INNOVATIVE MECHANICAL ENGINEERING **ISSN 2812-9229 (Online)**
University of Niš, Faculty of Mechanical Engineering
VOL. 2, NO 1, 2023, PP. 52 - 63

Original scientific paper *

CAD - BASED FE MODELING OF THIN-WALLED STRUCTURES

Predrag Milić[1], Dragan Marinković[1,2], Žarko Ćojbašić[1]

[1]Faculty of Mechanical Engineering, University of Niš, Serbia

[2]Department of Structural Analysis, Berlin Institute of Technology, Berlin, Germany

Abstract. *The paper presents an overview of the advantages and disadvantages of using different finite element types for the discretization of thin-wall structures. Thin-walled structures are usually described in modern CAD program packages as a solid geometry. Boundary surfaces of a solid geometry are usually defined as a nonuniform rational basis spline (NURBS). Contrary to CAD models, FE modelling of thin-walled structure models is often performed using the shell element type with different formulations. The paper provides an overview of the obtained results and the convergence speed of the finite element method using different isoparametric and isogeometric types of finite elements.*

Key words: *Finite element, Shell element, Isogeometric analysis, Thin-walled structures*

1. INTRODUCTION

The structural analysis of thin-walled structural elements by using the finite element method is most often carried out with a shell element type. Usually, the reason for using this type of element is in the limitation of computing resources. A complex thin-walled structure modeled with this type of element has fewer numerical requirements compared to a model discretized with solid elements. Meshing with a shell element type requires some CAD modifications. The process consists of two stages: detail removal and dimension reduction. In the detail removal stage, small features such as fillets or holes, which do not affect the analysis results considerably, are excluded from the model. In the dimension reduction stage, the complex 3D-field is condensed to essential ingredients of the structural response described by a 2D-approach. These models are computationally efficient and allow rapid design optimization of the structure.

In the design process, a 3D CAD model is usually created due to the development of the production process. The main difference, compared to the initial design, is that for the

***Received: February 27, 2023 / Accepted April 03, 2023.**
Corresponding author: Predrag Milić
Faculty of Mechanical Engineering, University of Niš, Serbia
E-mail: predrag.milic@masfak.ni.ac.rs

meshing with the shell finite element it is necessary to define the middle surface of thin-walled structures. The advantage obtained by using a shell element type in terms of solution execution time is partly reduced by the necessity to create a suitable idealization from a 3D-model. In practice, a shell element type with six degrees of freedom at the node is mostly used [1, 2, 3, 4, 5]. Conditionally, a shell finite element with only translational degrees of freedom in the nodes forms a structure with a smaller number of unknowns [6, 7, 8]. On the other hand, the size of the stiffness matrix band is usually larger due to the higher degree of continuity at the element boundary, which requires a longer computing time. The shell type of a finite element is also applicable to structures made of complicated material systems [9, 10]. The so-called "solid-shell" element type, which is between the solid and the shell element type, has only translational degrees of freedom [11]. The degree of element basic functions is an important parameter that affects the quality of the solution. The classical FEM is mainly based on the application of isoparametric finite elements [12]. These types of elements use the same shape functions, typically Lagrange polynomials, to approximate both the displacement field and the geometry of element. A special area of finite element analysis is isogeometric analysis [13]. This type of analysis uses the same basic functions to describe the CAD geometry, the displacement field approximation, as well as the geometry of the element itself [14, 15, 16, 17]. Isogeometric analysis uses finite elements of higher degree basis functions with a higher degree of continuity at the element boundaries. This feature of isogeometric analysis gives a number of advantages in terms of convergence speed compared to the classical finite element method. The paper presents an extract from the conducted investigation.

2. Finite Element Types for Thin-Walled Solids Modeling

As previously mentioned, the discretization of thin-walled structures is most often performed with a solid or shell element type. Among different types of 3D finite elements, the authors were primarily focused on tetrahedral and hexahedral elements. Tetrahedral elements are candidates due to their capability of modeling any geometry and a rather low numerical effort required for one element. Hexahedral elements, on the other hand, are candidates due to relatively high accuracy they offer, which is, however, paid by a higher numerical effort for one element. Of course, the ratio between accuracy and numerical effort for the whole model eventually becomes the decisive factor. Wedge elements are not particularly considered, but it is clear that they, together with tetrahedral elements, would be the choice in situations where the geometry meshing require them.

As already mentioned, modeling of thin-walled structures with a shell element type requires appropriate modifications of the initial CAD geometry. Depending on the ratio of plate thickness and structure dimensions, stress changes in the direction normal to the middle surface of the shell can be different. Depending on the stress change, different types of shell elements are used for modeling thin-walled structures.

2.1. Tetrahedral elements in modeling thin-walled solids

As already emphasized, it is the numerical efficiency of a single element combined with high meshing ability that renders this type of element a candidate for the investigation. However, the limitation imposed on the aspect ratio of the tetrahedral element by commercial meshers requires the elements to be similar in size in all directions. The authors kept the aspect ratio less than 2. This means that many elements are required to model a region that has large lateral dimensions but small thickness. For instance, when modeling a plate with in-plane dimensions of 1500×1000 mm and with thickness of 15 mm, this requirement results in 79200 elements with the aspect ratio equal to 1 and one element over the thickness, or in 9700 elements by loosing the requirement for aspect ratio to 2. Hence, though the numerical effort for a single element is rather small, the complete model could not offer any advantage regarding this aspect. Furthermore, the element demonstrates too stiff behavior. With linear shape functions, this is obviously to be attributed to the fact that the linear tetrahedral element is capable of representing only constant strain states. For the above mentioned plate at least 4 linear elements over the thickness, which means a total number of elements in the order of 10^6 for the whole model, were necessary to obtain results comparable with those from the shell element. The quadratic tetrahedral element does not require such a fine mesh, but the overall numerical effort is still very high since a single element has a greater number of degrees of freedom compared to the linear element. Therefore, as expected, it was promptly decided to switch the focus to the hexahedral element.

2.2. Hexahedral elements in modeling thin-walled solids

As for the mesh with the hexahedral element, it was decided to use the strategy that implies the same in-plane mesh as with the shell element and to start with one element across the thickness. The calculation of examples should then point out if a greater number of elements across the thickness are required, when it is aimed at global behavior of thin-walled structures. Since the quadratic shell element with reduced integration is used for comparison purposes, it was decided that the hexahedral element should also use quadratic shape functions as well as the reduced integration technique. The comparison of results is done for linear and geometrically nonlinear computations.

2.3 Shell elements in modeling thin-walled structures and isogeometric approach

Modeling of thin-walled structures is most often done with thin Kirchhoff-Love shell elements or the thick Reissner-Mindlin shell element type. Other types of higher-order shell element types are less common in commercial software packages and are used mostly for scientific considerations. If the normal to the middle surface of the undeformed model remains normal to the middle surface and unstretched in the deformed state, the Kirchhoff-Love shell type model can be used. This shell model requires C^1 continuity at the element boundaries and has three degrees of freedom at the node. Due to the continuity condition, which is difficult to achieve for standard finite elements based on Lagrange shape functions, this type of element is less common. The Reissner-Mindlin element type incorporates transverse shear strains, which cannot be neglected in thick shell analysis. Due to the six degrees of freedom in the node, it requires more computing resources. This type of element with a reduced number of integration points was used in the paper. The reduced number of integration points is a measure of reducing the effect of shear locking.

Modeling of thin plates and shells in the isogeometric approach can be performed with a solid or shell element type. What makes this finite element method different from the classical one is that it can achieve a high degree of continuity at the element boundaries. The basic functions of an element are predetermined for a defined surface and, depending on the degree of the basic functions and the knot vector, they extend through several elements. This is also the reason why this type of analysis is suitable for the application of the Kirchhoff-Love element type which requires C^1 continuity at the element boundary and has three degrees of freedom at the node [8, 18,19].

Due to the comparison of the results with the shell-type element in the commercial software package, we decided to use the Reissner-Mindlin isogeometric element type with a second-order of basic functions in both directions [20, 21, 22]. Changing the density of the mesh and the degree of the NRUBS basic functions is achieved by the techniques of knot inserting in the knot vector (parametric space) as well as by increasing the degree of the surface basic functions. In this way, a new surface is obtained, which is identical to the previous one in the geometric sense, but is defined with a larger number of elements and different degrees of basic functions. In the case of curved NURBS surfaces, not all control points are located on the surface itself. This makes it impossible to place the load in the control point (it does not have a direct projection on the surface itself) without violating the achieved degree of continuity at the boundaries of the elements. For this reason, the surface pressure and concentrated forces at the boundary of the structure were used as load.

Using higher-order basis functions reduces the risk of shear-locking effects. Several authors have presented the procedure for reducing the number of integration points and the application of special quadrature formulas, taking into account that the basic functions extend through several elements [23, 24]. In order to compare the results with the classic finite element method, a NURBS element with second-order basic functions was used in the paper. The results obtained by the 20-node quadratic solid element are compared with the results obtained with the 8-node quadratic shell element from the ABAQUS finite element library. Both finite elements have a reduced number of integration points.

3. SOLUTION CONVERGENCE - EXAMPLES

The paper presents three characteristic examples of a flat plate, a single and a double curved shell. All models are formed with isoparametric shell element type (S8R), isogeometric shell type (NURBS) and solid element type (C3D20R) of single, double and triple elements in the direction of thickness. The study provides an overview of the results of linear and non-linear static analysis for all models.

3.1 An example of a double-sided clamped plate.

The first observed structure is the plate with dimensions 1500×1000×15 mm, made of steel (with modulus of elasticity $E=2.1\cdot10^5$ N/mm^2 and Poisson coefficient $\nu=0.33$). The plate is exposed to a constant surface pressure p=120 kPa and clamped over two shorter edges (Fig. 1a). The pressure was chosen so that noticeable geometrically nonlinear effects would occur during deformation and, hence, linear and geometrically nonlinear results show significant difference. Point A, which is a mid-point of the plate free edge, is chosen as a representative point to observe its deflection, w.

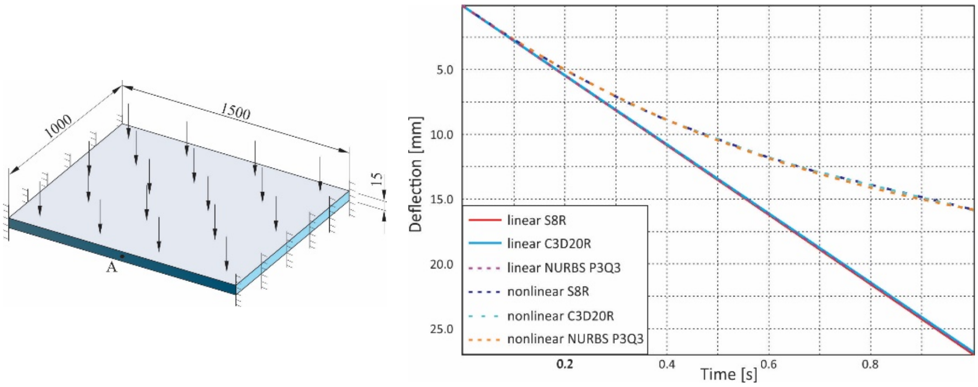

Fig. 1 a) Double-sided clamped plate exposed to surface pressure;
b) linear (straight line) and nonlinear (curved line) analysis of point A deflection

The example is first computed with quadratic shell elements and results for the mesh with 8×12 elements may be accepted as a representative solution for both linear and nonlinear computation. Of course, in linear analysis the behavior is bending dominated, whereas in nonlinear analysis the membrane behavior rapidly gains in importance as deformation progresses, thus rendering the structure stiffer. The diagram in Fig. 1b depicts the obvious differences between linear and geometrically nonlinear results. Table 1 shows that the results from the hexahedral element with only 1 element across the thickness converge to the same values as the results from the shell element, though the convergence is slower, as expected. It may be noticed that the hexahedral element with the mesh of 8×12×1 elements yields a displacement that is 3.6 % stiffer for the linear and 1.6 % stiffer for the nonlinear case compared to the result from the model with shell elements.

Table 1 Results of linear and non-linear analysis of various models of a double-sided clamped plate

		shell (ABAQUS S8R)		3D solid ABAQUS C3D20R x 1		3D solid ABAQUS C3D20R x 2		NURBS P2Q2	
Mesh in-plane	In-plane number of elem.	linear	non-linear	linear	non-linear	linear	non-linear	linear	non-linear
2×3	6	25.68	15.56	21.69	14.61	24.55	15.32	25.73	15.59
4×6	24	26.84	15.81	24.99	15.36	26.30	15.69	26.86	15.81
8×12	96	26.92	15.81	25.98	15.56	26.63	15.75	26.93	15.81
16×24	384	26.94	15.82	26.46	15.69	26.78	15.78	26.94	15.82
32×48	1536	26.94	15.82	26.70	15.76	26.85	15.80	26.94	15.82
64×96	6144	26.94	15.82	26.82	15.80	26.89	15.82	26.94	15.82
128×192	24576	26.94	15.82	26.88	15.81	26.90	15.82	26.94	15.82

The results for meshes with 2 hexahedral elements over the thickness show that the convergence is obviously somewhat faster and for the mesh of 8×12×2 elements the result is 1.1 % stiffer in the linear and 0.4 % stiffer in the nonlinear case, again compared to the result from the model with shell elements. Hence, in this case, the model with hexahedral elements that uses the same in-plane mesh as the shell element can satisfactorily describe the global behavior of the considered structure. The number of elements across the thickness is to be determined by accuracy requirements, but one may notice that one element over the thickness already suffices to generally represent the structure's global behavior.

The diagrams in Fig. 2 depict the convergence of results as a function of the in-plane mesh, whereby a different number of elements across the thickness have been considered. The shell element demonstrates quite fast convergence and the convergence of the hexahedral element improves as the number of elements across the thickness increases from 1 element (denoted with 1ED) to 3 elements (denoted with 3ED).

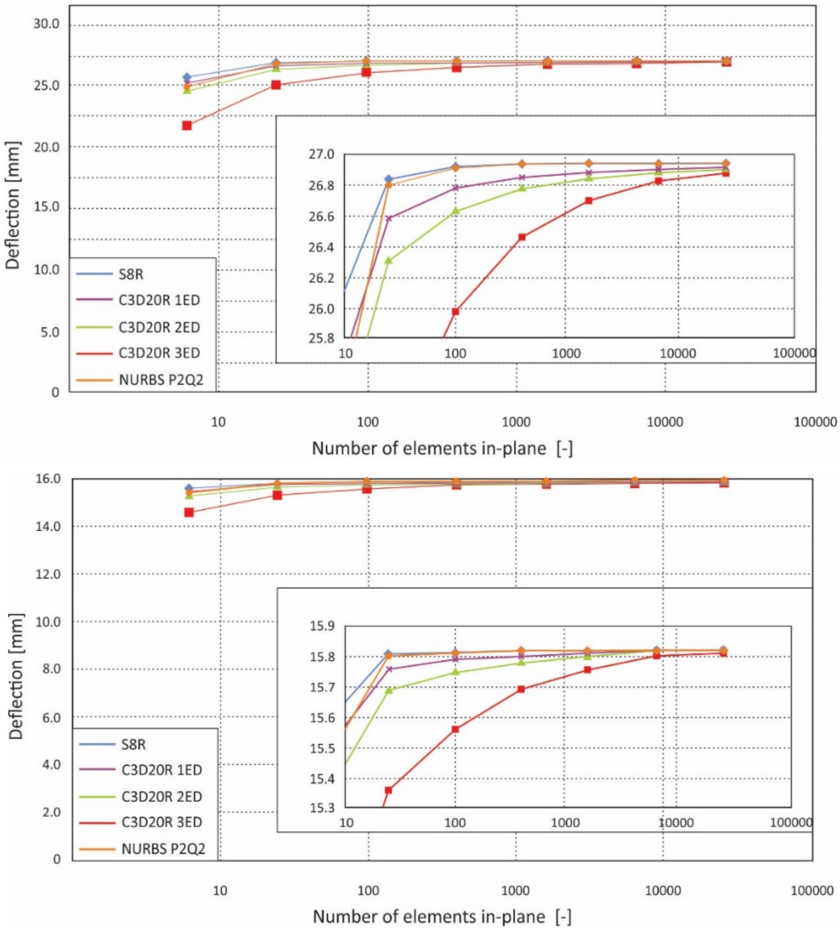

Fig. 2 Plate structure – point A deflection as a function of in-plane mesh: a) linear analysis; b) geometrically nonlinear analysis

As a part of the conducted investigation, the authors have also considered the same structure but with different sets of boundary conditions and load cases. All those cases led to very similar results and are, for the sake of brevity, not given here.

3.2 Example of a shell curved in one plane

A curved shell structure implies more complicated structural behavior as well as induced strain and stress states. Such a structure resists the external excitations in a more complex way. Those are also typical structures in the automotive industry, the aerospace industry, etc. Therefore, in the next step the authors have considered a shell structure curved in only one plane.

The considered shell structure has the same dimensions as the previously considered plate structure but is curved around one axis with the radius of curvature r=9.75 m (Fig. 3). It is exposed to the same excitation, i.e. the surface pressure of p=120 kPa. The shell is clamped along the two shorter edges.

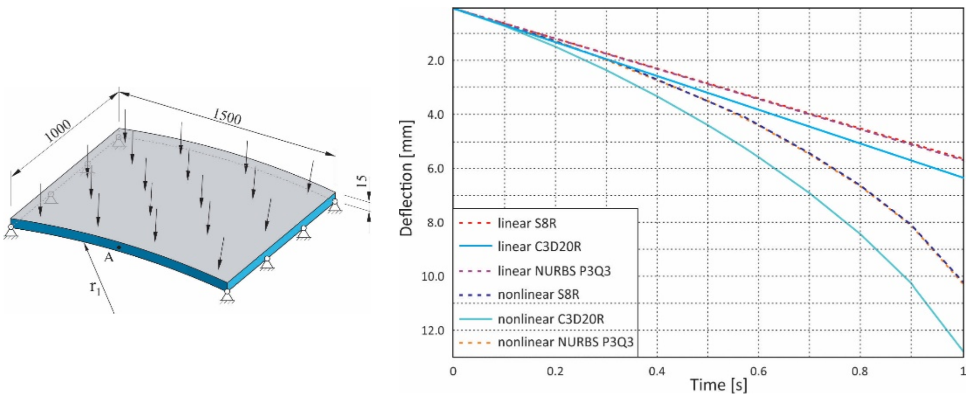

Fig. 3 Simply curved shell exposed to surface pressure; b) linear (straight line) and nonlinear (curved line) deflection of point A

The same pattern of analysis used in the first example has been followed here as well. Interestingly, the convergence from both shell and hexahedral elements progresses at very similar pace and, furthermore, the result from the shell FEM model is somewhat stiffer. This trend is to be recognized from Table 2 and from the diagrams in Fig. 4. Point A, upon which the external excitation acts, is chosen as a representative one. By analyzing the obtained results, it can be seen that there are significant differences between linear and non-linear analysis, as well as between the results obtained with solid elements and with shell elements. The reasons are multiple. Greater deformation than the thickness dimension of the shell causes special stress-deformation states in the element which are manifested differently in linear and non-linear analysis. In the case of a very fine mesh, there are changes in the aspect ratio of the elements, which leads to deviations in convergence.

Table 2 Results of linear and non-linear analysis of various models of a double-sided supported simply curved shell

		shell (ABAQUS S8R)		3D solid ABAQUS C3D20R x 1		3D solid ABAQUS C3D20R x 2		NURBS P2Q2	
Mesh in-plane	In-plane number of elem.	linear	non-linear	linear	non-linear	linear	non-linear	linear	non-linear
2×3	6	5.684	9.649	5.829	9.614	5.528	8.643	5.401	9.410
4×6	24	5.673	10.12	6.105	11.830	6.065	11.470	5.668	10.190
8×12	96	5.675	10.18	6.113	12.030	6.174	12.040	5.675	10.195
16×24	384	5.677	10.20	6.123	12.070	6.173	12.250	5.680	10.205
32×48	1536	5.678	10.20	6.143	12.140	6.193	12.330	5.683	10.209
64×96	6144	5.678	10.20	6.206	12.370	6.228	12.450	5.684	10.209
128×192	24576	5.678	10.20	6.502	13.530	6.290	12.690	5.684	10.210

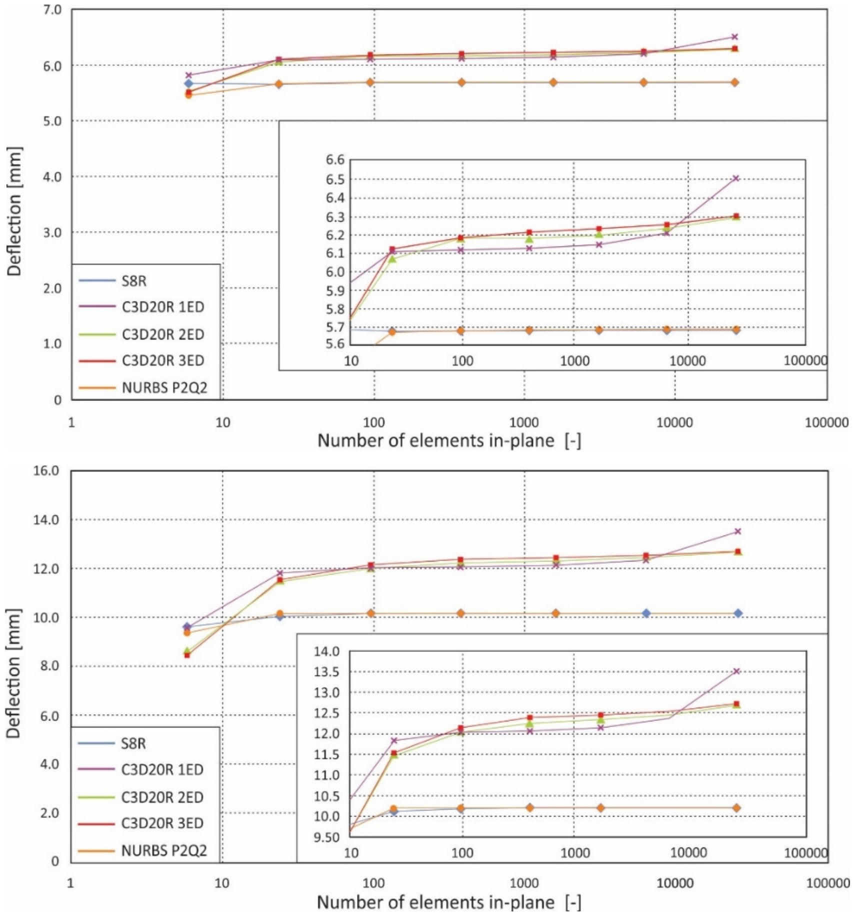

Fig. 4 Simple-curved shell – deflection of point A as a function of in-plane mesh: a) linear analysis; b) geometrically nonlinear analysis

3.3 Example of a double-curved shell

The level of complexity of structural behavior is once again raised by considering a structure that is double-curved in space. So, the dimensions are the same as in the previous examples, but the shell is curved around two perpendicular axes with radii of r_1=7 m (for the longer pair of edges) and r_2=5.5 m (for the shorter pair of edges). The structure is clamped over a shorter edge and is exposed to a single vertical force F=5000N acting at a free corner point of the structure (Fig. 5).

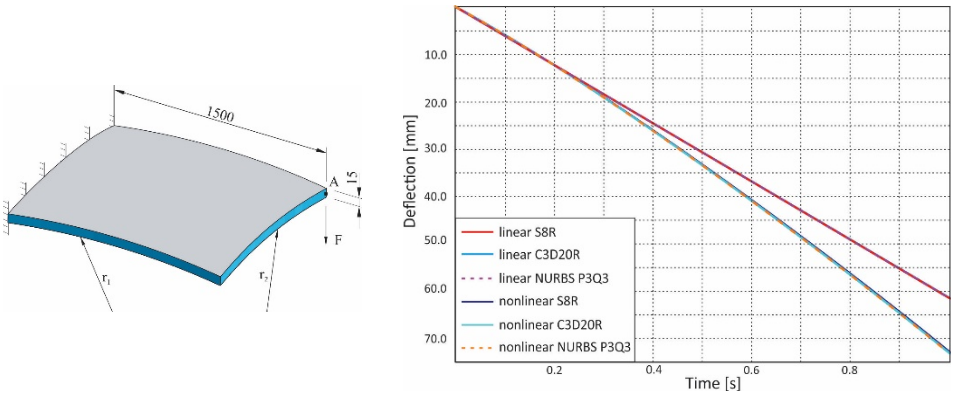

Fig. 5 a) Double-curved shell exposed to single force; b) linear (straight line) and nonlinear (curved line) deflection of point A

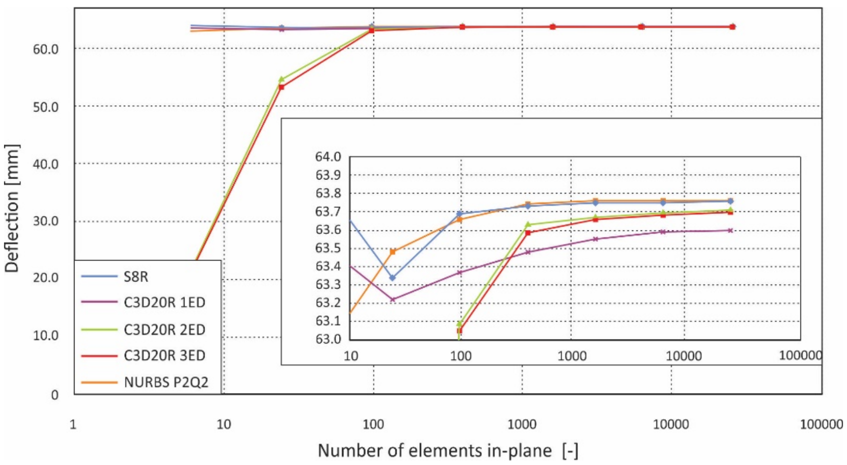

Fig. 6 Double-curved shell – deflection of point A as a function of in-plane mesh: linear analysis

The same analysis pattern has been followed again and, for the sake of brevity, the results are given in the form of diagrams in Fig. 6 and 7. An interesting convergence behavior of the hexahedral element should be observed in this and the previous example.

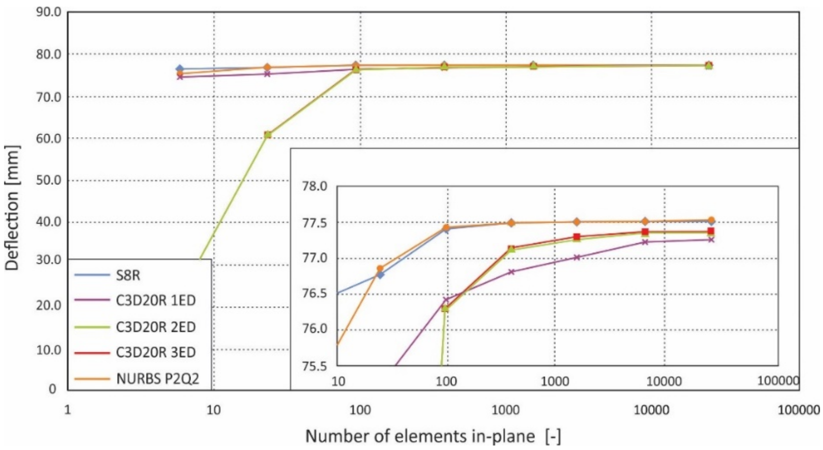

Fig. 7 Double-curved shell – deflection of point A as a function of in-plane mesh: nonlinear analysis

Table 3 Results of linear and non-linear analysis of various models of a double-curved shell exposed to single force

		shell (ABAQUS S8R)		3D solid ABAQUS C3D20R x 1		3D solid ABAQUS C3D20R x 2		NURBS P2Q2	
Mesh in-plane	In-plane number of elem.	linear	non-linear	linear	non-linear	linear	non-linear	linear	non-linear
2×3	6	63.84	76.36	63.51	74.51	21.68	21.34	62.95	75.18
4×6	24	63.34	76.76	63.22	75.21	54.49	53.26	63.48	76.85
8×12	96	63.69	77.41	63.37	76.42	63.05	63.09	63.66	77.43
16×24	384	63.73	77.49	63.48	76.81	63.59	63.63	63.74	77.5
32×48	1536	63.75	77.51	63.55	77.01	63.66	63.67	63.76	77.51
64×96	6144	63.75	77.51	63.59	77.22	63.68	63.69	63.76	77.52
128×192	24576	63.76	77.52	63.6	77.26	63.7	63.71	63.76	77.53

One may notice that the FEM model with one hexahedral element across the thickness has very similar convergence pace as the shell element, whereas a greater number of hexahedral elements across the thickness give stiffer results for the coarsest meshes. Of course, as the in-plane mesh gets finer, those models converge to a result quite close to the result from the model with only one element over the thickness. At first glance, the result may cause some raised eyebrows. However, this is a consequence of the induced complex stress state combined with non-recommendable aspect ratio for the hexahedral element. For the coarsest meshes, ABAQUS issues warning that the aspect ratio of 100 has been exceeded. Obviously, such a breach of meshing recommendations becomes an issue with more complex stress states, since no similar effect was recognized in the first considered case, which involves less complex stress states induced in the structure, but is seen in the second and third example.

4. CONCLUSIONS

The paper presents an extract from the authors' investigation into an adequate choice of a standard 3D finite element and isogeometric meshing strategy for modeling thin-walled structures, conditioned by a given 3D CAD model, i.e. without the extraction of the reference surface. Since it is a relatively short extract, it was primarily aimed at demonstrating the applied approach to the problem and giving some of the most important conclusions.

It is a common knowledge in the FEM community that tetrahedral elements are typically rather stiff and that they require a rather fine mesh (and hence a great numerical effort) to produce sufficiently accurate results. However, they are, together with wedge elements, necessary in a number of situations where geometry meshing requires their application. The first few cases in the authors' investigation already confirmed this common knowledge.

Therefore, the rest of the investigation was focused on hexahedral elements. Although not presented here, the linear element and full integrated elements were also examined more closely. Their behavior was rather stiff, thus requiring very fine meshes. So, the final choice and the authors' recommendation is the quadratic hexahedral element with the reduced integration technique included. Furthermore, one should follow the recommendation regarding the aspect ratio for the element. This recommendation is particularly important in relation to the number of elements across the thickness of the structure. A greater number of elements over the thickness is expected to result in better accuracy, but, by keeping the same in-plane mesh, this might jeopardize the mentioned recommendation and, with complex stress states, the accuracy may unexpectedly deteriorate. The deformed configuration, i.e. the displacement field, was used in the paper as a criterion to determine the adequacy of 3D element choice and meshing strategy. Though not presented here, in their investigation, the authors also considered the computed strain and stress fields, which represent a stricter criterion. This aspect did not affect the nature of the conclusions drawn here.

Acknowledgement: *This research was financially supported by the Ministry of Science, Technological Development and Innovation of the Republic of Serbia (Contract No. 451-03-47/2023-01/ 200109) and by the Science Fund of the Republic of Serbia (Serbian Science and Diaspora Collaboration Program, Grant No. 6497585)..*

REFERENCES

1. Ahmad, S., Irons, B. M., Zienkiewicz, O. C., 1970, *Analysis of thick and thin shell structures by curved finite elements*, International journal for numerical methods in engineering, 2(3), 1970, pp. 419-451.
2. Bucalem, M. L., Bathe, K. J., 1997, *Finite element analysis of shell structures,* Archives of Computational Methods in Engineering, 4(1), pp. 3-61.
3. Rama, G., Marinkovic, D., Zehn, M., 2016, *Efficient co-rotational 3-node shell element*, American Journal of Engineering and Applied Sciences, 9(2), pp. 420-431.
4. Rama, G., Marinkovic, D., Zehn, M., 2018, *A three-node shell element based on the discrete shear gap and assumed natural deviatoric strain approaches*, Journal of the Brazilian Society of Mechanical Sciences and Engineering, 40, 356.
5. Rama, G., Marinkovic, D., Zehn, M., 2018, *High performance 3-node shell element for linear and geometrically nonlinear analysis of composite laminates*, Composites Part B: Engineering, 151, pp. 118-126.

6. Kirchhoff, G. R, 1850, *Uber das Gleichgewicht und Bewegungen einer elastischen Scheibe*, Journal für die reine und angewandte Mathematik, 40, pp. 51-88.
7. Love, A. E. H., 1888, *On the small free vibrations and deformations of elastic shells*, Philosophical trans. of the Royal Society (London), Série A, 179, pp. 491–549.
8. Kiendl, J., Bletzinger, K.-U., Linhard, J., Wüchner, R., 2009, *Isogeometric shell analysis with Kirchhoff–Love elements,* Computer Methods in Applied Mechanics and Engineering, 198(49-52), 3902-3914.
9. Giap, X.H., Zehn, M., Marinkovic, D., Fragassa, C., 2019, *Dealing with Nap-Core Sandwich Composites: How to Predict the Effect of Symmetry*, Materials, 12(6), 874.
10. Giap, X.H., Marinkovic, D., Zehn, M., 2019, *Parametric investigations of mechanical properties of nap-core sandwich composites*, Composites Part B: Engineering, 161, pp. 427-438.
11. Bischoff, M., 1999, *Theorie und Numerik einer dreidimensionalen Schalenformulierung*, Dissertation, Institut für Baustatik, Universität Stuttgart.
12. Zienkiewicz, O. C., Taylor, R. L., 1989, *The finite element method - Volume 1: Basic formulation and linear problems*. McGraw-Hill, New York.
13. Hughes, T.J.R., Cottrell, J.A., Bazilevs, Y., 2005, *Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement*, Computer Methods in Applied Mechanics and Engineering, 194(39–41), 4135–4195.
14. Piegl, L., Tiller, W., 1996, *The NURBS Book (Monographs in Visual Communication)*, Springer.
15. Rogers, D. F., 2001, *An Introduction to NURBS with Historical Perspective*, Academic Press.
16. Gerald, F., 1993, *Curves and surfaces for computer aided geometric design: a practical guide. 3rd. edition*, Academic Press Inc.
17. Milić, P., Marinković, D., 2013, *Isogeometric structural analysis based on NURBS shape functions*, Facta Universitatis, Series: Mechanical Engineering, 11(2), pp. 193-202.
18. Nguyen-Thanha, N., Valizadehb, N., Nguyenc, M.N., Nguyen-Xuand, H., Zhuange, X., Areiasf, P., Zih, G., Bazilevsg, Y., Lorenzisa, De L., Rabczukb, T., 2015, *An extended isogeometric thin shell analysis based on Kirchhoff–Love theory*, Computer Methods in Applied Mechanics and Engineering, 284, pp. 265-291.
19. Predrag, M., Dragan, M. 2015, *Isogeometric FE analysis of complex thin-walled structures*, Transactions of FAMENA, 39(1), pp. 15-26.
20. Benson, D.J., Bazilevs, Y., Hsu, M.C., Hughes, T.J.R., 2010, *Isogeometric shell analysis: The Reissner–Mindlin shell*, Computer Methods in Applied Mechanics and Engineering, 199(5-8), pp. 276-289.
21. Dornisch, W., Klinkel, S., Simeon, B., 2013, *Isogeometric Reissner–Mindlin shell analysis with exactly calculated director vectors*, Computer Methods in Applied Mechanics and Engineering, 253, pp. 491-504.
22. Milić, P., Marinković, D., Klinge, S., Petrović, Ž., 2023, *Reissner-Mindlin Based Isogeometric Finite Element Formulation for Piezoelectric Active Laminated Shells*, Technical gazette, 30(2), 2023. https://doi.org/10.17559/TV-20230128000280
23. Hughes, T.J.R., Reali, A., Sangalli, G., 2010, *Efficient quadrature for NURBS-based isogeometric analysis*, Computer Methods in Applied Mechanics and Engineering, 199, pp. 301–313.
24. Adam, C., Bouabdallah, S., Zarroug, M., Maitournam, H., 2015, *Improved numerical integration for locking treatment in isogeometric structural elements. Part II: Plates and shells*, Computer Methods in Applied Mechanics and Engineering, 284, 106-137.