Published online on the page: https://jurnal.mifandimandiri.com/index.php/jpvi

J P V I

Jurnal Pendidikan Vokasi Indonesia

| ISSN (*Online*) 2962-4649 |

FEM Mesh Convergence of a Cantilever Beam Using COMSOL Multiphysics for Vocational Engineering Education

Berkadh Simanjuntak[1,*]

[1]*Politeknik Tanjungbalai, Tanjungbalai*, Indonesia

Article Information

Article History:
Submit: 10 Juli 2026
Revision: 15 Juli 2026
Accepted: 17 Juli 2026
Published: 30 Agustus 2026

Keywords

Finite Element Method; Mesh Convergence; Cantilever Beam; COMSOL Multiphysics; Vocational Engineering Education

Correspondence

E-mail: berkatjuntak@gmail.com*

A B S T R A C T

Finite element method (FEM) accuracy is influenced by mesh discretization, whereas excessively fine meshes can impose disproportionate computational costs. This study analyzes mesh convergence in a rectangular cantilever-beam model and identifies the mesh level that provides the best balance between accuracy and efficiency in COMSOL Multiphysics as an instructional case for vocational engineering education. A numerical experiment was performed using five physics-controlled mesh levels: Coarse, Normal, Fine, Finer, and Extra Fine. The three-dimensional domain was discretized using an unstructured free-tetrahedral mesh with quadratic displacement interpolation. The observed parameters were element count, degrees of freedom (DOF), maximum displacement, maximum stress, relative error between successive mesh levels, and computation time. Maximum displacement increased from 0.432 mm with the Coarse mesh to 0.470 mm with the Extra Fine mesh, while maximum stress increased from 112.45 to 132.05 MPa. At the Finer level, the relative errors in displacement and stress decreased to 0.64% and 1.26%, respectively, with a computation time of 8.6 min. Refinement to Extra Fine changed displacement by only 0.21% and stress by 0.48%, but increased computation time by 148.84%. Based on an error threshold below 2% and resource efficiency, the Finer mesh was selected as the optimum configuration. The results also demonstrate how convergence metrics and stress-contour comparisons can train students to distinguish visually plausible outputs from numerically verified solutions before FEM results are used for engineering evaluation.

1. Introduction

The finite element method (FEM) is a numerical approach widely used to solve engineering problems in which geometry, material properties, loading, or boundary conditions cannot be adequately represented by analytical solutions. In FEM, a continuous domain is subdivided into a finite number of elements, and the governing equations are solved at discrete points to approximate the system response. This capability has made FEM an important tool for structural analysis, heat transfer, fluid flow, electromagnetics, and various multiphysics problems (Bathe, 2014; Szabó & Babuška, 2021).

The reliability of FEM results depends not only on the accuracy of the physical model but also on the quality of mesh discretization. A mesh divides the geometry into elements over which approximation functions are applied. As the element size is reduced, the numerical solution should theoretically approach the continuous solution for a well-posed problem. However, changes in element shape, size, and distribution can affect the solution, particularly in regions with high response gradients. Mesh sensitivity analysis is therefore required to ensure that the principal outputs no longer depend materially on the mesh density (COMSOL AB, n.d.-a; He & Huang, 2021).

A mesh convergence study is conducted by solving the same model at several mesh-density levels and comparing the relevant scalar outputs. In structural analysis, displacement and stress are commonly used as indicators because they represent deformation and the intensity of the mechanical response. Their convergence behavior, however, is not necessarily identical. Displacement is generally smoother, whereas stress is derived from the displacement gradient and is therefore more sensitive to element size, element shape, stress concentrations, and local singularities. Consequently, local stress values may require greater mesh refinement than displacement values to achieve numerical stability (COMSOL AB, n.d.-b; Zhao & Ji, 2019).

Mesh refinement increases the number of elements and degrees of freedom (DOF), but it also increases memory requirements and solution time. An excessively coarse mesh may produce an unstable response, whereas an excessively fine mesh can impose a substantial computational burden without a meaningful gain in accuracy. Selecting an optimum mesh is therefore a trade-off between numerical precision and computational cost. Gukop et al. (2021) showed that mesh density affects both stress results and computation time in a support bracket. Kim et al. (2023) also emphasized the need for guidance in selecting element size so that structural-strength predictions are not obtained through excessive refinement. Parvez et al. (2024) similarly argued that improvements in accuracy should be evaluated together with the corresponding increase in computational cost.

Applied studies have demonstrated that mesh convergence should be assessed using output parameters that are directly related to the objective of the analysis. Patil and Jeyakarthikeyan (2018) used discretization error to evaluate convergence in a clutch-disc hub. Hotma et al. (2024) compared deformation and von Mises stress in an induction-motor shaft using several mesh sizes. Cadet and Paredes (2024) showed that mesh density and element type can be optimized to obtain accurate results efficiently. In other fields, Lee et al. (2024) and Rijal et al. (2024) also treated convergence testing as part of numerical-model verification before further interpretation of the simulation results.

In vocational engineering education, mesh convergence should be taught not merely as a software-setting exercise, but as part of numerical verification and professional engineering judgment. Simulation-based and project-based instruction can strengthen students' ability to connect mechanics theory, CAE procedures, and practical design decisions, while also requiring them to critically evaluate software outputs rather than accept visually convincing contours at face value (Guo et al., 2020; Fernandes et al., 2020; Kondo et al., 2023). A mesh that is too coarse can underestimate deformation or localized stress, potentially leading to unsafe dimensions, inappropriate material selection, or premature design approval in industrial practice. Accordingly, training students to compare mesh levels, quantify relative error, interpret stress contours, and justify a computationally efficient discretization is directly relevant to vocational competencies in mechanical design, manufacturing, maintenance, and structural analysis.

The preliminary data in this study show substantial changes in maximum displacement and maximum stress during the transition from a coarse to an intermediate mesh, followed by progressively smaller changes as the mesh is refined. At the same time, computation time increases sharply. This condition raises the question of which mesh level is sufficiently stable for use without adding an inefficient computational burden. The assessment cannot be based solely on the finest mesh because that configuration does not necessarily provide the best ratio of numerical benefit to computational cost.

This study analyzes the effects of mesh-density variation on the number of elements, DOF, maximum displacement, maximum stress, relative error, and computation time in FEM simulations using COMSOL Multiphysics. It also determines the optimum mesh level based on result stability and computational efficiency. The findings are expected to provide a technical basis for selecting a defensible discretization before the model is used for further evaluation or design development.

2. Research Methods

2.1. Research Design

This study employed a comparative numerical-experiment approach. A single FEM model was solved repeatedly using five mesh-density levels, while the geometry, material properties, boundary conditions, loading, study type, and solver settings were kept constant. The independent variable was mesh density, while the response variables comprised the number of elements, DOF, maximum displacement, maximum stress, and computation time.

Numerical verification was performed through a mesh convergence study. This procedure was not intended to directly validate the model against physical conditions; rather, it assessed whether the solution was sufficiently independent of the discretization. Physical validation would still require comparison with experimental data, analytical solutions, or appropriate reference data. Distinguishing verification from validation is important to ensure that the conclusions do not exceed the available numerical evidence (Szabó & Babuška, 2021).

2.2. Model and Simulation Configuration

The numerical model represented a rectangular cantilever beam with a length of 150 mm, a width of 30 mm, and a thickness of 20 mm. The beam was made of structural steel modeled as an isotropic linear-elastic material with an elastic modulus of 200 GPa, a Poisson's ratio of 0.30, and a density of 7,850 kg/m^3. One end face of the beam was assigned a fixed constraint, while an equivalent distributed load of 1.75 kN was applied vertically downward to the opposite end face. All geometric parameters, material properties, boundary conditions, and loading conditions were held constant for every mesh variation so that changes in the simulation outputs reflected only the effect of element discretization.

The simulations were performed in COMSOL Multiphysics version 6.2 using the Solid Mechanics physics interface and a Stationary study. The three-dimensional solid domain was discretized using the default physics-controlled meshing sequence, which generated an unstructured free-tetrahedral volume mesh; the visible surface facets were therefore triangular. Quadratic shape functions were used for the displacement field, corresponding to second-order interpolation within the tetrahedral elements (COMSOL AB, n.d.-c). The resulting system of equations was solved using the PARDISO direct solver with a relative tolerance of 10^{-3}. Small-deformation and linear-elastic material assumptions were applied; therefore, geometric nonlinearity was disabled. Identical element family, interpolation order, and solver settings were used for every mesh level. The computations were conducted on a computer equipped with an Intel Core i7-12700 processor with 12 cores and 20 threads, 16 GB of RAM, and the 64-bit Windows 11 Pro operating system. The hardware configuration and solver settings are reported because both influence the computation time obtained for each mesh variation.

2.3. Mesh Variations and Observed Parameters

The discretization variations used five physics-controlled mesh levels in COMSOL Multiphysics: Coarse, Normal, Fine, Finer, and Extra Fine. Each level generated a different number and characteristic size of free-tetrahedral elements and therefore a different number of DOF. The tetrahedral element family and quadratic interpolation order were retained across all cases; only the predefined physics-controlled element-size level was changed. This sequence made it possible to observe the solution from coarse to very fine discretization without modifying the physical definition or numerical formulation of the model.

The observed parameters were selected to represent three principal aspects: model complexity, mechanical response, and computational cost. Model complexity was represented by the element count and DOF; mechanical response was represented by maximum displacement and maximum stress; and computational cost was represented by the solution time for each case. The parameters and their evaluation functions are presented in Table 1.

Table 1. Observed Parameters and Evaluation Criteria

Parameter	Unit	Evaluation Function
Element count	elements	Represents the level of geometric discretization
Degrees of freedom (DOF)	DOF	Represents the size of the numerical equation system
Maximum displacement	mm	Evaluates the stability of the deformation response
Maximum stress	MPa	Evaluates the stability of the stress response
Relative error	%	Compares the result with the preceding mesh level
Computation time	min	Evaluates numerical-solution efficiency

Source: Processed research data

2.4. Data Analysis

The relative error in maximum displacement and maximum stress was calculated by comparing the result at the current mesh level with the result at the preceding mesh level. The following equation was used:

$$E^i\,(\%) = |(X_i - X_i\text{-}1) / X_i| \times 100\% \qquad (1)$$

In Equation (1), E_i denotes the relative error at the i-th mesh level, X_i denotes the output value at the current mesh level, and X_(i-1) denotes the output value at the preceding mesh level. No relative error was calculated for the first mesh because no preceding result was available for comparison. The calculations were performed separately for maximum displacement and maximum stress.

The operational criteria for the optimum mesh were defined as follows: the relative errors in displacement and stress were below 2%, the output curves displayed a stable trend, and subsequent refinement did not provide a change in the results proportional to the increase in computational requirements. The 2% threshold was used as an internal criterion for this study rather than as a universal limit for all FEM models. The final selection was made by comparing the additional accuracy with the corresponding growth in element count, DOF, and computation time.

3. Results and Discussion

3.1. Changes in Model Complexity and Computation Time

Increasing mesh density consistently increased the element count, DOF, and computation time. The model-complexity data for each mesh level are presented in Table 2. The Coarse mesh generated 7,865 elements and 24,110 DOF with a solution time of 0.7 min. At the Extra Fine level, these values increased to 188,640 elements and 566,920 DOF, with a computation time of 21.4 min.

Table 2. Model Complexity and Computation Time for Each Mesh Variation

Mesh Variation	Element Count	DOF	Computation Time
Coarse	7,865	24,110	0.7 min
Normal	18,420	56,880	1.4 min
Fine	42,530	130,450	3.2 min
Finer	91,875	278,360	8.6 min
Extra Fine	188,640	566,920	21.4 min

Source: Processed research data

From Coarse to Extra Fine, the number of elements increased by nearly 24 times, the number of DOF increased by more than 23 times, and the computation time increased by approximately 30.6 times. This pattern indicates that the growth in computational demand is not a simple linear function of a single parameter; it is influenced by the equation-system size, matrix structure, memory requirements, and solver characteristics. This finding is consistent with the explanation by COMSOL AB (n.d.-a) that resource requirements are directly related to DOF growth and with the results of Gukop et al. (2021), who demonstrated a trade-off between mesh density and computational efficiency.

3.2. Convergence of Maximum Displacement

Maximum displacement increased from 0.432 mm with the Coarse mesh to 0.457 mm with the Normal mesh. The relative change of 5.47% indicates that the Coarse mesh had not yet produced a stable solution. When the mesh was refined to Fine, maximum displacement increased to 0.466 mm with an error of 1.93%. This result approached the operational threshold of the study, although stronger stability was observed at the Finer level.

Table 3. Mechanical Response and Relative Error for Each Mesh Variation

Mesh Variation	Max. Displacement (mm)	Displacement Error (%)	Max. Stress (MPa)	Stress Error (%)
Coarse	0.432	-	112.45	-
Normal	0.457	5.47	124.88	9.95
Fine	0.466	1.93	129.76	3.76
Finer	0.469	0.64	131.42	1.26
Extra Fine	0.470	0.21	132.05	0.48

Source: Processed research data

At the Finer level, maximum displacement reached 0.469 mm, and the change from the Fine mesh was only 0.64%. Further refinement to Extra Fine produced a displacement of 0.470 mm, corresponding to a change of 0.21%. The absolute difference between Finer and Extra Fine was only 0.001 mm. The displacement curve had therefore entered a plateau region, indicating that the deformation output was becoming increasingly independent of the discretization.

3.3. Convergence of Maximum Stress

Maximum stress showed greater sensitivity than displacement. The stress increased from 112.45 MPa with the Coarse mesh to 124.88 MPa with the Normal mesh, corresponding to a relative error of 9.95%. With the Fine mesh, the stress increased to 129.76 MPa and the error decreased to 3.76%, but this value remained above the operational criterion of 2%. The Finer mesh produced a maximum stress of 131.42 MPa with an error of 1.26%, whereas the Extra Fine mesh produced 132.05 MPa with an error of 0.48%.

The difference between the convergence behavior of displacement and stress is consistent with the characteristics of FEM. Stress is derived from the gradient of the displacement field and is therefore more sensitive to changes in element size and local conditions. Zhao and Ji (2019) showed that stress-convergence behavior can be affected by element type and integration, while Cadet and Paredes (2024) found that mesh density and element type should be selected according to the output being evaluated. Consequently, using displacement alone as the convergence indicator may lead to a premature conclusion.

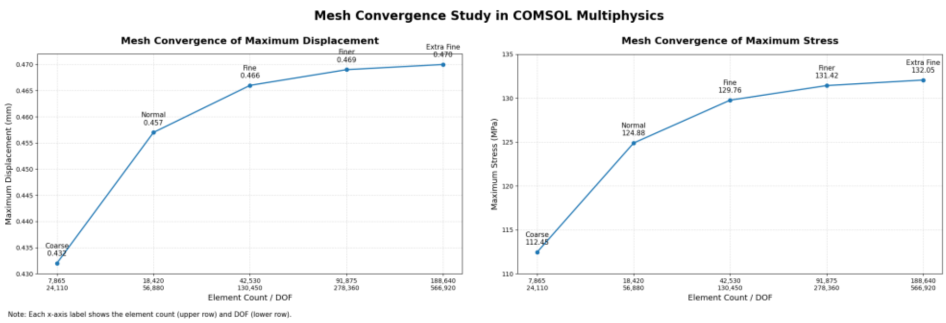

Figure 1. Convergence Curves of Maximum Displacement and Maximum Stress

Figure 1 shows that the largest changes occurred between the Coarse and Normal levels, after which the slopes of the curves decreased through the Fine, Finer, and Extra Fine levels. This asymptotic pattern provides visual evidence that the solution was approaching a stable condition. Numerical stability must nevertheless be interpreted together with relative error and computational cost because an apparently flat curve does not, by itself, identify the most efficient configuration.

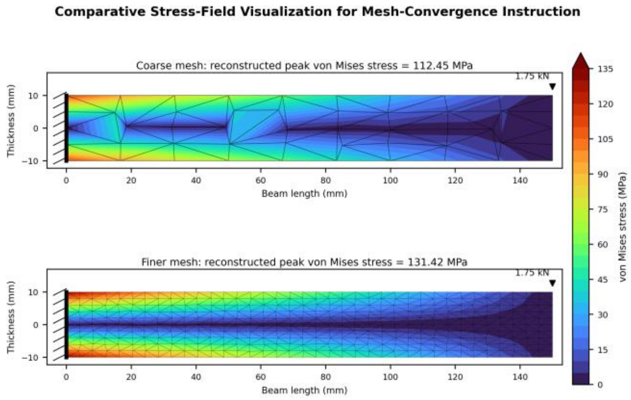

Figure 2. Comparative Reconstructed von Mises Stress Contours for the Coarse and Finer Meshes

Source: Pedagogical reconstruction based on processed research data

Figure 2 provides a side-by-side educational visualization of the stress-field representation at the Coarse and Finer levels. Both contours locate the highest stress near the fixed end, consistent with cantilever bending, but the Coarse representation displays broader, less resolved stress bands and the lower reported peak of 112.45 MPa. The Finer representation shows a smoother gradient and the higher localized peak of 131.42 MPa. This comparison helps learners recognize that a visually plausible contour can remain mesh-dependent. Because the original native COMSOL contour exports were not embedded in the source manuscript, Figure 2 is a pedagogical reconstruction based on the reported maximum-stress values and the expected bending-stress pattern; native COMSOL result plots should replace it in final simulation documentation when available.

3.4. Selection of the Optimum Mesh

The Finer mesh satisfied all operational criteria of the study: a displacement error of 0.64%, a stress error of 1.26%, and stable output trends. Although the Extra Fine mesh produced lower errors, this additional benefit must be compared with the increase in computational requirements. A direct comparison between the Finer and Extra Fine meshes is presented in Table 4.

Table 4. Efficiency Comparison between the Finer and Extra Fine Meshes

Indicator	Finer	Extra Fine	Change
Element count	91,875	188,640	+105.33%
DOF	278,360	566,920	+103.66%
Computation time	8.6 min	21.4 min	+148.84%
Maximum displacement	0.469 mm	0.470 mm	+0.21%
Maximum stress	131.42 MPa	132.05 MPa	+0.48%

4. Conclusion

Mesh-density variation directly affected the stability of the FEM outputs and the computational requirements. Refinement from Coarse to Extra Fine increased maximum displacement from 0.432 to 0.470 mm and maximum stress from 112.45 to 132.05 MPa. Relative error decreased progressively with mesh refinement, whereas the number of elements, DOF, and computation time increased sharply. Displacement reached an error below 2% with the Fine mesh, but stress did not meet this criterion until the Finer mesh was used. The Finer mesh was identified as the optimum configuration because it produced a maximum displacement of 0.469 mm, a maximum stress of 131.42 MPa, a displacement error of 0.64%, a stress error of 1.26%, and a computation time of 8.6 min. Refinement to Extra Fine increased computation time by 148.84% while changing the outputs by less than 0.5%, and therefore did not provide a proportionate benefit. These results confirm that mesh selection should be based on evidence of convergence and computational efficiency rather than solely on the highest available density. As a vocational learning case, the cantilever-beam exercise provides a structured way to teach students to integrate element-type awareness, contour interpretation, relative-error calculation, and computing-resource considerations before making engineering judgments. Future work should validate the numerical findings against experimental data or an appropriate reference solution.

References

Bathe, K. J. (2014). *Finite element procedures (2nd ed.)*. Klaus-Jürgen Bathe.

Cadet, G., & Paredes, M. (2024). Convergence analysis and mesh optimization of finite element analysis related to helical springs. *Mechanics & Industry, 25*, 22. https://doi.org/10.1051/meca/2024018

COMSOL AB. (n.d.-a). *Finite element mesh refinement definition and techniques*. https://www.comsol.com/multiphysics/mesh-refinement

COMSOL AB. (n.d.-b). *Performing a mesh refinement study*. https://www.comsol.com/support/knowledgebase/1261

COMSOL AB. (n.d.-c). *The mesh node. COMSOL Multiphysics Reference Manual*. https://doc.comsol.com/6.3/doc/com.comsol.help.comsol/comsol_ref_mesh.24.11.html

Fernandes, F. A. O., Marques, C., Castelan, J., Fritzen, D., & Alves de Sousa, R. J. (2020). Learning processes in mechanics of structures: Allying analytical and numerical approaches. *Education Sciences, 10*(4), 114. https://doi.org/10.3390/educsci10040114

Gukop, N. S., Kamtu, P. M., Lengs, B. D., Babawuya, A., & Adegoke, A. (2021). Effect of mesh density on finite element analysis simulation of a support bracket. *FUOYE Journal of Engineering and Technology, 6*(3), 34–38. https://doi.org/10.46792/fuoyejet.v6i3.632

Guo, Y., Yang, L., Chen, X., & Yang, L. (2020). An engineering-problem-based short experiment project on finite element method for undergraduate students. *Education Sciences, 10*(2), 45. https://doi.org/10.3390/educsci10020045

He, Y., & Huang, W. (2021). *Mesh sensitivity analysis for finite element solution of linear elliptic partial differential equations*. arXiv. https://doi.org/10.48550/arXiv.2111.10935

Hotma, L., Majid, N. C., Marsalyna, Hendrawan, J., Kinasih, N. A., Febriansyah, D. J., Novariawan, B., Alfin, M. R., Putri, I. S., Ghufron, H., Afif, M. T., Fadjrin, B. N., & Purnomo, E. D. (2024). Mesh convergence test by using finite element analysis (FEA) in shaft loading of 2 kW induction motor. *Rekayasa Mesin, 15*(3), 1271–1280. https://doi.org/10.21776/jrm.v15i3.1516

Kim, D. K., Ban, I., Poh, B. Y., & Shin, S.-C. (2023). A useful guide of effective mesh-size decision in predicting the ultimate strength of flat- and curved plates in compression. *Journal of Ocean Engineering and Science, 8*(4), 401–417. https://doi.org/10.1016/j.joes.2022.02.014

Kondo, A., Hayashi, H., & Toyoshi, T. (2023). Project-based learning of mechanical design utilizing CAE structural analyses. *Education Sciences, 13*(7), 687. https://doi.org/10.3390/educsci13070687

Lee, J.-H., Seo, J.-H., Park, S.-W., Kim, W.-G., Jung, T.-G., & Lee, S.-J. (2024). A finite element analysis study of edentulous model with complete denture to simulate masticatory movement. *Bioengineering, 11*(4), 336. https://doi.org/10.3390/bioengineering11040336

Liu, J., Wu, L., Yin, K., Song, C., Bian, X., & Li, S. (2022). Methods for solving finite element mesh-dependency problems in geotechnical engineering—A review. *Sustainability, 14*(5), 2982. https://doi.org/10.3390/su14052982

Parvez, N., Amjad, S. N., Dey, M. K., & Picu, C. R. (2024). Methodological aspects and mesh convergence in numerical analysis of athermal fiber network material deformation. *Fibers, 12*(1), 9. https://doi.org/10.3390/fib12010009

Patil, H., & Jeyakarthikeyan, P. V. (2018). Mesh convergence study and estimation of discretization error of hub in clutch disc with integration of ANSYS. *IOP Conference Series: Materials Science and Engineering, 402*(1), 012065. https://doi.org/10.1088/1757-899X/402/1/012065

Rijal, M., Amoateng-Mensah, D., & Sundaresan, M. J. (2024). Finite element simulation of acoustic emissions from different failure mechanisms in composite materials. *Materials, 17*(24), 6085. https://doi.org/10.3390/ma17246085

Szabó, B., & Babuška, I. (2021). *Finite element analysis: Method, verification and validation (2nd ed.)*. Wiley. https://doi.org/10.1002/9781119426479

Zhao, W., & Ji, S. (2019). Mesh convergence behavior and the effect of element integration of a human head injury model. *Annals of Biomedical Engineering, 47*, 475–486. https://doi.org/10.1007/s10439-018-02159-z