Alternative approach to the finite element mesh convergence testing in simulation modelling of maritime structures mechanical parts

Josip Matas, Nenad Vulić

Abstract- In modern engineering world the analysis of the strength of modern maritime structures almost always relies upon the methods of finite elements (FEM). These structures are usually modelled by means of powerful 3D-modelling applications, such as Autodesk Inventor, Pro/Engineer, SolidWorks or similar, containing simulation modelling FEM tools for the analysis of linear behaviour of mechanical static deformable bodies as their intrinsic part. However, the quality of the obtained results strongly depends upon the type and size of the finite elements used. The FEM tools in these applications often recommend the size of the elements forming the actual mesh, so modern design engineers usually take these recommendations for granted, thinking of them as the unique solution to their problem.

Unfortunately, this is not true. Static analysis of solid deformable bodies with their material behaving linearly (following Hooke's law) may be described by systems of partial differential equations formulated in the theory of elasticity, incorporating functions such as displacements, strains and stresses. These equations may be solved analytically only in the cases of simple structural shapes and special boundary conditions, far away from real problems. So, numerical solutions of such problems based upon FEM are only the approximation, so testing of convergence of the preselected FE mesh is required.

This paper presents another proposal for the judgement of the FEM results quality, additional to the testing of mesh convergence by reducing the element size and increasing number of nodes and/or the changing the degree of interpolation polynomials within the elements. The basic idea is to compare the FEM numerical 3D simulation modelling results with the ones obtained by 1D analytical models (classical beam theory) and 2D analytical models (theory of elasticity) for a simple case where the analytical solutions can be formulated in closed form formulae. The typical example of such a system is the linearly elastic cantilever beam of finite height. The 3D results obtained by the FEM tool in SolidWorks on the 3D model of this cantilever obtained for different mesh sizes and element types have been compared to the 2D and 1D analytical solutions with the aim to judge whether the element size recommended by the SolidWorks actually produces the best numerical solutions in terms of stresses and displacements.

The actual showcase taken to implement this alternative way of testing mesh convergence is the part of small maritime offshore structure used for lifting of civil engineering objects. The FEM results for the element size recommended by SolidWorks and for the element size obtained by the previously described procedure of comparison with the analytically obtainable 2D and 1D results show that there exists the difference in terms of stresses numerically calculated by these two meshes, which may mislead the designers of the structure in their final conclusions and decisions about the acceptability of this structural part.

Keywords: FEM, mechanics of materials, theory of elasticity, analytical solutions, 1D/2D/3D models, offshore structural parts

1 Introduction

There is no more doubt that the best practice of today for the design of modern engineering structures, machinery and their parts is based upon three-dimensional (3D) design computer applications such as Autodesk Inventor, Pro/Engineer, SolidWorks, etc. rather than implementing so-called drawing board tools, such as AutoCad or similar. These 3D applications practically offer the designers parametric modelling of the structural parts based upon simple sketches, enabling them to amend the structural part and its interconnected parts in an easy way [1]. One of the most important features of 3D modelling applications is that they always contain the finite element modelling tools (FEM) within themselves. These tools are used to determine the mechanical response of the structural parts (displacements, strains, stresses and safety against static failure) in an easy manner, originating from in the 3D application already defined structural shape, preselected dimensions, linear elastic material properties, static external loading and boundary conditions.

In the past, design offices often had to employ technical specialists dedicated exclusively to the FEM analyses, using the special tools and needing extra skills to ease them. This has changed completely. Engineers in modern design offices may be rather relaxed regarding the implementation of these FEM analyses within their powerful 3D applications, because of the ease of their use and obtaining the FEM output results by a simple invoking the simulation modelling tool within their 3D applications. These analyses may also go beyond the usual mechanical response linear static structural analysis. However, exactly due to the mentioned relaxed modelling situations, engineers shall always bear in mind that the finite element solution is not more than the numerical model for the behaviour of the elastic structural part, highly dependent upon boundary conditions, type of the finite elements, their size and the level of interpolation within them.

Consequentially, engineers are taught that they have to check out the quality of their FEM results, based upon the convergence testing of the FE mesh by e.g. reducing the size of the preselected elements type. This size reduction is expected to produce results of a better precision. On the other hand, reduction of element size in the model of the same structural part also means increasing numbers of nodes, as well as increasing the number of numerical equations to be solved. In addition to this, it is well-known that the FE solutions can only approach the actual ones in asymptotic terms, meaning that they are always above or below the actual ones regardless of how fine the FE mesh may be. The real behaviour of the actual structural part under mechanical loading does not depend upon the models, because it is only the consequence of the laws of physics, regardless of the fact whether we are able to describe them properly or not.

The aim of the paper is to warn the engineers about necessity of rethinking about the quality of their results obtained by FEM analyses within their 3D modelling applications. This quality evaluation shall never be based upon the using the finite element size recommended by the program itself. The designer shall go above and below this element size to judge out what is happening with the numerically calculated displacements and stresses, or to think out and implement another approach with respect to this.

The actual goal of the paper is to present and propose the alternative approach additional and complementary to the common test of mesh convergence based upon element size. This alternative approach may be based upon comparison of the FEM results obtained by 3D modelling with the ones obtained by analytically solvable 2D theory of elasticity models or even 1D classic strength of materials models. Basically, in order to judge the quality of FEM results obtained by the element size recommended by the 3D design application, the user takes the simple problem with the known analytical 2D or 1D solutions, such as cantilever beam of finite length, breadth and height, made of linear elastic homogenous isotropic material, with the uniform load on its upper surface. After that, the user produces 3D numerical results for different FEM mesh sizes and compares these results for stresses and displacements in the sections of 3D models with the analytical ones in 2D or 1D model. This procedure may lead to the different mesh size producing the best results in comparison with the size recommended by 3D modelling tool or the one obtained by classical convergence testing.

The material section describes and presents the original cantilever beam problem to be solved analytically as the simplest 1D and the better suitable 2D analytical model. Both of these analytical models will serve as the reference for the later 3D numerical models.

The methods section presents the analytical equations for 1D model and 2D model, as well as the finite elements types and their interpolation functions proposed to be used within the 3D numerical models.

The results section presents the two cases. The first one are the simple cantilever beam models with the tables of numerical values of their analytical (1D and 2D) and numerical (3D) solutions obtained in preselected cross sections far enough from the clamped end, so that the Saint Venant's principle applies. The 3D results have been compared with the 1D and 2D ones. The second case is the actual model of the maritime offshore structure part to which the analysis implementing 3D design computer application such as SolidWorks (and its recommended FE mesh size) has been applied, together the results brought out by 25%, 50%, 150% and 200% of the original recommended element size. The referent value to compare these 3D results is the one obtained by the previous cantilever beam model as the reference, so all other 3D results are compared with the thus obtained referent one.

The discussion section comments the compared results of 3D model against the 2D and 1D for the cantilever beam model with the known analytical solution. It also compares the actual results for the maritime offshore structure part for the mesh element sizes as described before with the ones obtained by the recommended element size and comments the obtained deviations.

The conclusion briefly reflects the aim and goal of the paper itself. It also gives a brief overview of the two analyses cases: cantilever beam as the fundamental case and the maritime offshore structure part as a showcase. The usability applicability of this alternative approach is briefly commented and the procedure to be used has been shortly reviewed. The engineers in design offices have been warned that they may not rely solely upon the FE element and mesh sizes recommended to them by the 3D design application. They are informed not only about the necessity to use mesh convergence tests and/or the here presented alternative approach by comparison with the analytical solutions, but also the validation measurements once the structure has been manufactured, assembled and subject to testing.

- *Author Josip Matas is currently pursuing master's degree program in marine engineering at University of Split, Faculty of Maritime studies. E-mail: jmatas1@pfst.com*
- *Co-Author dr.sc. Nenad Vulić is Full Tenured Professor / Scientific Advisor Faculty of Maritime Studies / University of Split R. Boskovica 37, HR-21000 SPLIT, Croatia Full member of the Scientific Council for Maritime Affairs, of the Croatian Academy of Sciences and Arts, ZAGREB. E-mail: nenad.vulic@pfst.hr*

2 MATERIAL

Cantilever beam was used as a referent example. Regarding boundary conditions it was subjected to the continuous load on the top surface with one fixed and one free end. The dimensions of the cantilever beam are presented in the Table 1.

Tab. 1. Cantilever beam dimensions

Length [mm]	Width [mm]	Height [mm]
2150	20	240

For the purpose of simulation analysis on a computer in the program SolidWorks, an example material selected was AISI 304 for the purpose of calculating stresses and displacements.
Uniform surface loading used on the cantilever beam had the value of 0.8 [N/mm2] as described in Figure 1.

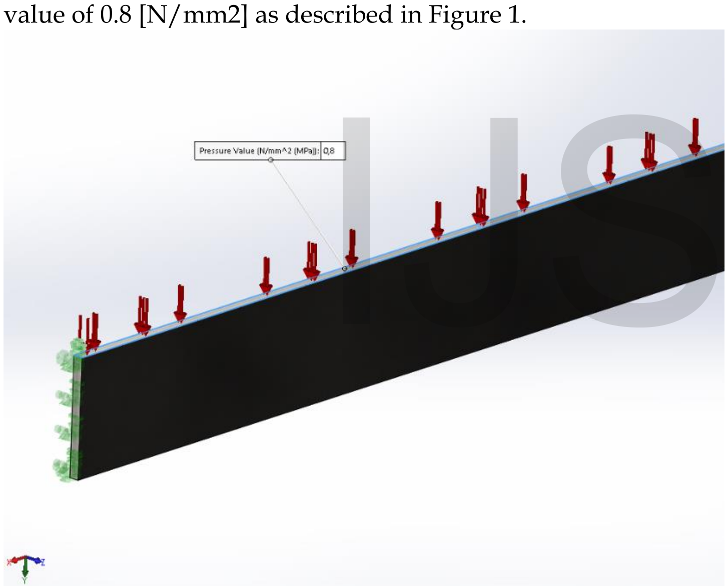

Fig. 1. Continuous load of cantilever beam

After the analysis of the cantilever beam, the task was to analyse the small offshore structure component, as shown in Figure 2.

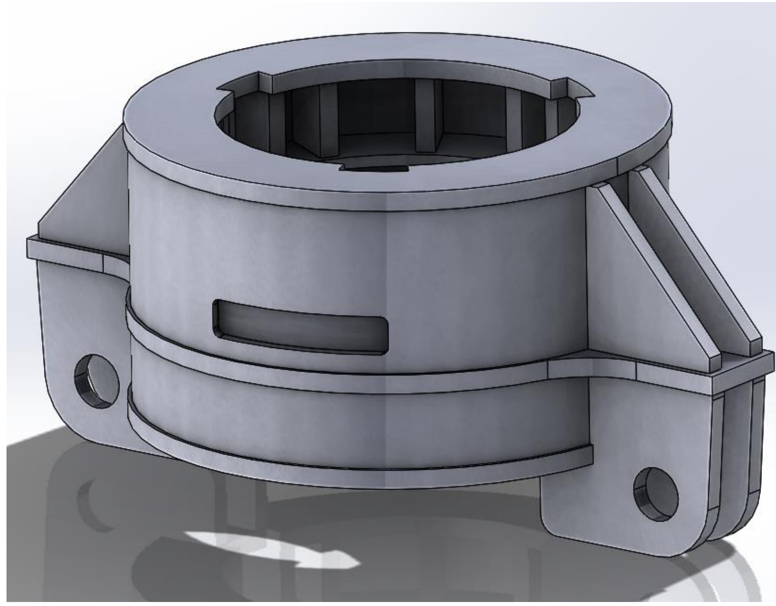

Fig. 2. Small offshore structure component to be analysed

A small offshore structure has been subjected to the load of two hydraulic cylinders that hold on each pair of eye shaped circular inserts using a pin. Dimensions, loading, and material properties as well as the working principle will be omitted due to the confidentiality agreement with the customer.

3 METHODS

The methods to be used in 1D and 2D models of the cantilever beam are the analytical solutions based upon the strength of materials (1D) and theory of elasticity (2D). Key spots for calculating values of stress and displacements are shown in Figure 3.

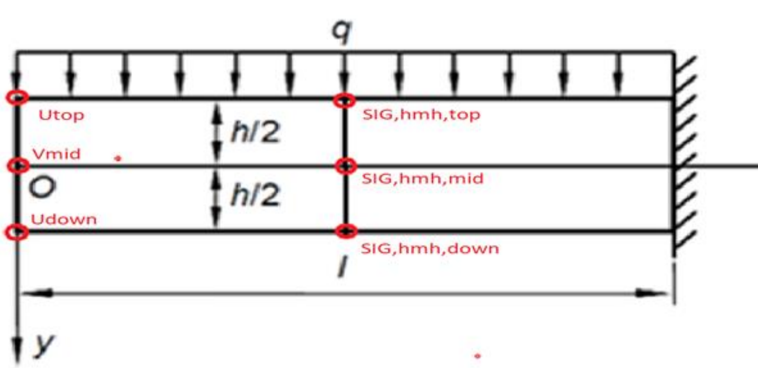

Fig. 3. Key spots for analysis of cantilever beam

3.1 Analytical solutions of 1D cantilever beam model

The analytical solutions in 1D case are well-known and described in [2-3] and verified implementing MD Solids program as described in [3-4].

$$Q_z = \frac{dM_y}{d_x} \quad (1)$$

$$M_y = -\frac{1}{2}\, q\, x^2$$

$$\beta = \frac{q\, L^3}{6\, E\, I_y} + \left[1 - \left(\frac{x}{L}\right)^3\right]$$

$$\omega = \frac{q\, L^4}{24\, E\, I_y} \left[\left(\frac{x}{L}\right)^4 - 4\, \left(\frac{x}{L}\right) + 3\right]$$

$$\sigma_x = \frac{M_y}{I_y}\, z = \frac{qx^2}{2\, I_y}\, z$$

$$\sigma_y = \frac{q}{2\, b} \left(\frac{2\, z}{H} - 1\right)$$

$$\tau_{x,y} = \frac{q\, x}{2\, I_y} \left(\frac{H^2}{4} - z^2\right)$$

Where:

L –Length of the beam;
b - Section width of the beam;
h - Section height of the beam;
I- Moment of inertia of cross section;
E - Young's module;
q – Uniform loading;
Q – Shear force;
M – Bending moment;
β – Slope;
w – Displacement;
σx –Horizontal component of the normal stress;
σy –Vertical component of the normal stress;
τx,y –Tangential stress.

Furthermore, the graphical representation of the displacement and stress values for the 1D model is presented, using MS Excel.

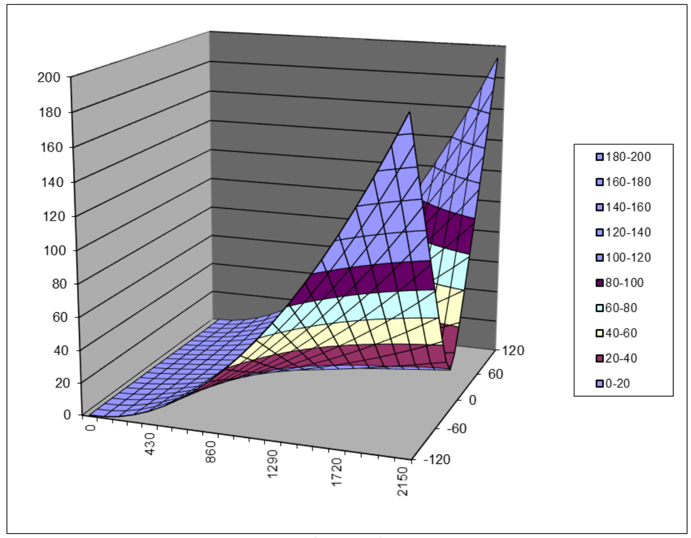

Fig. 4. HMH Stress for 1D model

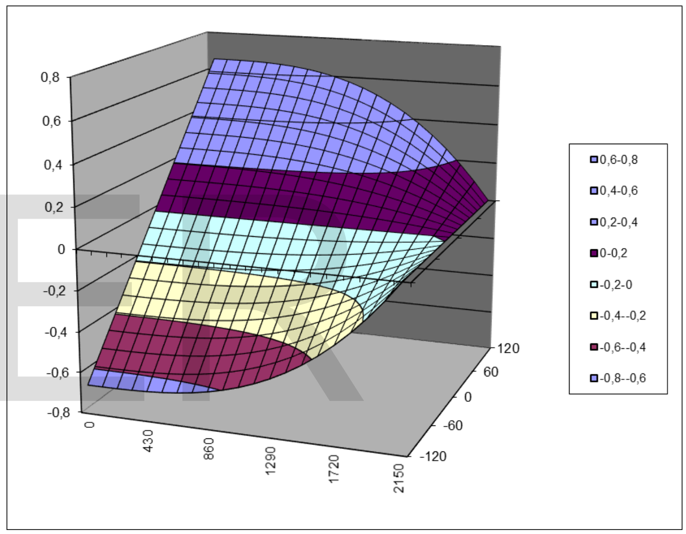

Fig. 5. Horizontal displacements for 1D model

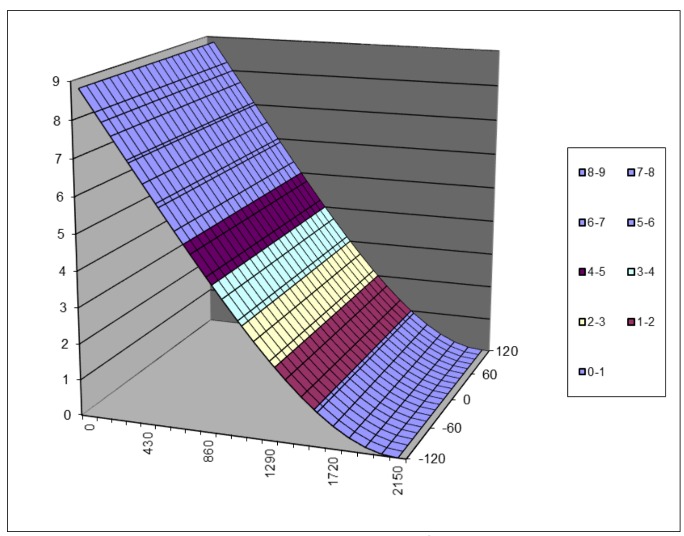

Fig. 6. Vertical displacements for 1D model

3.2 Analytical solutions of 2D cantilever beam model

These solutions have been developed based upon the theory described in [5], with the formulae and details described in [6].

$$I = \frac{b\,h^3}{12} \qquad (3)$$

$$\beta = \frac{8+9\nu}{2+\nu}$$

$$A_4 = -\frac{1}{120\,l}\,qh^2 - \frac{1}{6l}$$

$$A_5 = 0$$

$$u_0 = -\frac{q}{2\,E\,I}\;\nu\;\frac{L\,h^3}{12}$$

$$v_0 = -\omega\,L - \frac{q}{2\,E\,I}\left(\frac{\nu}{B}\,L^2 h^2\right)$$

$$\omega = \frac{q}{2\,E\,I}\left[\frac{L^3}{3} + \left(\frac{1}{10}+\frac{\nu}{4}\right)L\,h^2\right]$$

$$A_1 = A_2 = 0\;;\; A_3 = -\frac{q}{4}$$

$$\sigma_x = -\frac{1}{I}\left(\frac{1}{2}\,q\,x^2\right)y + \frac{1}{3\,I}\,q\,y^3 - \frac{1}{2}\,A_1\,y^2 + 6\,A_4\,y + 2\,A_5$$

$$\sigma_y = \frac{q}{2\,I}\left(\frac{1}{4}\,y\,h^2 - \frac{1}{3}\,y^3\right) + \frac{1}{2}\,A_1\,x^2 + 6\,A_2\,x + 2\,A_3$$

$$\tau_{xy} = -\frac{1}{2\,I}(q\,x)\left(\frac{h^2}{4} - y^2\right)$$

$$\sigma_1 = \frac{(\sigma_x+\sigma_y)}{2} + \sqrt[2]{\left(\frac{\sigma_x-\sigma_y}{2}\right)^2 + \tau_{xy}^2}$$

$$\sigma_2 = \frac{(\sigma_x+\sigma_y)}{2} - \sqrt[2]{\left(\frac{\sigma_x-\sigma_y}{2}\right)^2 + \tau_{xy}^2}$$

$$\sigma_{HMH} = \sqrt[2]{\sigma_1^2 + \sigma_2^2 - \sigma_1\,\sigma_2}$$

Where:

b - Section width
h - Section height
I - Moment of inertia of cross section
E - Young's module
v –Poisson coefficient
q - Continuous load
A - Integration constant
u0, v0 - Integration constants
σx - Longitudinal stress component
σy - Transverse stress component
σ1 - Normal stress
σ2 - Tangential stress
σHMH - Equivalent stress

Furthermore, the graphical representation of the displacement and stress values for the 2D model is presented, using MS Excel and VBA (Visual Basic for Applications).

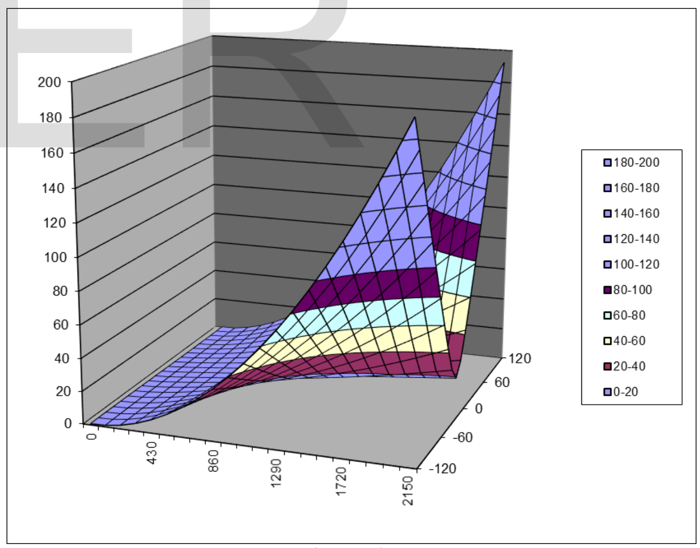

Fig. 7. HMH Stress for 2D model

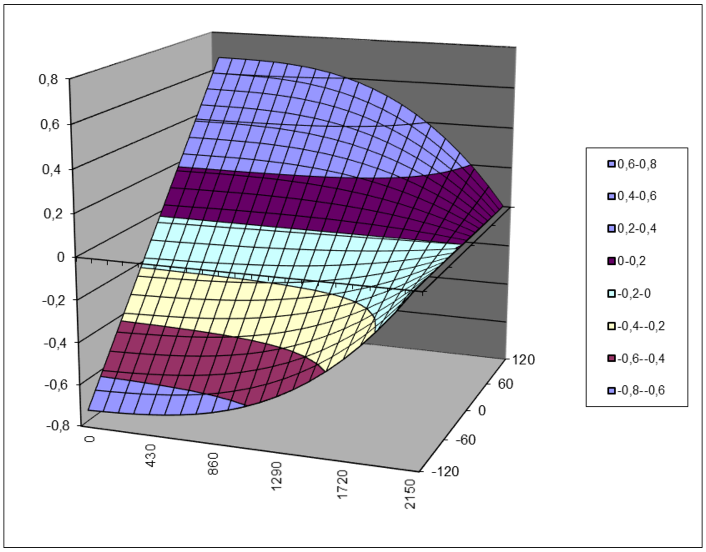

Fig. 8. Horizontal displacements for 2D model

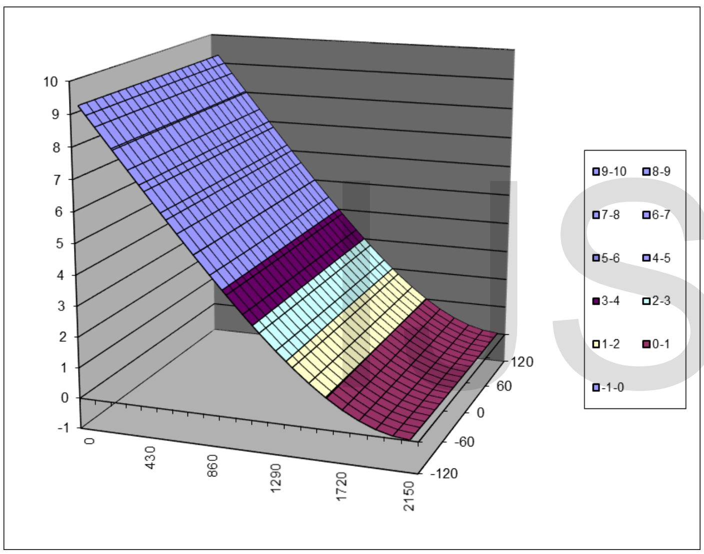

Fig. 9. Vertical displacements for 2D model

3.3 Numerical solutions for the 3D cantilever beam model in Solid Works

Numerical approach for solving 3D cantilever beam consisted of choosing boundary conditions, loading, material properties, type and size of the elements. This analysis used 3D tetrahedral elements with different mesh size. Recommended size of the mesh by the software was 21.77 [mm] with the properties in the Table 2. This paper actual purpose was to verify this recommendation.

Tab. 2. Mesh details for the recommended mesh size

Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points for High quality mesh	16 points
Element size	21,7751 [mm]
Tolerance	1,08875 [mm]
Mesh quality	High
Total nodes	20014
Total elements	11297
Maximum Aspect Ratio	5,6609
Percentage of elements with Aspect Ratio < 3	97,9
Percentage of elements with Aspect Ratio > 10	0
% of distorted elements (Jacobian)	0
Number of distorted elements	0
Time to complete mesh (hh:mm:ss)	00:00:02

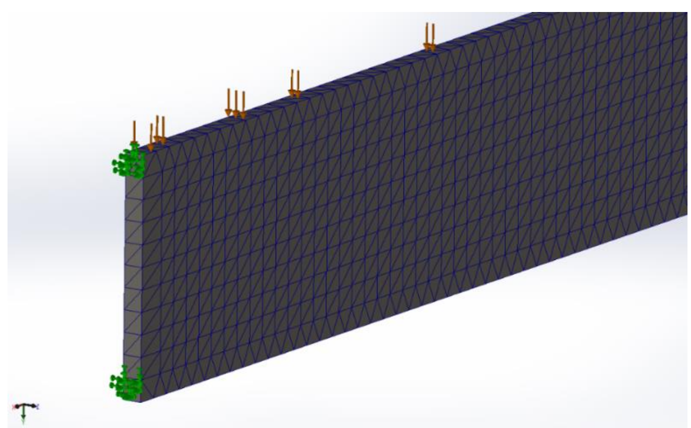

Fig. 10. Finite element mesh of the cantilever beam model in SolidWorks

4 RESULTS

4.1 Cantilever beam

Based on the previously mentioned dimensions, shapes, materials, boundary conditions and loads, the simulation in SolidWorks was approached. At the beginning, a simulation was performed for the recommended size of the elements, while the continuous load was shown in the form of pressure on the upper surface with the amount of 0.8 [MPa]. After running the analysis, program gave results in form of maximum equivalent HMH stress and displacements, from that point, the comparison of known analytical expressions from the strength of materials (1D) and the theory of elasticity (2D) with the obtained results from the simulation was approached. The comparison was performed at 6 key positions shown in Figure 1. With the help of software tools offered by SolidWorks, the results of HMH stress was extracted at half the length of the beam, in the middle part of the upper and lower surface and the middle part of the half beam height. Table 3 shows the stress comparison between results obtained by well-known analytical equations and numerical approach with their relative difference.

Tab. 3. Stress results comparison between 1D, 2D and 3D model

1D			2D (zha2015)				3D			3D vs 2D, relative		
$\sigma_{HMH,TOP}$ [MPa]	$\sigma_{HMH,MID}$ [MPa]	$\sigma_{HMH,DOW}$ [MPa]	$\sigma_{HMH,TOP}$ [MPa]	$\sigma_{HMH,MID}$ [MPa]	$\sigma_{HMH,DOW}$ [MPa]	Element size [mm]	$\sigma_{HMH,TOP}$ [MPa]	$\sigma_{HMH,MID}$ [MPa]	$\sigma_{HMH,DOW}$ [MPa]	$\Delta\sigma_{HMH,TOP}$ [%]	$\Delta\sigma_{HMH,MID}$ [%]	$\Delta\sigma_{HMH,BOT}$ [%]
48,556	9,318	48,151	48,396	9,318	47,991	2	48,396	9,433	47,983	0,000	1,230	-0,017
48,556	9,318	48,151	48,396	9,318	47,991	5	48,393	9,719	48	-0,006	4,299	0,019
48,556	9,318	48,151	48,396	9,318	47,991	10	48,389	9,318	47,96	-0,014	-0,004	-0,065
48,556	9,318	48,151	48,396	9,318	47,991	15	48,397	9,34	47,979	0,002	0,232	-0,025
48,556	9,318	48,151	48,396	9,318	47,991	21,77	48,408	9,399	47,924	0,025	0,865	-0,140
48,556	9,318	48,151	48,396	9,318	47,991	30	48,387	9,488	47,944	-0,019	1,820	-0,098
48,556	9,318	48,151	48,396	9,318	47,991	40	48,358	9,518	48,075	-0,079	2,142	0,175
48,556	9,318	48,151	48,396	9,318	47,991	50	48,385	9,353	47,987	-0,023	0,372	-0,008
48,556	9,318	48,151	48,396	9,318	47,991	60	48,337	9,673	48,061	-0,122	3,806	0,146
48,556	9,318	48,151	48,396	9,318	47,991	70	48,39	9,442	48,007	-0,012	1,327	0,033
48,556	9,318	48,151	48,396	9,318	47,991	80	48,389	9,661	47,847	-0,014	3,677	-0,300
48,556	9,318	48,151	48,396	9,318	47,991	90	48,371	10,168	47,694	-0,052	9,118	-0,619
48,556	9,318	48,151	48,396	9,318	47,991	100	47,642	10,88	48,718	-1,558	16,759	1,515

Regarding the results obtained by the recommended mesh size (marked yellow), it is obvious that they are not the best by looking into the relative difference. It is easy to notice that the 15 [mm], element size (green marked) is more accurate with respect to the analytical results. Furthermore, Figures 11 through 13 present the calculated stress curves for the different 3D mesh sizes against 1D and 2D results.

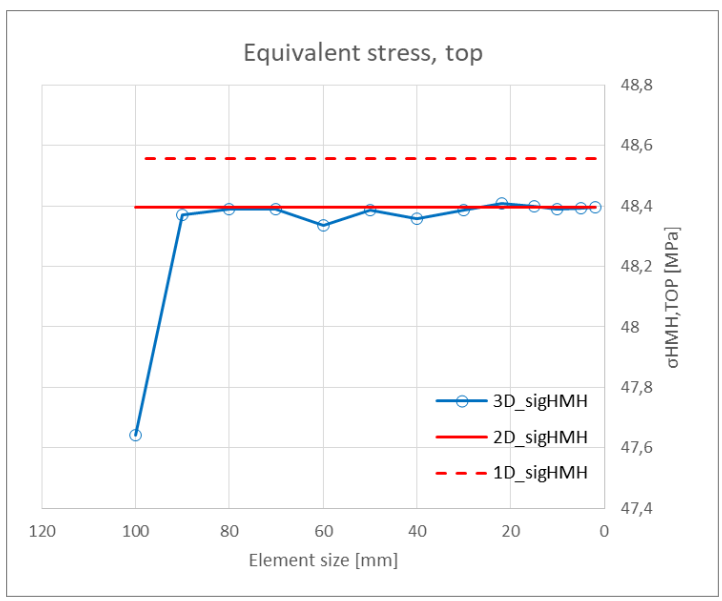

Fig. 11. Comparison between different element sizes for HMH stress values of top point

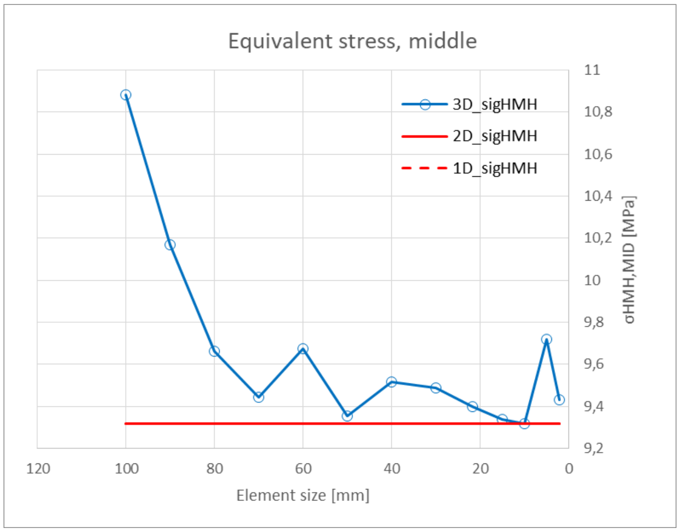

Fig. 12. Comparison between different element sizes for HMH stress values of middle point

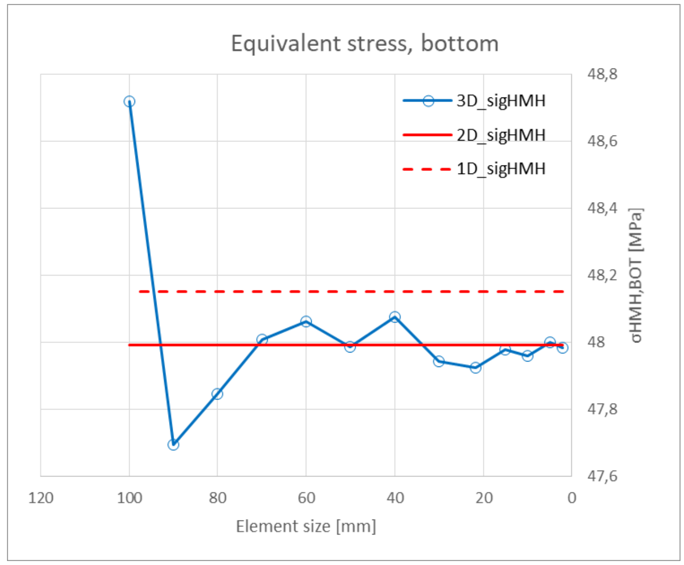

Fig. 13. Comparison between different element sizes for HMH stress values of bottom point

Also as the potentially relevant values for comparison, the displacements at the free end of the beam were also drawn for three points on the centre of weight of the cross section of the model. The relative displacement of the upper and lower points is horizontal, while the midpoints are pure vertical. Next table shows displacements comparison between results obtained by well-known analytical equations and numerical approach.

Tab. 4. Displacements results comparison between 1D, 2D and 3D model

1D			2D (zha2015)			Element size [mm]	3D		
V_{mid} [mm]	U_{top} [mm]	U_{down} [mm]	V_{mid} [mm]	U_{top} [mm]	U_{down} [mm]		V_{mid} [mm]	U_{top} [mm]	U_{down} [mm]
8,832	-0,657	0,657	9,277	-0,718	0,669	2	9,869	-0,728	0,725
8,832	-0,657	0,657	9,277	-0,718	0,669	5	9,869	-0,728	0,725
8,832	-0,657	0,657	9,277	-0,718	0,669	10	9,867	-0,728	0,725
8,832	-0,657	0,657	9,277	-0,718	0,669	15	9,866	-0,727	0,725
8,832	-0,657	0,657	9,277	-0,718	0,669	21,77	9,861	-0,727	0,725
8,832	-0,657	0,657	9,277	-0,718	0,669	30	9,862	-0,727	0,725
8,832	-0,657	0,657	9,277	-0,718	0,669	40	9,855	-0,727	0,724
8,832	-0,657	0,657	9,277	-0,718	0,669	50	9,845	-0,726	0,724
8,832	-0,657	0,657	9,277	-0,718	0,669	60	9,847	-0,726	0,724
8,832	-0,657	0,657	9,277	-0,718	0,669	70	9,831	-0,726	0,723
8,832	-0,657	0,657	9,277	-0,718	0,669	80	9,825	-0,725	0,723
8,832	-0,657	0,657	9,277	-0,718	0,669	90	9,822	-0,725	0,722
8,832	-0,657	0,657	9,277	-0,718	0,669	100	9,813	-0,725	0,722

From the figure 12, it can be seen that displacements impact on verification of mesh size has no major impact so only stress comparison has been taken as relevant.

4.1 Offshore structure part

As already mentioned in the introduction, offshore structure part was analysed implementing mesh sizes 25%, 50%, 150% and 200% of the original recommended element size in this case, i. e. 87,62 [mm]. Regarding boundary conditions, an offshore structure part is subjected to equivalent load from the hydraulic cylinders. The loading position is on the bore opening of the ear-shaped carrier. Constrained nodes are placed on the surface that is in contact with the surface that holds on to the jack-up leg.

Tab. 5. Maximum stress values for different mesh size of an offshore structure part

	Element size [mm]	σ_{HMH} [Mpa]
Best from 1D/2D	15	273,649
25%	21,905	223,802
50%	43,81	154,17
Recommended	87,62	130,875
150%	131,43	121,599
200%	175,24	137,953

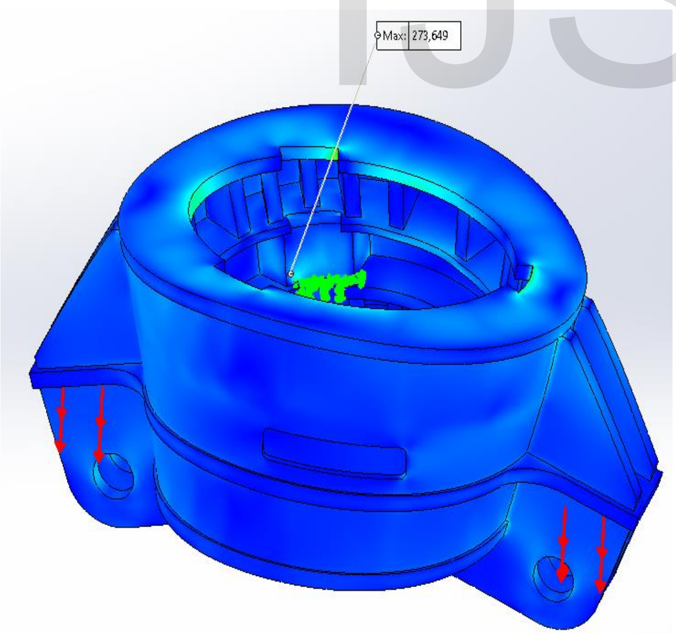

Fig. 16. Stress result of an offshore structure part for a mesh size of 15 [mm]

5 DISCUSSION OF THE RESULTS

Displacements of the free end in the 3D model do not significantly change by decreasing of the element size and do not significantly converge to their analytical values in either 2D or 1D model. The relative difference of the displacements (3D vs. 2D) is in the range 5.7% ... 8.3%. This difference is still fair, but the one found in the HMH stresses is much less.

The 1D analytical model gives a fair reference to judge the final numerical 3D results. In order to obtain a better reference, the 2D model and its analytical solution would be a far better reference in terms of equivalent stresses.

The element size proposed by SolidWorks (21.77 mm) does not provide the best accuracy in terms of equivalent stresses in the numerical solution of the analytical example. This is the most important outcome of the entire analysis performed.

The element size of 15 mm (68% of the one recommended by SolidWorks) provides a better approximation to the solution, as seen both from the graphs and relative change of HMH stresses (3D vs. 2D).

Reducing the element size below 15 mm decreases numerical accuracy, possibly owing to increased number of equations for the FEM to solve, as visible from the relative change of HMH stresses (3D vs. 2D). The numerical solution oscillates around the analytical one.

When the actual (marine structure) part is modelled by the 68% element size of the one recommended by SolidWorks, the maximal HMH stress changes from 130,875 [MPa] to 273.649 [MPa]. This means that regarding safety, materials with higher yield strength should be considered. This proves that the qualified decision about the selected FE mesh size cannot be brought solely upon some fast and shallow convergence testing based upon stresses only.

5 CONCLUSIONS

The analysed example proves the importance of convergence checking in the FEM analyses. Without this, the user would not be able to judge about the best compromise between the element size and the accuracy.

Numerical solution with larger elements reduces accuracy due to coarse mesh, whereas the smaller elements increase the number of equations to be solved, thus compromising accuracy again. All of this has been well-known and proven in practice.

The users of 3D modelling tools, such as SolidWorks shall never rely solely upon the element size recommended by the program itself and take it for granted. It is important to analyse the solution and its convergence by the comparison of the outcome for 1 or 2 or even several coarser and finer meshes.

The paper proposes the different approach to the convergence checking: comparison to a known analytical solution, rather than only increasing and decreasing element size.

The analysed example showed that incomparably better results in terms of convergence checking are obtained by comparing the HMH stresses numerical 3D vs. analytical 2D values, than comparing the relevant displacements.

Implementing thus obtained element size, rather than the proposed one, in reality, would provide a better solution in terms of safety (or optimisation), as shown in the presented show case of an actual marine offshore structural part.

Further on, obtaining analytical solutions in future, relevant for the testing of accuracy in the presented way will be worth investing the efforts to prepare these analytical solutions for the reference in future.

This paper gives some deeper introduction in using of different mesh size, but to validate real results, the structural part should be manufactured, assembled exposed to its actual loading conditions and thus be tested in reality. This will be the matter of further work.

REFERENCES

1.: Willis J., Dogra S.: *SOLIDWORKS Simulation 2018: A Power Guide for Beginners and Intermediate Users 2nd Edition* , CreateSpace Independent Publishing Platform; 2nd edition (February 23, 2018)
2. Alfirević I.: *Mechanics of Materials 1 (in Croatian)*, Tehnička knjiga, Zagreb, 1989.
3. T. A. Philpot, Mechanics of Materials - An Integrated Learning System, 4th edition, John Wiley & Sons, Inc., Hoboken, New Jersey, 2017.
4. MDSolids: Educational Software for Mechanics of Materials, https://web.mst.edu/~mdsolids, accessed on 18th June 2022
5. Jecić S.: *Theory of Elasticity*, Sveučilišna naklada Liber, Zagreb, 1990.
6. Zhan, Cx., Liu, Yh.: *Plane elasticity solutions for beams with fixed ends*. J., Zhejiang Univ. Sci. A 16, 805–819 (2015).