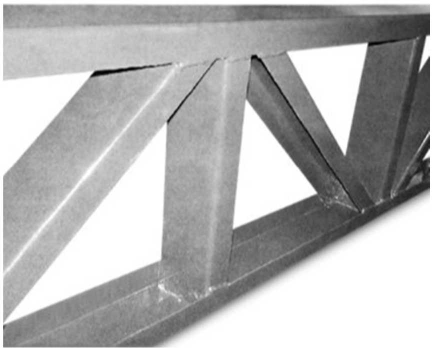


Figure 4 - C-Shaped Truss

Source: https://produto.mercadolivre.com.br/MLB-702711191-trelica-de-3-polegadas-_JM

3 RESULTS AND DISCUSSIONS

3.1 Cantilever Beam

A total of 8 simulations were run using SHELL93, this finite element has 8 nodes, therefore it is a quadratic element. It has 6 degrees of freedom per node: 3 translations and 3 rotations related to the *x*, *y* and *z*-axis (ANSYS RELEASE, 2004)**.** The first mesh considered was generated with quadrilaterals of size of 20 mm. The mesh convergence study is done by taking the previous result minus the actual result over the previous result.

Results for SHELL93 are shown on Table 1. A curve that shows the mesh convergence is presented on Figure 5.

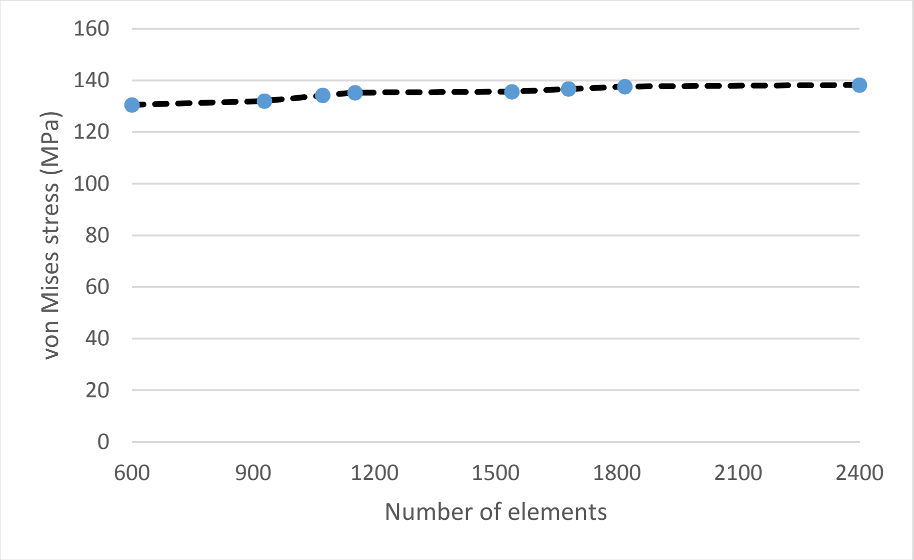
Table 1 – Results for SHELL93 – Cantilever Beam

Number of elements	Maximum deflection (mm)	von Mises stress (MPa)	von Mises stress variation
600	14.417	130.56	-
928	14.417	132.00	-1.10%
1072	14.417	134.22	-1.68%
1152	14.417	135.23	-0.75%
1540	14.418	135.60	-0.27%
1680	14.418	136.68	-0.80%
1820	14.418	137.58	-0.66%
2400	14.418	138.25	-0.49%

Source: Authors

Results of deflections and von Mises stresses were compared with the number of the elements of each of the 8 meshes generated. From these simulations, the best mesh was determined to be the one with 1152 elements and element size of 14 mm. The results were increasing in a slow pace towards the chosen mesh and the next value increment was only 0.27%, which means that only computational costs were being added.

Figure 5 - Mesh convergence – SHELL93 – Cantilever Beam



Source: Authors

Simulations with the element SOLID95 were run after the simulations with SHELL93. SOLID95 is defined by 20 nodes having 3 degrees of freedom in each node (ANSYS RELEASE, 2004). The first element size adopted was 15 mm, and a total of 8 simulations were carried out. These finite elements were tetrahedra instead of hexahedra, due to the fact that hexahedral elements need extra shape functions to work properly on bending dominated problems. Also, it is noteworthy to point out, that tetrahedral elements give basically the same results as hexahedral elements if both are quadratic (WANG, NELSON & RAUCH, 2004).

Table 2 shows the results for the SOLID95 element. Figure 6 represents the mesh convergence; it relates the von Mises stress with the number of finite elements that were used in that specific mesh.

Table 2 - Results for SOLID95 – Cantilever Beam

Number of elements	Maximum deflection (mm)	von Mises stress (MPa)	von Mises stress variation
6510	14.390	112.41	-
9678	14.398	119.06	-5.92%
14982	14.402	120.25	-1.00%
17580	14.403	121.57	-1.10%
22616	14.405	123.57	-1.64%
29277	14.406	125.14	-1.27%
38190	14.408	126.51	-1.10%
55356	14.410	127.74	-0.97%

Source: Authors

After the simulations were run, it was decided that the best mesh for this structure was the one with 9678 elements and element size of 12 mm. The von Mises stress result for the 10 mm size element was slightly bigger than the result for the 12 mm size element. However, this value is only 1% bigger, which would only increase computational costs. Values of von Mises stress for finer meshes were increasing about the same rate of 1%.

A comparison between the solutions given by SHELL93 and SOLID95 is presented on Figure 7. As mentioned before, a comparison with the analytical solution would be made. Analytical solution gives a result of 14.294 mm for displacement, and a result of 110.80 MPa for von Mises stress. These results are obtained, respectively, by (HIBBELER, 2010):

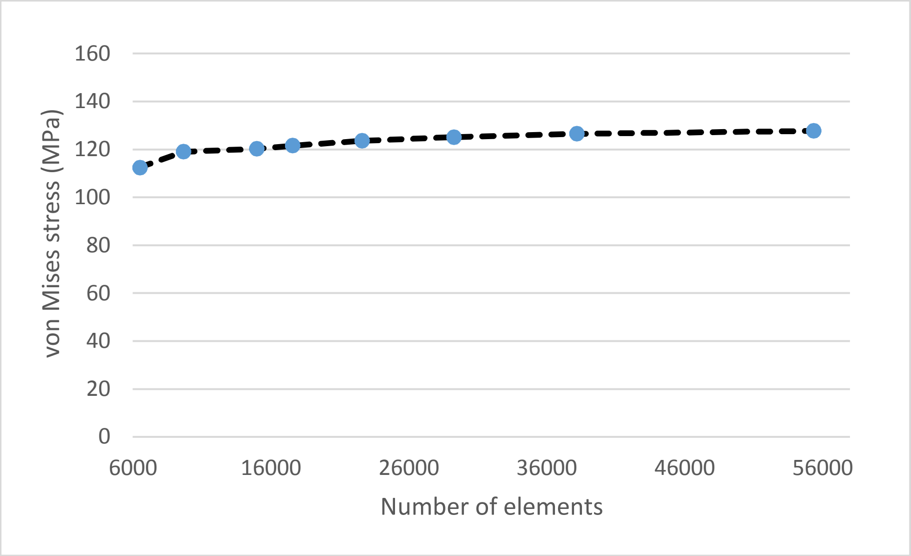
$$\delta=-\frac{PL^3}{3EI} \tag{2}$$

where: *P* is the load at a point on the structure, *L* is the length, *E* is the Young's modulus, and *I* is the second moment of area.

$$\sigma_{Fmáx} = \frac{Mc}{I} \qquad (3)$$

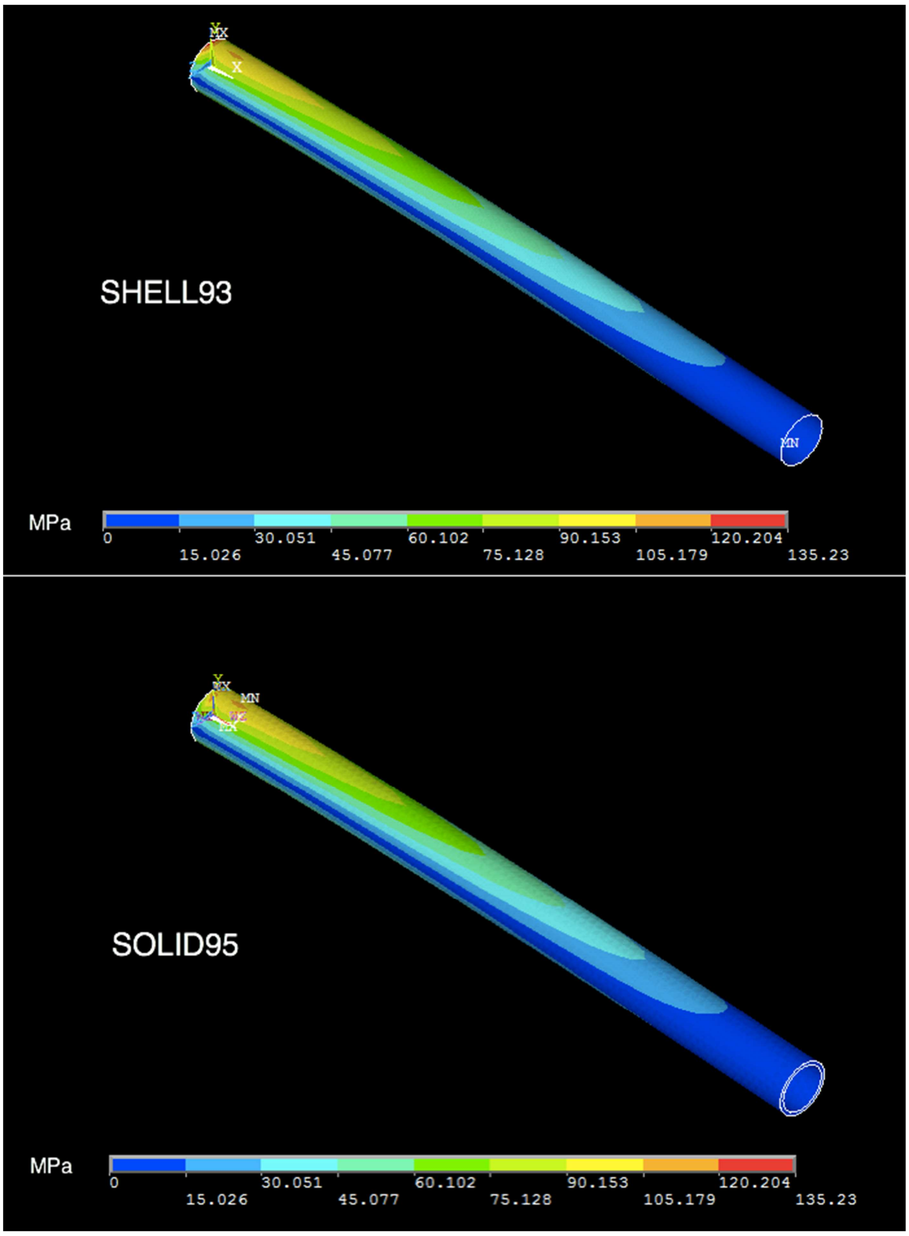
where: *M* is the bending moment, and *c* is the maximum distance away from the neutral axis.

Figure 6 - Mesh convergence – SOLID95 – Cantilever Beam



Source: Authors

Figure 7 - von Mises stress – SHELL93 VS SOLID95 – Cantilever Beam



Source: Authors

The range of stress' scale is the same, and both elements behavior about the same, which can be inferred from Figure 7.

3.2 Tubular Frame

Six simulations were performed using SHELL93 and SOLID95 elements, and the first element size to be taken to do both simulations had a size of 3 mm, quadrilaterals for SHELL93, and tetrahedral for SOLID95. Bigger elements would not be suitable on this structure, for it is as tube with a relative small diameter and a thin wall.

The results for SHELL93 element are presented on Table 3. Figure 8 shows the mesh convergence.

Table 3 - Results for *SHELL93* – Tubular Frame

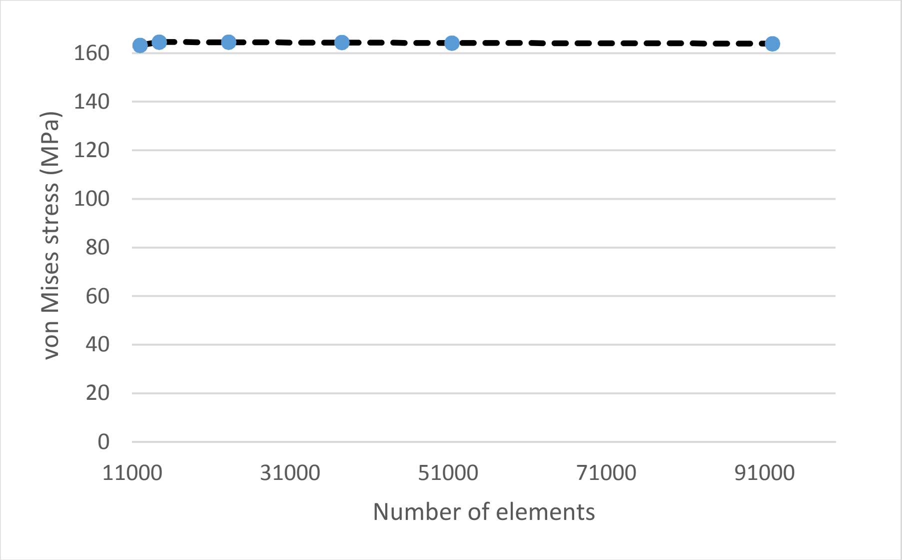
Number of elements	Maximum deflection (mm)	von Mises stress (MPa)	von Mises stress variation
12028	14.698	163.22	-
14436	14.698	164.52	-0.79%
23241	14.699	164.46	0.03%
37530	14.700	164.34	0.08%
51476	14.700	164.12	0.13%
91999	14.700	163.88	0.15%

Source: Authors

As shown on Table 3, the deflection and von Mises stress values are changing in a very small rate, however, one mesh must be chosen. Considering the values of von Mises stress variation, *i.e*. mesh convergence, the most interesting mesh on this case is the mesh with 37530 elements, each one having a size of 1.5 mm. The previous mesh would also be good, however, in order to be conservative, it is not a good decision to take the first mesh after a huge change on the slope of the curve, as can be seen in Figure 8.

As for SHELL93 elements, the total of simulations run for the SOLID95 elements is also 6. Table 4 presents the results for the SOLID95 element, while Figure 9 shows the mesh convergence study. The mesh with 213903 elements was chosen due the same reasons as for SHELL93 element. Each element of this mesh also has a size of 1.5 mm.

Figure 8 - Mesh convergence – SHELL93 – Tubular Frame



Source: Authors

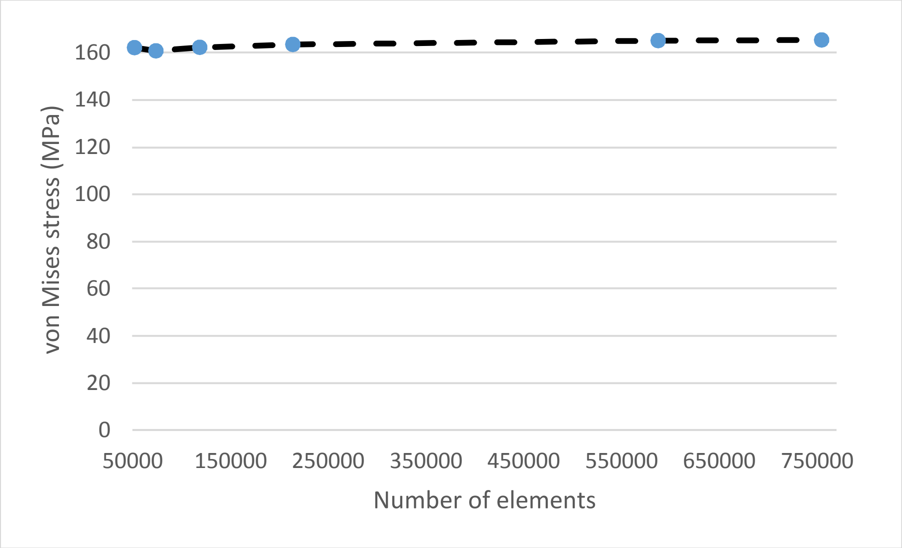
It is noteworthy to mention that these solid elements were tetrahedra instead of hexahedra. As mentioned by Wang, Nelson and Rauch (2004), hexahedral elements generally require extra shape functions to work properly on bending dominated problems, and as this design has a large amount of elements, extra shape functions would increase the computational cost.

Table 4 - Results for SOLID95 – Tubular Frame

Number of elements	Maximum deflection (mm)	von Mises stress (MPa)	von Mises stress variation
52048	14.683	162.02	-
73881	14.686	160.66	0.84%
118852	14.687	162.15	-0.93%
213903	14.689	163.37	-0.75%
587878	14.694	164.96	-0.97%
754958	14.695	165.22	-0.16%

Source: Authors

Figure 9 - Mesh convergence – SOLID95 – Tubular Frame



Source: Authors

The analytical solution for deflection can be found using the Virtual Work Method (VWM) considering this frame as two different straight parts. For further information on VWM, one can consult Kassimali (2011). By using this method, a value of 13.68mm in the y-axis direction is found.

Additionally, the analytical solution for the bending stress on the curve can be found by (PILKEY, 2004):

$$\sigma_F = \frac{M(r_n - y)}{Aey} \tag{4}$$

where: r_n is the dislocated neutral axis at the curve, y is the distance from the neutral axis, A is the area of the section, and e is the distance between the curvature radius and r_n.

The result found using Equation (4) must be updated using the procedures given in Pilkey and Pilkey (2008). Therefrom, the analytical result for bending stress is 177.73MPa. Since the dominating stress on this structure is due to bending, it turns out that the bending stress is also approximately the von Mises stress.

Figure 10 shows the comparison for von Mises stress between SHELL93 and SOLID95 discretization. The image is zoomed in, so that the details at the curve of the tube can be seen.

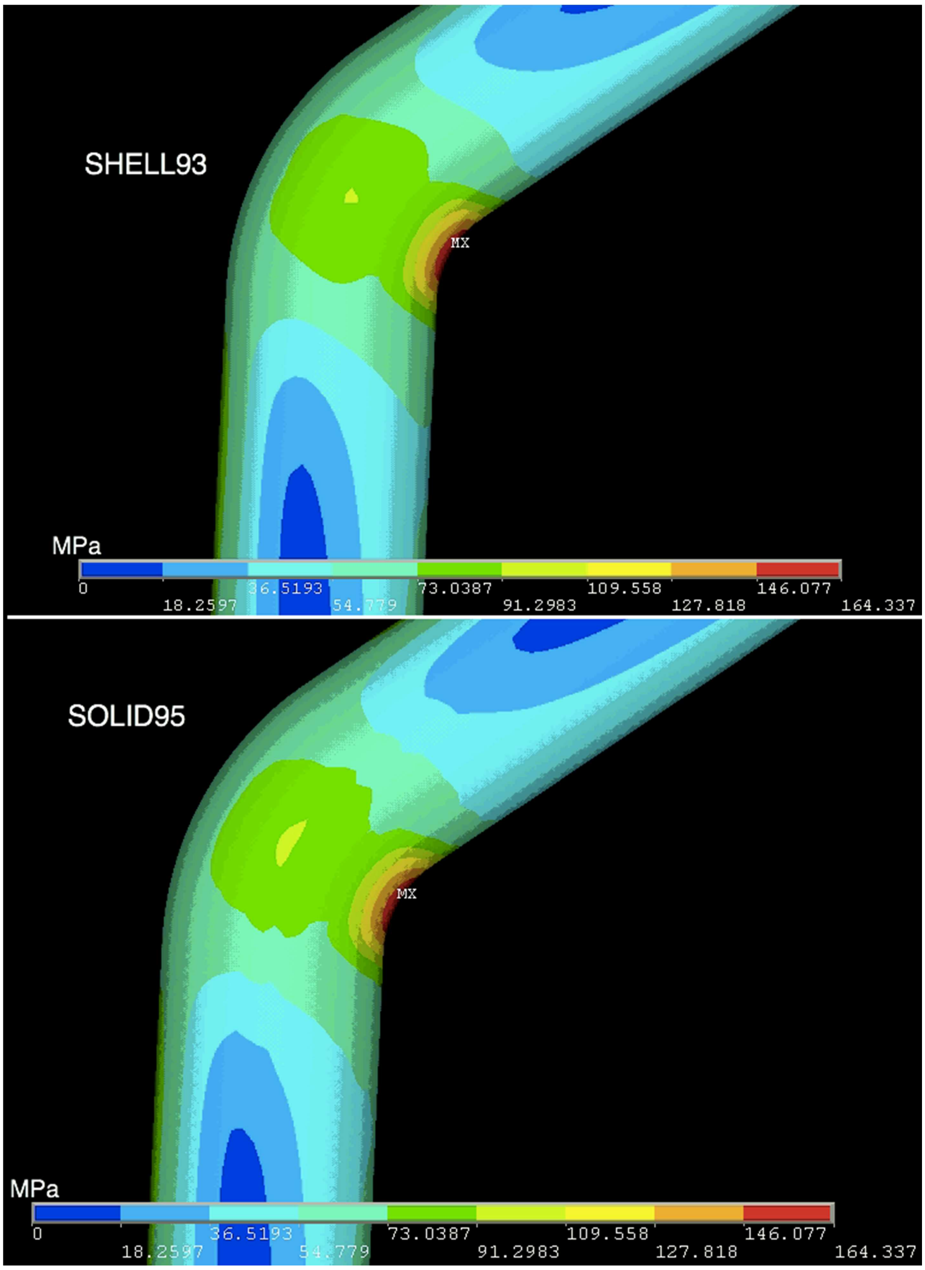


Figure 10 - von Mises stress – SHELL93 VS SOLID95 – Tubular Frame

Source: Authors

Again, there is a very small difference between the results for SHELL93 and SOLID95 elements. The color's profile on Figure 10 confirms this statement.

3.3 C-Shaped Roof Truss

Due to the facts that the stresses on a real truss are mostly normal stresses, and the stress distribution using both SHELL93 and SOLID95 elements on each member of the truss are not exactly the same along the whole member, the principal way to test the mesh convergence for this structure is then by its maximum deflection. The first element to be tested was 25 mm size, and the shape was quadrilaterals for every of the following SHELL93 simulations.

The results for SHELL93 element are presented on Table5.

Table 5 - Results for SHELL93 – C-Shaped Roof Truss

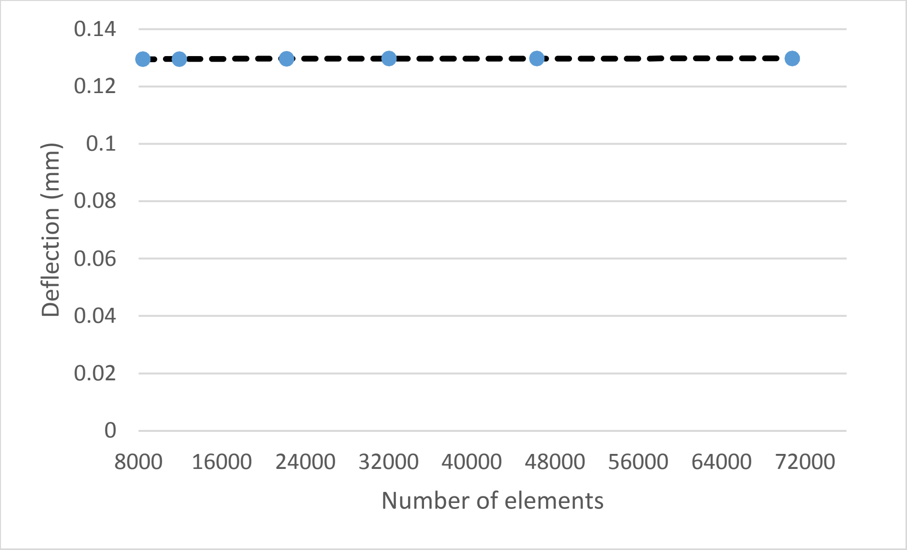
Number of elements	Maximum deflection (mm)	Deflection variation
8400	0.12950	-
11913	0.12957	-0.052%
22209	0.12966	-0.069%
32052	0.12974	-0.062%
46238	0.12975	-0.006%
70797	0.12977	-0.017%

Source: Authors

As shown on Table 5, the meshes returned very similar values. However, the smallest change was between the fourth and fifth elements. Therefore, the mesh with 32052 elements will be chosen. This mesh has element size of 12 mm in dimension. It is important to point out that all of the deflection values given on this section are absolute values.

The mesh convergence curve is shown on Figure 11.

Figure 11 - Mesh convergence – SHELL93 – C-Shaped Roof Truss



Source: Authors

After simulations with SHELL93, simulations using SOLID95 were performed. These simulations were done using tetrahedral elements, and the first element was also 25 mm size.

The results for the SOLID95 element are given in Table 6.

Table 6 - Results for SOLID95 – C-Shaped Roof Truss

Number of elements	Maximum deflection (mm)	Deflection variation
46201	0.11940	-
84034	0.11947	-0.061%
128426	0.11952	-0.029%
208621	0.11952	-0.014%
308710	0.11954	-0.010%
472954	0.11954	0.000%

Source: Authors

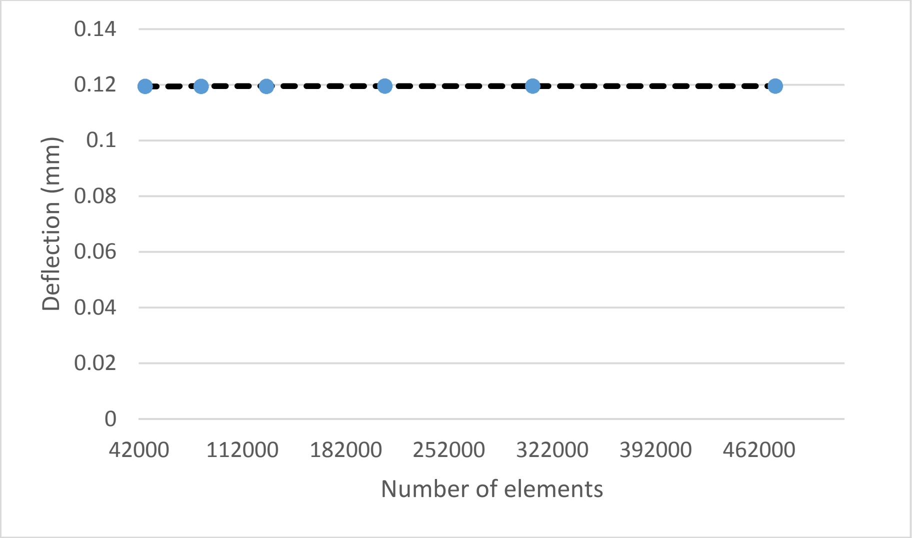
The results presented on Table 6 show that the values are increasing slightly, as the values from SHELL93 simulations. On the SHELL 93 simulations, it was chosen the value with the smallest change between the results. However, the SOLID95 has a greater computational cost than SHELL93, which makes the decision of taking the value with the smallest change a bad decision.

As the values are only increasing, and they are increasing in a very small rate, it is a good choice to take the second result, for the changes after this element size are really small.

Also, it is important to mention that the difference between the first and second values is small if it is considered the size of the structure. However, as bigger elements were not simulated, there are no results to verify the stability of the previous results.

The variation of the obtained results can be seen on the mesh convergence curve presented on Figure 12.

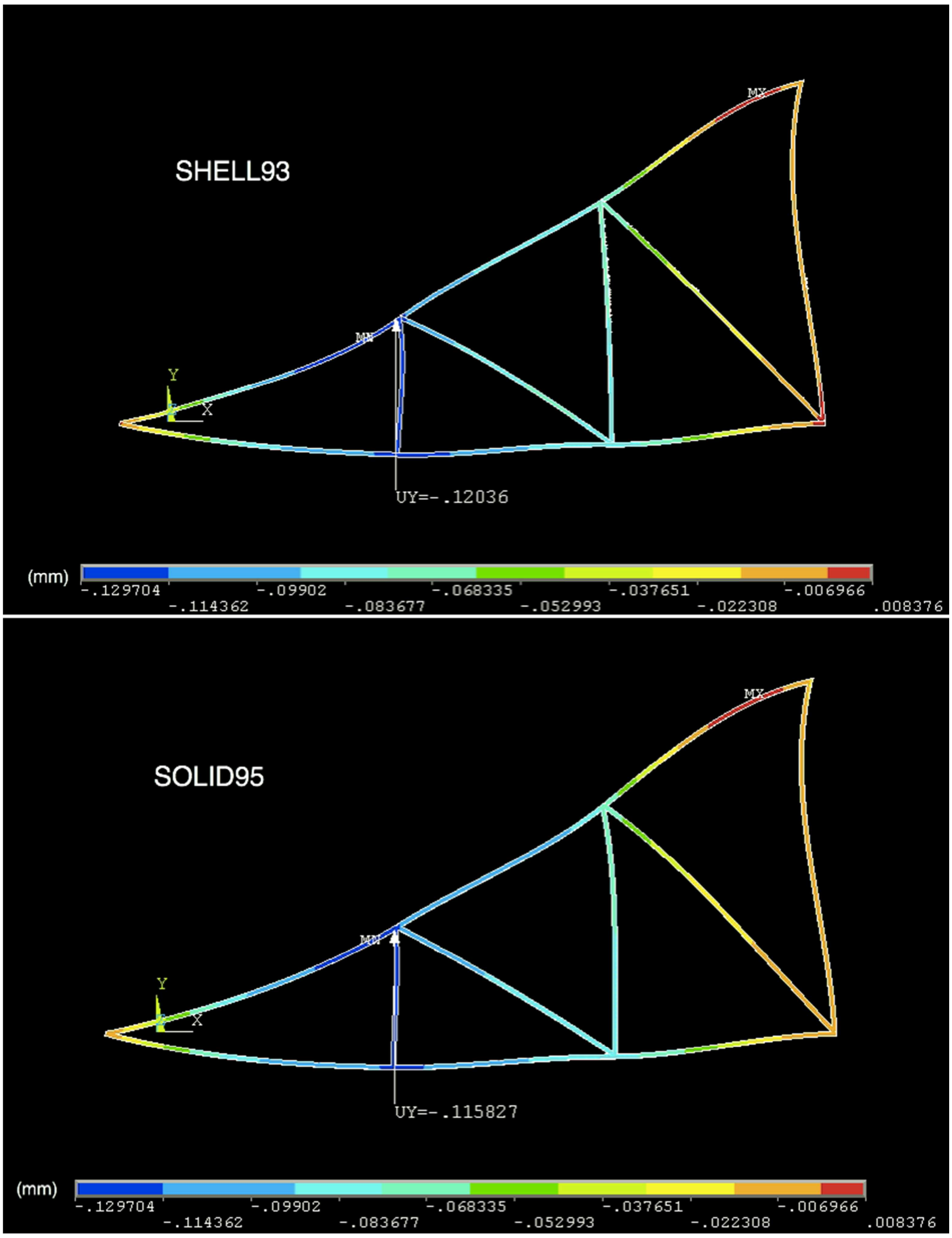
Figure 12 - Mesh convergence – SOLID95 – C-Shaped Roof Truss



Source: Authors

Figure 13 shows the comparison between the displacements in *y*-axis direction given for SHELL93 and SOLID95 models.

Figure 13 - Deflections – SHELL93 VS SOLID95 – C-Shaped Roof Truss



Source: Authors

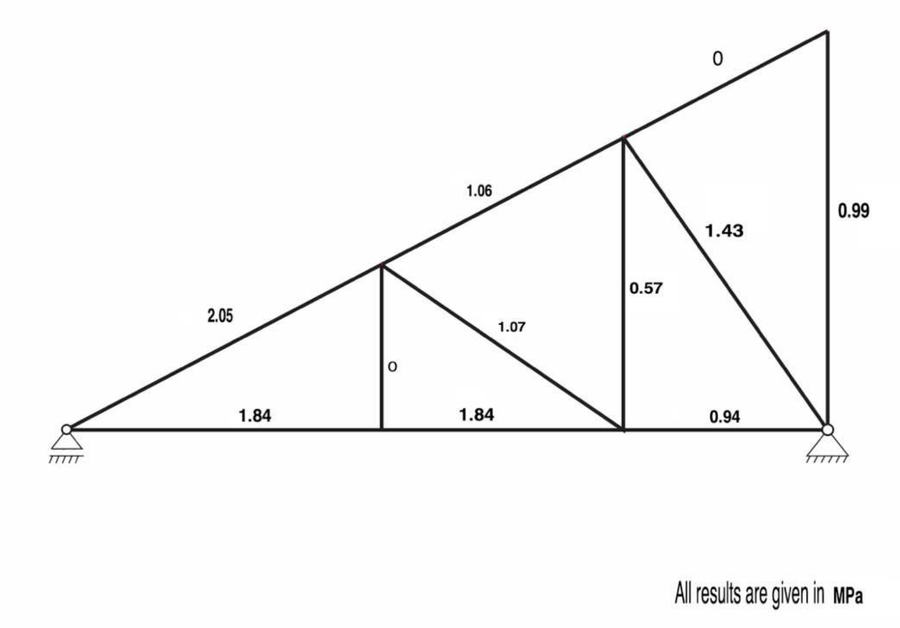
Figure 13 shows that the values for displacement for the two different finite element are very close. The places where the maximum and minimum displacement of each finite element occurred are pretty close to each other.

For comparison, the analytical deflection value for the point shown on Figure 13 was calculated using the Virtual Work Method (HIBBELER, 2010). The analytical displacement value in the *y*-axis direction for the point marked on

Figure 13 is 0.1205 mm downwards. This value is very close to both values found by FEA.

Additionally, Figure 14 presents the analytical values for normal stress. It can be obtained after finding the forces on each member by using the Nodal Method (HIBBELER, 2009).

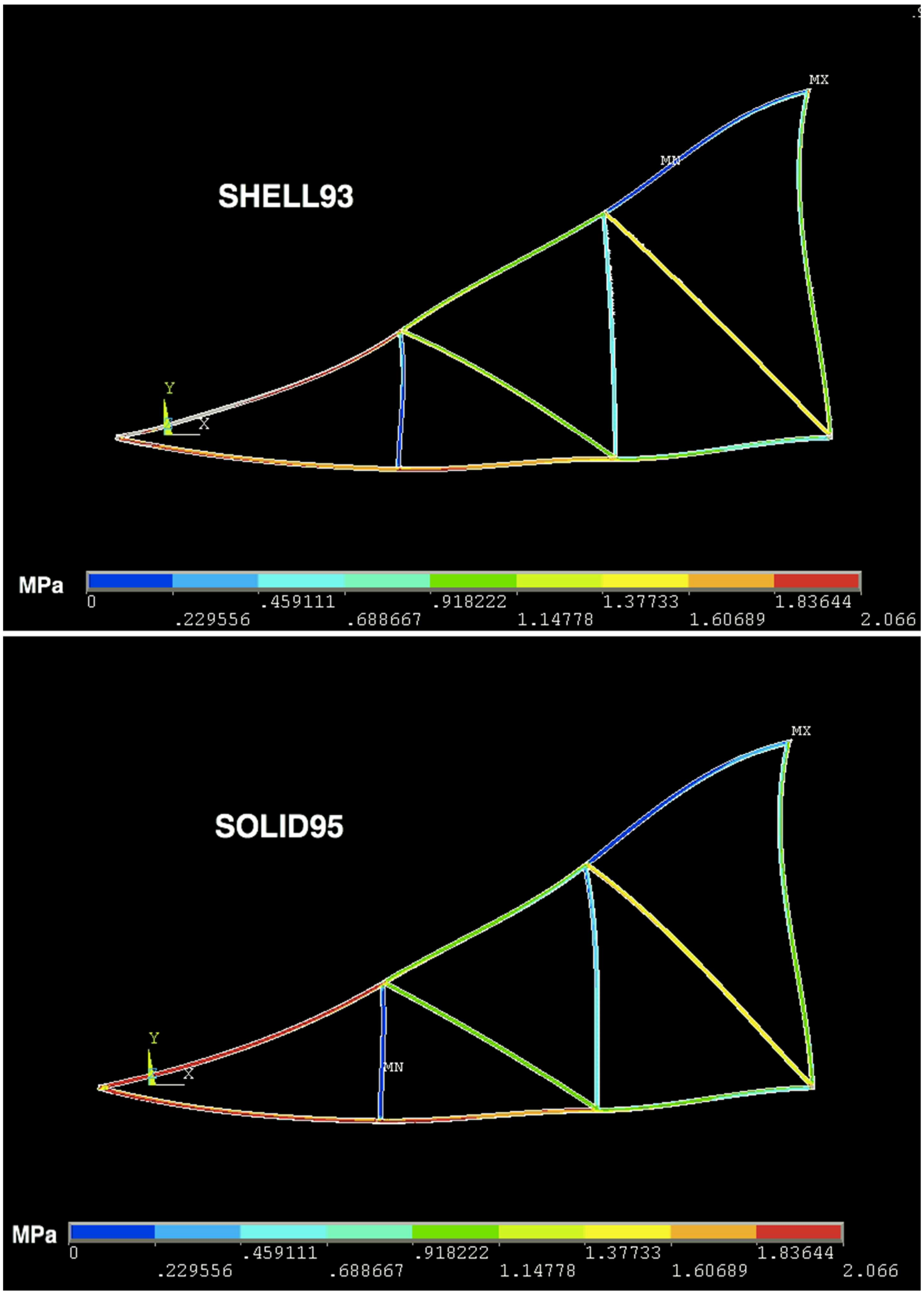
Figure 14 – Normal stresses on C-Shaped Roof Truss' members



Source: Authors

The results presented on Figure 14 can be compared with the values given on Figure 15, even though values for von Mises stress were not taken in consideration when the mesh refinement was made.

Figure 15 – von Mises stress – C-Shaped Roof Truss



Source: Authors

It is shown on Figure 15, that the values for von Mises stress obtained from FEA are very close to those obtained analytically. It is important to clarify, though, that the analytical solution is considering no moment at the truss' nodes, so that there are only normal stresses present at each member. The results are very similar to those from FEA due to the tiny intensity of the nodes' moments.

4 CONCLUSIONS

Results shown on this paper strongly suggest that SHELL elements can be used in a variety of problems where BAR or BEAM elements are commonly employed, for instance in tubular structures. This observation comes from the differences between all of the values for displacements given from SHELL93 and SOLID95 simulations. For the selected elements size after the mesh refinement, the differences are no bigger than 0.011 mm on the Tubular Frame, a structure that is about 1000 mm long. Also, differences between the analytical results and the FEA results are very close for all of the structures, if it is excluded the values caused by singularities (BARLOW, 1982).

On the cantilever beam the differences between the von Mises stresses for the SHELL93 and SOLID95 are only on the region where singularities occur. This statement can be confirmed by taking a look at the Figure 7, which shows the map of the von Mises stresses for both SHELL93 and SOLID95 finite elements

The von Mises stress on the Tubular Frame must be treated specifically, since it involves a stress concentration area due to the curve on the structure. On the analytical solution the value for the stress concentration is obtained from a table in the work of Pilkey & Pilkey (2008). The curves on that table might have been fitted accordingly a scattered data. On the other hand, the stress concentration on FEM is obtained in the same way that the rest of the values for the whole computational domain. The difference between analytical and FEM solutions are around 8%.

The results for stresses for the C-Shaped Roof Truss on Figure 15 are within the acceptable range, since the analytical solutions take in consideration no friction at the joints. Also, it is important to point out that the analytical solutions do not consider the stress concentration factor at the joints, whereas the analyses performed on this paper do.

It is suggested as a future work a study using SHELL93 in reticulated structures with different types of holes on its members to verify the SHELL93 behaviors on an optimization process using FEM.

REFERENCES

ANSYS RELEASE. **ANSYS, Inc. Theory Reference**. 9th. ed. Canonsburg: SAS IP, v. 1, 2004.

BARLOW, J. Constraint Relationships In Linear And Nonlinear Finite Element Analyses. **International Journal for Numerical Methods in Engineering**, v. 18, p. 521-533, 1982.

BATHE, K.-J. **Finite Element Procedures**. 2nd. ed. Watertown: Klaus-Jürgen Bathe, 2014.

COVILL, D. et al. Parametric finite element analysis of bicycle frame geometries. **Procedia Engineering**, v. 72, p. 441-446, 2014.

HIBBELER, R. C. **Engineering Mechanics:** Statics. 12ª. ed. Upper Saddle River: Pearson, 2009.

HIBBELER, R. C. **Resistência dos materiais**. 7ª. ed. São Paulo: Pearson Prentice Hall, 2010.

KAEWKULCHAI, G.; WILLIAMSON, E. B. Beam element formulation and solution procedure for dynamic progressive collapse analysis. **Computer & Structure**, v. 82, n. 7-8, p. 639-651, 2004.

KASSIMALI, A. **Structural Analisys**. 4th. ed. Stamford: Cengage Learning, 2011.

KHAN, I. A.; PARHI, D. R. Finite Element Analysis of Double Cracked Beam and its Experimental Validation. **Procedia Engineering**, v. 51, 2013.

KIM, J.-W.; DOH, J.-H.; LEE, Y.-H. Behavior Characteristic of Hypar Shaped Space Truss with Some Removed Members. **Strucutures Congress: Crossing Borders.** Vancouver, Canada: ASCE, April 2008**.**

KIM, J.; YOON, J.-C.; KANG, B.-S. Finite element analysis and modeling of structure with bolted joints. **Applied Mathematical Modelling**, v. 31, n. 5, p. 895-911, May 2007.

LOGAN, D. L. **A First Course in the Finite Element Method**. 4ª. ed. Toronto: Thomson, 2007.

MACKERLE, J. Finite element analysis and simulation of machining: an addendum: A bibliography (1996-2002). **International Journal of Machine Tools and Manufacture**, v. 43, n. 1, p. 103-114, January 2003.

PFEIL, W.; PFEIL, M. **Estruturas de Aço:** Dimensionamento Prático. 8th. ed. Rio de Janeiro: LTC, 2009.

PILKEY, W. D. **Formulas for Stress, Strain and Structural Matrices**. 2nd. ed. Hoboken: John Wiley & Sons, 2004.

PILKEY, W. D.; PILKEY, D. F. **Peterson's Stress Concentration Factors**. 3rd. ed. Hoboken: John Wiley & Sons, 2008.

SADOWSKI, A. J.; ROTTER, J. M. Solid or shell finite elements to model thick cylindrical tubes and shells under global bending. **International Journal of Mechanical Sciences**, v. 74, p. 143-153, September 2013.

WANG, E.; NELSON, T.; RAUCH, R. **Back to Elements: Tetrahedra vs. Hexaedra**. Online source. Available on: <https://support.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/confpaper/2004-Int-ANSYS-Conf-9.PDF>. Acessed on Obtober 21st, 2017.

XUE, S. -D.; LIU, Y.; LI, X. -Y. Dynamic Performance Analysis of Single-Layer Cylindrical Reticulated Shell Considering Pile-Soil-Structure Interaction. **Journal of the International Association for Shell and Spatial Structures** , v. 56, n. 2, p. 91-100, June 2015. ISSN 1996-9015.

ZHENG, L. Finite Element Analysis of a Single-Layer Reticulated Dome and the suspendome under Impact Loading. **Advanced Materials Research**, v. 594, p. 844-848, 2012. ISSN 1662-8985.

Enviado em: 28 nov. 2017
Aceito em: 29 nov. 2017

Editor responsável: Mateus das Neves Gomes