The effect of mesh density and element type on the analysis of 3D objects

Michala Weissova¹*, Vincent Kvocak², Daniel Dubecky³, and Patricia Vanova³

¹Technical University of Kosice, Faculty of Civil Engineering, Institute of Structural Engineering and Transportation Structures, 040 01 Kosice, Slovakia
²Technical University of Kosice, Faculty of Civil Engineering, Institute of Structural Engineering and Transportation Structures, Department of Load Bearing Structures and Intelligent Adaptive Structures, Vysokoskolska 4, 040 01 Kosice, Slovakia
³Technical University of Kosice, Faculty of Civil Engineering, Center for Research and Innovation in Construction, Park Komenskeho 10A, 042 00 Kosice, Slovakia

Abstract. This paper focuses on the usage of mesh density and element type in the analysis of 3D objects, which are crucial components of 3D object Finite Element Method (FEM) analysis. The document is based on theoretical principles and will be supported by practical numerical examples using the FEM software Abaqus. In order to conduct an accurate FEM analysis, it is vital to employ an appropriate mesh that is easily convertible for calculations and free from imperfections. The suitability of the mesh depends on two key parameters: density, which is determined by the material being used, and element type, such as tetrahedrons, hexahedrons, polyhedrons, and others. Different types of elements and mesh densities in FEM software often involve multiple settings that primarily impact the accuracy of the model, as well as the calculation time and convergence quality. This article is divided into three sections: the first part focuses on the theoretical foundations, the second part presents a practical example, and the third part offers conclusions and comparisons.

1 Introduction

Preparing the 3D mesh is a crucial initial step in obtaining accurate results from a model. This paper will discuss the various types of finite elements and their proper configuration, including density and size, in static models. The correct adjustment of finite elements significantly affects the calculation time. The presented knowledge will be illustrated through a selected example where the analysis will be conducted using Abaqus software. The fundamental types of 3D finite elements include tetrahedrons, hexahedrons, and triangular prisms, which can take on linear, quadratic, or cubic forms, among others. (Fig. 1) [1,2]

* Corresponding author: michala.weissova@tuke.sk

© The Authors, published by EDP Sciences. This is an open access article distributed under the terms of the Creative Commons Attribution License 4.0 (https://creativecommons.org/licenses/by/4.0/).
Apart from their size, finite elements also possess a density that is influenced by the material employed in the model. The size of the finite elements within the model is determined by factors such as the material density, the size and geometry of the problem being solved, and the stress-strain diagram. These aspects are interconnected and directly proportional. By maintaining the correct balance among these parameters, the calculation time can be reduced. [1,3]

2 FEM model

To showcase previous knowledge and experience, a simple example of a tensile test model has been chosen to determine the strength of the steel and generate the stress-strain diagram. (Fig. 2)

![Fig. 1. Types of 3D elements in the mesh.](image)

![Fig. 2. The results of the tensile test.](image)

The material properties, geometry, and loading conditions of the sample used in the model correspond to the results obtained from an experiment conducted at the Research and Innovation Centre for Construction. The experiment yielded the stress-strain diagram and material characteristics of the steel, which are represented by the model in the software. (Fig. 3).
Apart from their size, finite elements also possess a density that is influenced by the material employed in the model. The size of the finite elements within the model is determined by factors such as the material density, the size and geometry of the problem being solved, and the stress-strain diagram. These aspects are interconnected and directly proportional. By maintaining the correct balance among these parameters, the calculation time can be reduced.

To showcase previous knowledge and experience, a simple example of a tensile test model has been chosen to determine the strength of the steel and generate the stress-strain diagram. (Fig. 2)

The material properties, geometry, and loading conditions of the sample used in the model correspond to the results obtained from an experiment conducted at the Research and Innovation Centre for Construction. The experiment yielded the stress-strain diagram and material characteristics of the steel, which are represented by the model in the software. (Fig. 3)

Based on the previous figure, it is evident that the FEM model is accurate, allowing us to shift our focus towards assessing the mesh quality and the duration of the model calculation.

3 3D Mesh

3.1 Types of elements in the 3D mesh

The choice of mesh type and size significantly impacts the calculation process and, consequently, the obtained results. In the Abaqus software, the element shapes correspond appropriately to the overall model geometry. When employing the finite element method, calculations tend to converge more effectively when simpler element shapes are selected, although this may not always be feasible. Hexahedrons are elements that facilitate faster convergence and yield more accurate results. On the other hand, tetrahedrons are better suited for capturing complex problem geometries, but their convergence can be affected by element distortion. This distortion arises from the non-perpendicular walls of tetrahedrons, making their geometry more intricate compared to that of hexahedrons. (Fig. 4)

Fig. 3. Stress-strain diagram from FEM analysis and laboratory experiment.

Fig. 4. The samples in FEM models.
Both samples share the same mesh density and other conditions. The only distinction between the samples lies in the type of finite element used: hexahedrons on the right and tetrahedrons on the left in Figure 4.

- Sample with final elements in the form of hexahedrons:
  - Element count: 704
  - Calculation time: 8.8 sec
- Sample with final elements in the form of tetrahedrons:
  - Element count: 8450
  - Calculation time: 13711.5 sec

3.2 Element size in the 3D mesh

The final calculation time is affected by the size of the elements in the mesh. The overall convergence time of a calculation step is determined by the size of the smallest element within the entire model. Therefore, it is crucial to minimize significant differences in element sizes to ensure efficient convergence.

\[ \Delta t = L_e \left( \frac{\rho}{E} \right)^{1/2} \]  

The previous formula represents the relationship between the material characteristics, the element size and the calculation time where \( L_e \) is the size of the element, \( \rho \) is the material density, \( E \) is Young’s modulus and \( \Delta t \) is the increment stable time.

\[ L_e = \left( VOLUME \right)^{1/3} \]  

Based on these relationships, we can adjust the input values in Abaqus to shorten the calculation time and not affect the results. (Fig.5)

![Fig. 5. The comparison of the calculation time.](image)

3.3 The calculation time

The calculation time is impacted by various factors, including the complexity of the model geometry, the type of elements employed, their size, and their density. The material density incorporated in the model contributes to the overall calculation time and can be adjusted if necessary. Altering the material density may influence both the accuracy and the duration of the calculation. In models, the Mass Scaling function can be utilized to automatically increase the material density. The impact on the results will be minimal, contingent upon the chosen modeling approach.
The calculation time can be influenced by adjusting the utilization of computer logic processors and computer memory, with these settings being constrained by computer parameters.

In addition to modifying material density, it is also possible to adjust the calculation period within the range of 0.0 to 1.0. This adjustment affects both the total calculation time and the obtained results. Choosing a lower value from this interval reduces the convergence time for individual increments, leading to variations in results, especially within the plastic region of the graph. Figure 6 illustrates a comparison of certain model changes described above.

![Graph showing calculation time comparison](image)

**Fig. 6.** Results of tensile test from the different FEM models.

Figure 6 displays a graph depicting the relationship between displacement and force at a reference point chosen on the site of the tensile test sample during the actual experiment. The image presents a comparison of the test results obtained using different types of mesh and calculation periods. The graph showcases the results of four models, with each individual curve representing:

- TET TP-1.0 – tetrahedron elements, time period 1.0
- HEX TP-1.0 – hexahedron elements, time period 1.0
- TET TP-0.1 – tetrahedron elements, time period 0.1
- HEX TP-0.1 – hexahedron elements, time period 0.1

![Comparison of time in different types of samples](image)

**Fig. 7.** Comparison of time in different types of samples.
4 Conclusion

The accuracy and efficiency of the analytical model are influenced by the 3D mesh, as well as the proper adjustment of load conditions, material characteristics, and geometry. The software compiles stiffness matrices and equations based on the model's mesh, which are essential for the overall analysis.

This paper has demonstrated how mesh settings impact both the calculation time and the accuracy of the results. The analytical models clearly show that changing the element type and the calculation period leads to differences in results, with a significant impact on the total calculation time.

The conclusion drawn from these findings is that when selecting a 3D mesh, it is important to provide correct and appropriate inputs in order to strike a balance between the model's accuracy and the calculation time.

The presented paper was supported by the project: VEGA 1/0172/20 “Stress and deformation analysis of load-bearing components made out of steel, glass and composite materials” of the Scientific Grant Agency of the Ministry of Education, Science, Research and Sport of the Slovak Republic.

References