Research on influence of mesh parameters modification on solution accuracy of finite element analysis

M Handrik, J Majko, M Vaško, F Dorčiak and P Kopas
Faculty of Mechanical engineering, University of Žilina, Univerzitná 8215/1, 010 26 Žilina, Slovakia

Corresponding author e-mail address: marian.handrik@fstroj.uniza.sk

Abstract. The paper deals with the solution accuracy of the stress in the structure using the finite element analysis. In general, hexahedron elements are more accurate than tetrahedron elements and quadratic elements are more accurate than linear elements. The primary aim of the article is to perform comparison of the obtained results and calculation parameters (such as time and so on) for different types of elements and the elements size. Usage less accurate elements like linear tetrahedron under certain circumstances could lead to sufficiently accurate result of stress analysis.

1. Introduction
Significant development and availability of computer technology have become the basis for the market expansion of finite element method. The developers of FEA programs simplify the user interface significantly and creates various modules appropriate for a wide range of applications in practice. As a result, we can recently observe that a large group of users does not have a solid theoretical knowledge of finite element method and the subsequent operation of FEA programs. We can often encounter misinterpretation of the essential features of FEM, for example:

- Hexahedron elements provide more accurate results than tetrahedron elements, therefore we must use hexahedron elements.
- Quadratic elements provide more accurate results than linear elements, so usage of quadratic elements is necessary condition.
- Mapped mesh of hexahedron elements provides more accurate results than free meshing with tetrahedral elements. Therefore, we divide body volume solely using mapped meshing.

These misinterpretations lead to erroneous decisions during the solution of computational problems. The primary aim of the paper is to prove that even linear tetrahedral elements provide appropriately accurate results, which significantly depend on correctly adjusted element size. In addition, we will show that the use of this type of element is effective from more perspectives: in terms of the time required to create a model, verification of the generated model, a task solution and so on. The calculations are performed using the FEA program ADINA 9.7 SMP [1-3].

2. Example model
In the following lines, we describe the influence of individual mesh types on results in the selected example. The first necessary step is the definition of a task. The geometry consisted of infinite volume with the hole located in the centre of the body. The volume thickness and hole diameter were specified.
as 1. The loading presented a pressure of magnitude 1 MPa. The material model was linear elastic with values typical of steel – Young modulus of elasticity $2.1 \times 10^5$ MPa and Poisson’s number 0.3.

The example was solved as a 3D solid with defined geometry dimensions 8x8. This type of task depends on the ratio between the size of geometry and the diameter of the hole. Therefore, the dimension and dimensions units had not importance. Assignment of example allowed to apply plane of symmetry two times, which led to the simplification of the model into a quarter part. The geometry is shown in Figure 1. The aims were following. Monitoring the stress concentrations and evaluation of the stress ratios in specified locations (around the hole and place of loading). Presented computation was performed on a computer with 765 GB RAM, 2 x Intel Xeon E5-2687W v3 3.10GHz processors and twenty cores.

![Geometry and boundary conditions of model example.](image)

3. Solution of example model
Firstly, we analyzed the solution accuracy depending on selected type of elements. The generated mesh comprised elements of size 0.1 with the following modifications:

1. 4-node linear tetrahedron element,
2. 8-node linear hexahedron element,
3. 10-node quadratic Serendipity tetrahedron element,
4. 11-node quadratic Lagrange tetrahedron element,
5. 20-node quadratic Serendipity hexahedron element,
6. 27-node quadratic Lagrange hexahedron element.

Since the element size was 0.1, the thickness of the body and quarter of the circle hole were divided into 10 elements. Generated number of elements around the hole was suitable for a sufficiently accurate approximation.

The computed stress concentration values for different element types with the element size of 0.1 are shown in Table 1. The theoretically calculated value of the stress concentration at the hole location is 3. Evaluation of the stress concentration was performed using equivalent von Misses stresses.
Table 1. Comparison of solution accuracy in case of element size 0,1.

| Element type | Number of elements | Number of nodes | Stress concentration value | Error [%] |
|--------------|--------------------|-----------------|---------------------------|-----------|
| 4 Node       | 555 224            | 98 354          | 2.743976                  | 0.086     |
| 10 Node      | 555 224            | 771 629         | 3.013551                  | 0.004     |
| 11 Node      | 555 224            | 1 326 853       | 3.010640                  | 0.004     |
| 8 Node       | 62 512             | 66 951          | 2.798172                  | 0.067     |
| 20 Node      | 66 618             | 207 334         | 3.003281                  | 0.001     |
| 27 Node      | 62 512             | 507 169         | 2.998281                  | 0.001     |

Based on the obtained results, firstly we observed that the worst results provide linear tetrahedron elements with a solution error of 10%. On the other hand, hexahedron or quadratic elements usage leads to a decrease of the error to 1%.

The resulting stress concentration values are shown in Figure 2. In the case of mesh consisting of quadratic hexahedron elements, part of generated mesh were two elements with an inappropriate shape and a negative value of Jacobian. Elimination of this problem is based on mesh quality check using Jacobian value and the subsequent re-meshing of inappropriately shaped elements and their surroundings.
Figure 2. Smoothed effective stress [Pa], and the corresponding value of the stress intensity factor in case of: (a) linear tetrahedrons, (b) quadratic hexahedrons.

4. Comparison of solution complexity

Determination of the demands on system resources consists of an independent evaluation of the time required to model generation. The process of model generation should be divided into the following steps: loading of the model, mesh generation and saving the model file with possible optimization of matrix bandwidth. In the Gaussian elimination method for sparse matrices, optimization of matrix bandwidth has significant importance. Firstly, we compared the results of the time needed to model generation (Table 2).

Table 2. Time [s] needed to generate the model

| Element type – number of nodes | With bandwidth optimization | Without bandwidth optimization |
|-------------------------------|-----------------------------|-------------------------------|
| 4 Node                        | 41.4                        | 40.9                          |
| 10 Node                       | 72.5                        | 69.9                          |
| 11 Node                       | 77.6                        | 74.2                          |
| 8 Node                        | 155.2                       | 154.3                         |
| 20 Node                       | 212.8                       | 210.9                         |
| 27 Node                       | 169.9                       | 164.9                         |

Secondly, we compared various types of solvers according to RAM and time consumption:
1. Sparse Gauss elimination with matrix bandwidth optimisation.
2. Sparse Gauss elimination without matrix bandwidth optimisation.
3. Multigrid solver.
4. Iterative solver based on Conjugate gradient method.
5. 3D iterative solver, special iterative solver specifically intended for quadratic elements.

The time consumption and RAM requirements depending on the solver are shown in Table 3 and Table 4, respectively.

**Table 3. Time [s] needed to solve the task**

| Element type | Input phase | Matrix assembly | Sparse optimize | Sparse non-optimize | Multigrid | Iterative | 3D iterative |
|--------------|-------------|----------------|-----------------|---------------------|-----------|-----------|-------------|
| 4 Node       | 7           | 4              | 4               | 5                   | 6         | 192       | -           |
| 10 Node      | 15          | 26             | 100             | 120                 | 100       | Non conv. | 21          |
| 11 Node      | 18          | 39             | 127             | 163                 | 197       | Non conv. | 46          |
| 8 Node       | 5           | 1              | 3               | 4                   | 9         | 97        | -           |
| 20 Node      | 7           | 13             | 27              | 34                  | 112       | Non conv. | 25          |
| 27 Node      | 8           | 31             | 60              | 64                  | 116       | Non conv. | 34          |

**Table 4. RAM requirements [MB]**

| Element type | Number of equations | Original bandwidth | Optimized bandwidth | Model storage | Sparse optimized | Sparse non-optimized |
|--------------|---------------------|--------------------|---------------------|---------------|------------------|----------------------|
| 4 Node       | 293 318             | 292 448            | 6 832               | 478           | 1 782            | 1 793                |
| 10 Node      | 2 308 257           | 2 307 591          | 52 568              | 2 788         | 34 329           | 34 522               |
| 11 Node      | 3 973 929           | 3 971 366          | 90 767              | 4 087         | 36 911           | 37 291               |
| 8 Node       | 199 181             | 197 318            | 5 310               | 199           | 1 218            | 1 195                |
| 20 Node      | 806 160             | 803 290            | 27 273              | 1 308         | 11 029           | 10 866               |
| 27 Node      | 1 515 165           | 1 465 453          | 37 356              | 2 270         | 20 173           | 19 432               |

Gauss elimination method with sparse solver requires memory for the solution of equations system. The memory size depends on the computational model and matrix bandwidth.

Optimization of matrix bandwidth does not significantly increase the time of model generation. On the other hand, the case of sparse solver affects the solving speed of the system of equations and influences the necessary size of the required memory. Matrix bandwidth optimization has a more significant effect on the solution of complex constructions.

Quadratic elements form significantly larger systems of equations with a considerably larger matrix bandwidth than linear elements. Linear element types (tetrahedrons and hexahedrons) have comparable the equations system size and solution requirements. Differences between tetrahedrons and hexahedrons occur during the meshing process of a complex model. Complex models usually appear as an unsolvable problem for the hexahedron mesh generation algorithm. The algorithm is not able to generate a mesh, and the process ends with an error statement. The popular alternative in these cases is the generation of tetrahedron mesh using Delaunay triangularization, which is known as a very reliable algorithm and fails very rarely.

Sparse solver is a very effective tool for solving systems of equations. The estimation of computational time is based on a large number of equations. The advantage of the sparse solver is usability to solve approximately 15 - 20 million equations for inappropriately conditioned systems with a difference on the matrix main diagonal more than $10^6$.

A multigrid solver is very effective for tetrahedron mesh. A partial decrease of efficiency occurs with quadratic elements and a considerable difference in values on the matrix main diagonal. The multigrid solver converged to a maximum of 40 iterations.
The conjugate gradient method is an iteration-based solver with a default number of iterations 1000. In analysis, we increased the number of iterations to the maximal available value of 2000. Despite the double increase of iterations, some examples did not converge.

Iterative solver, which is acknowledged as a special solver for quadratic 3D elements, is a very effective tool for quadratic elements. In the case of linear elements, its uselessness leads to replacement by a sparse solver. The solver also contributes to the successful solution of very poorly conditioned systems, such as contact with stiffness matrix stabilization.

The choice of mesh generation method and subsequent solution solver plays a significant role in solving quasi-linear and nonlinear problems where the solution is typical of a large number of steps [4 - 6].

5. Determining the correct element size

In large and complex models, the significant observed aspects are speed of model generation and obtaining results. During the model generation process, the identification of possible errors in the model is the most recognized parameter [7 - 9]. The practice demonstrated that the combination of linear tetrahedron elements and sparse solver is the most suitable alternative. As we have shown in previous lines, the generation of linear tetrahedron elements is the fastest choice. In addition, the fastest opportunity to obtain the solution is the sparse solver. However, a singular stiffness matrix characteristic of zero on the matrix main diagonal terminate the solver operation. On the other hand, iterative solutions usually continue the calculation until the maximum number of iterations is exceeded.

After verification of model correctness and functionality, the user may modify the mesh to refine the calculated stress values. The estimation of solution error determines mathematical methods, but their complexity limits their implementation in FEM programs.

We will show a simple procedure to determine the suitable size of linear tetrahedron elements. The modification of mesh density depends on the monitoring of the stress distribution in the finite element mesh. During the subsequent mesh refinement, we customized the local mesh dimension or division of lines and areas.

In the first step, we proposed a calculation model which necessarily met the following requirements:

- Division of material thickness into four elements. It is the minimum number of elements in a location of bending loading.

- At least 20 elements per circle circumference for sufficiently accurate geometry approximation.

In general, the mesh density for the whole body was 0.25. Exception presented the cylindrical surface of the hole with a mesh density of 0.1. In the case of linear tetrahedron elements, the program offers the automatic increase of mesh density. Definition of results displaying allows modification of stress legend into the form within which one colour corresponds to 10% of the maximum value of unsmoothed equivalent von Misses stress. The colours on both ends of the spectrum presented the excess of minimum or maximum values in the displayed range. Figure 3 illustrate the stress distribution around the curvature. The results showed that colour change between elements is more than one step on the legend. It means the difference is more than 10 %. Therefore, we could state that mesh density is insufficient. The maximum value of smoothed equivalent von Misses stress (in this case stress concentration) was 2.677.
Figure 3. Distribution of unsmoothed equivalent von Mises stress [Pa]. Element size in place of hole is 0.1.

In the second step, we increased the mesh density in all locations with stress concentration. The modified value of a mesh density was 0.05. Figure 4 presents updated results. Repeated comparison of colour change between adjacent elements showed that mesh density was insufficient. The stress concentration value was 2.847.

Figure 4. Distribution of unsmoothed equivalent von Mises stress [Pa]. Element size in place of hole is 0.05.
In the third step, we increased the mesh density near the curvature. The element size was 0.025. The results in Figure 5 show that the colour change of spectrum between the adjacent elements progresses by only one element. Therefore the mesh density was sufficient. The value of the stress concentration was 2.983. The solution error was estimated at 0.005%. The mesh consisted of 85 131 elements and 16 424 nodes. The total time needed to model generation and problem solution was nine seconds. The resulting distribution of smoothed von Misses stresses is in Figure 6.

Figure 5. Distribution of unsmoothed equivalent von Mises stress [Pa]. Element size in place of hole is 0.025.

Figure 6. Resulting distribution of stress concentration coefficient of linear tetrahedron elements with an appropriately selected mesh density.
6. Conclusion

The presented results show the justification of linear tetrahedron elements in FEA for solving the stress state. Although this type of element is the least accurate, the correctly adjusted mesh density allows obtaining sufficiently accurate results of the stress state with minimal RAM requirements and time consumption. The linear tetrahedron elements with a sparse solver are the most suitable alternative for testing the created models. In contrast to quadratic elements with a 3D iterative solver, verification of model correctness or finding an error in the model is several times faster. The presented procedure for determining the accuracy of the mesh density for linear tetrahedron elements allows obtaining results with a guaranteed error of less than 5%, which is sufficient for most cases of stress state analysis.

References

[1] Adina 2021 Theory and modeling guide volume 1 Adina solids and structures
[2] Adina 2021 Adina user interface. Command reference manual volume 1 solid and structures
[3] Bathe K J 2016 Finite element procedures (Watertown: K. J. Bathe)
[4] Sága M and Jakubovičová L 2014 Computational analysis of contact stress distribution in the case of mutual sweeping of roller bearing rings Poland Scientific journal of silesian university of technology-series transport 84 113
[5] Rybansky D, Sotola M, Marsalek P, Poruba Z, Fusek M 2021 Study of Optimal Cam Design of Dual-Axle Spring-Loaded Camming Device Switzerland Materials 14 1940
[6] Halama R, Macura P, Pecenka L, Fojtik F, Sofer, M 2011 Experimental Analysis of Residual Stresses in Backup Roll and FE solution, Experimentalni analyza napeti - experimental stress analysis, Czech Republic 85 – 90
[7] Flizikowski J and Macko M 2004 Competitive design of shredder for plastic in recycling. Ed. By Horvath I and Xiouchakis P. Conference: 5th Int. Symp. on Tools and Meth-ods of Competitive Engineering (Lausanne, Switzerland) Tools and methods of competitive engineering vols 1 and 2 1147-1148
[8] Krawiec A and Marlewski 2016 Profile design of noncircular belt pulleys. Journal of Theoretical and Applied Mechanics 54 (2) 561–570
[9] Dudziak M, Domek G, Kołodziej A and Talaśka K 2014 Contact Problems Between the Hub and the Shaft with a Three-angular Shape of Cross-section for Different Angular Positions Procedia Engineering – 2014 96 50-58

Acknowledgments

This work is supported by project KEGA 054ŽU-4/2021.