El Cezeri
Journal of Science and Engineering
ISSN: 2148-3736

Review Article

Enhancing Transparency and Reproducibility in Finite Element Analysis through Comprehensive Reporting Parameters: A Review

Aun Haider[1a]
[1] Institute of Aeronautics and Avionics, Air University, Islamabad, Pakistan

aunbhutta@gmail.com

DOI : 10.31202/ecjse.1436203
Received: 13.02.2024 Accepted: 24.04.2024
How to cite this article:
Aun Haider, " Enhancing Transparency and Reproducibility in Finite Element Analysis through Comprehensive Reporting Parameters: A Review", El-Cezeri Journal of Science and Engineering, Vol: 11, Iss:3, (2024), pp.(212-222).
ORCID: [a]0009-0000-5279-2829

Abstract : This paper highlights the importance of thorough documentation in Finite Element Analysis (FEA) studies to ensure transparency and reproducibility of results. It points out the lack of standardized guidelines for reporting and communication in FEA, which can lead to confusion and hinder evaluation. The paper aims to address this gap by proposing key reporting parameters covering various aspects of FEA studies such as analysis description, model identification, solver settings, and validation techniques. It emphasizes the significance of sensitivity analysis, verification, and validation for establishing the reliability of FEA models. The abstract concludes by advocating for the sharing of FEA models to promote scrutiny and improvement, using a case study to demonstrate how the proposed reporting parameters can enhance the quality and credibility of simulation studies. Additionally, it suggests that as modeling techniques evolve, the reporting parameters should also adapt accordingly, incorporating any supplementary factors that affect the accuracy of FEA models.

Keywords : Finite Elements Analysis, Meshing, Pre-processing, Post Processing, Reporting Parameters, Solver Setting, Sensitivity Analysis, Validation, Verification

1 Introduction

In the past few decades, Finite Element Analysis (FEA) has played a pivotal role in structural analysis, offering researchers and engineers invaluable insights into structural responses under varying environmental conditions [1]. The continuous evolution of FE codes has significantly enhanced our ability to comprehend complex structural behaviors.

FEA, as a numerical technique, provides an approximate solution to partial differential equations [2]. The process involves discretizing the problem domain to create a mesh model, with elements connecting at nodes. Equations are then solved at these nodes, and discrete nodal values are interpolated across the domain to generate field values. Standard FEA procedure encompasses geometry development, assignment of material properties, mesh discretization, and application of boundary conditions and loads, culminating in the computation of desired results [3].

It is important to note that, strictly speaking, all FE models are inherently imperfect, yet they remain useful tools for representing physical phenomena [4]. The accuracy of these results hinges on effective error control within the FEA process. Modeling errors may arise from assumptions regarding geometry, material models, loads, and boundary conditions, while inadequacies in the mesh can lead to discretization errors. Additionally, numerical schemes themselves may introduce errors [5]. Assessment of errors arising from inherent idealizations during Finite Element Analysis (FEA) necessitates meticulous documentation, serving as a cornerstone for establishing confidence in the accuracy and reliability of FE results [6]. Detailed documentation not only ensures repeatability and reproducibility of FE results but also underscores the importance of investing in both model improvement and comprehensive documentation [7].

2 Literature Review

The availability of extensive guidelines on model development, verification, and validation from organizations such as the American Society of Mechanical Engineers (ASME) and the National Agency for Finite Element Methods and Standards (NAFEMS) has significantly contributed to the advancement of Finite Element Analysis (FEA) in various engineering fields [8]. However, a notable gap exists in these standards, as they do not explicitly address critical aspects of reporting and communication of FE studies, resulting in a discrepancy between the widespread application of this tool and the quality of reports [9].

In FEA, intricacies of model definition and development are intrinsically linked with analysis settings and capabilities of FE code [10]. A diverse array of simulation software packages available for structural analysis, each with distinct pre-, solution, and post-processing algorithms, further complicates matters. Differences in syntax among these FE codes, if not adequately addressed in FE reports, can impede the reproducibility of results across different software platforms [7].

The lack of a standardized documentation basis contributes to a perceived lack of transparency in reported FE studies, hindering the essential processes of verification, validation, and accreditation. Additionally, a prevailing issue is FE analysts' potential unawareness of the limitations inherent in the chosen model and simulation platform, leaving reviewers and users uninformed [11].

Upon reviewing published FE results, it becomes evident that while some simulation parameters are generally reported, the information provided may be insufficient for result reproduction due to a lack of detail about the simulation environment [12]. This study aims not to prescribe best practices in FEA but to introduce reporting parameters that comprehensively assess the overall quality and scientific rigor of simulation studies. By presenting comprehensive information about the simulation environment, solution verification and validation (V & V), and model availability, this paper illustrates these guiding principles [13].

A review of published FE results reveals that few simulation parameters like type of analysis, selection of finite elements, boundary conditions, loads, etc. have generally been reported [14]. In some cases, authors have only indicated the conformance of models with standard modeling practices without commenting on the variability or sensitivity of results. At best, the reported set of parameters has traditionally been minimal to fulfill publication criteria. Unfortunately, sometimes published FE results cannot be reproduced due to insufficient information about the simulation environment [15]. The guidelines presented herein serve as a valuable resource for journal editors and reviewers in the evaluation of manuscripts, helping to identify potential knowledge gaps between theoretical concepts and practical applications of FEA.

While intended for engineers and scientists in academia, industry, and government agencies involved in the preparation, dissemination, and evaluation of simulation studies, readers need to acknowledge that not all considerations may be reported in every FEA study [16]. While justifications for omitted steps must be documented, adherence to these guidelines enhances confidence in disseminating FE results. Nevertheless, these guidelines do not exclude the incorporation of additional details driven by the fidelity of the simulation model.

3 Reporting Parameters

3.1 Analysis Description

The preamble of a reporting document ideally should offer a comprehensive description of the FE study under consideration.

3.2 Study Objective

FEA report should elaborate on a research question that the study seeks to answer. In cases where the primary objective is the structural qualification of a component, as opposed to exploring design spaces, it is advisable to include acceptance criteria as part of the report [17].

3.3 Motivation

The motivation for employing Finite Element Analysis (FEA) should be clearly stated, emphasizing its advantages in comparison to the capabilities or limitations of existing analytical, empirical, or experimental methods.

3.4 Scope

The report should provide a detailed scope, outlining how FEA of the problem domain will be conducted to generate solution space. This includes a clear description of the methodology, modeling approach, and key steps involved in the FEA process.

3.5 Mathematical Model

It is advisable to elaborate on the constitutive representation of field variables, presenting a mathematical relationship between required inputs and desired outputs within the FEA framework [18].

3.6 Type

The type of analysis conducted, whether it be static, dynamic, modal, buckling, thermal analysis, or any other specific type, should be identified in the report.

3.7 Time Scale

Specify the simulation's time scale in the report to discern the nature of the response, distinguishing between steady-state and transient responses.

3.8 System of Units

System of units for FE simulation, such as SI System, British Engineering System, MKS System, etc., should be explicitly stated. It is worth noting that certain Finite Element (FE) codes may not adhere to any specific unit system [11], placing responsibility for unit consistency squarely on the analyst in such cases.

3.9 Analysis Hardware and Software

Reporting solution time with the used computer system is essential for providing an assessment of computation costs [19]. Therefore, hardware and software employed for Finite Element Analysis (FEA) should be thoroughly documented. While in-house proprietary codes may be utilized, FEA is generally conducted using commercial software packages encompassing geometry modelers, pre-processors, solvers, and post-processors. Although vendors typically offer sufficient documentation on code verification, it is crucial to include comments on the suitability of these software packages for the intended analysis. Despite claims of upward and downward compatibility by code developers among various versions, it is advisable to report specific software versions used [20].

3.10 Related Publications

If there is published work on similar analyses, it is recommended to refer to these publications, as they provide a valuable basis for the verification and validation (V & V) of the current analysis [21].

3.11 Highlight

A detailed exposition of the analysis's distinctive features, emphasizing any innovative elements that differentiate it from existing approaches, will augment the overall value of the FEA report.

4 Model Identification

4.1 Model Name

Assigning an illustrative name to the FE model, differentiating it from similar models, is recommended. Furthermore, enhancing the model's discoverability in research repositories can be achieved by incorporating relevant keywords into its description.

4.2 Version

During most investigations, the FE model typically undergoes refinements upon the initial receipt of results, ultimately leading to the adoption of a refined or improved model for reporting FE outcomes. If the developmental history of the model has the potential to contribute valuable insights to the existing knowledge bank, it is advisable to incorporate this information in the report.

4.3 Region of Interest

During structural design, identifying areas or regions of interest in the model is essential. Typically, geometric or material transitions, supports, load application regions and fastening mechanisms are major concerns that warrant careful attention.

4.4 Related Models

When related models exist, it's important to present the strengths and limitations of these complementing or competing models to adequately justify the employed model.

4.5 Utility

Delineating the primary (or secondary, if any) utility of the model in relevant engineering domains is essential.

4.6 Case Study

The aircraft wing comprises structural members such as ribs, spars, stringers, and skin [22]. It is designed to accommodate external stores at designated stations. Figure 1 depicts the internal structural members of the aircraft wing. The analysis of the wing rib located at wing stations 2 and 6 is necessary to determine the maximum stress under applied aerodynamic and inertial loads within the flight envelope of the aircraft.

The aircraft wing, being a statically indeterminate structure, exhibits support reactions on the rib (local model) isolated from the wing (global model) that depend on the stiffness of the attached structure. Analytical or empirical analyses of the rib isolated from the wing do not fully account for the stiffness of the global model [23]. Experimental analyses based on strain gauge techniques can provide accurate point values, but interpolating these values is necessary to generate field values for the wing rib. Moreover, the cost and safety implications of conducting actual ground or flight tests can be prohibitive. Therefore, Finite Element (FE) analysis of the wing rib under design load emerges as the most suitable method.

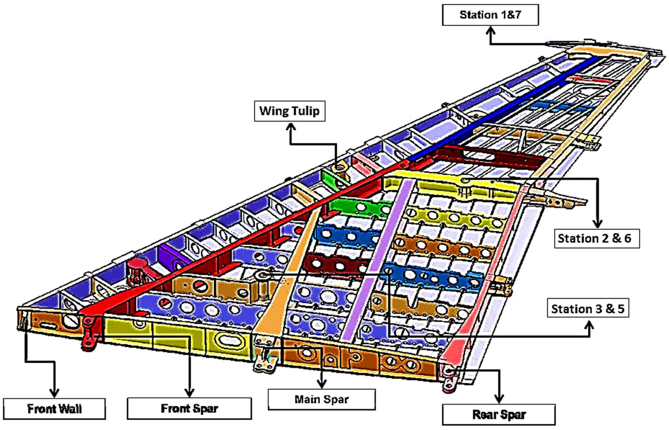

Figure 1: Wing Internal Structure

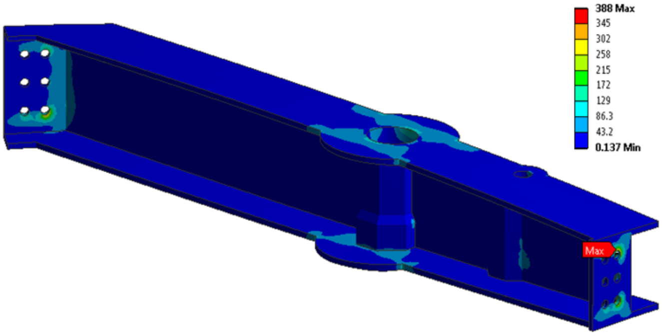

Figure 2: Wing Rib with Fixed Support

The CAD model of the wing rib is developed in Ansys Design Modeler® from manufacturing drawings. Appropriate material properties given in aircraft technical publications are assigned to the model. The stiffness of the attached wing structure is represented by elastic support boundary conditions applied at attachment bolt holes. Static structural analysis of the wing rib is carried out at design load.

Analysis of structural members isolated from the global model is generally carried out with fixed support as a boundary condition [24]. This is a conservative approximation that under predicts the performance of local models. As an aircraft wing is a statically indeterminate structure, reaction forces and moments at supports (constraints) depend upon the stiffness characteristics of the wing itself. This necessitates that the stiffness of the global wing model must be represented in a solution of the local rib model.

The application of elastic support introduces a finite stiffness of the global model in a solution of the isolated local model. Elastic support is the most appropriate boundary condition for FE analysis of local models. With elastic supports, it is predicted that the value of maximum stress will be reduced. Therefore, the maximum load-carrying capacity of the wing station can be predicted accurately.

Region of interest are bolt holes used for attachment of rib with wing structure. The SI system of units is used for analysis. FE analysis is performed in Ansys ® Software version 14.5 installed on a workstation with a core i7 Intel processor and 16 GB RAM.

A similar FE study of the same wing rib with fixed support boundary conditions under design load is available [25]. Application of fixed support as a boundary condition for FE analysis of structural members isolated from the global model implies infinite stiffness of the global model. Therefore, it underestimates the maximum load-carrying capacity of the wing rib. Figure 2 presents a stress contour plot for the wing rib under design load with fixed support as boundary conditions. The maximum stress observed is 388 MPa at the bolt hole.

5 Model Structure

5.1 Coordinate Systems

Defining both global and local coordinate systems in the report is imperative. Without clear definitions, the accurate application of environmental conditions may be compromised, potentially leading to misinterpretation of subsequent results.

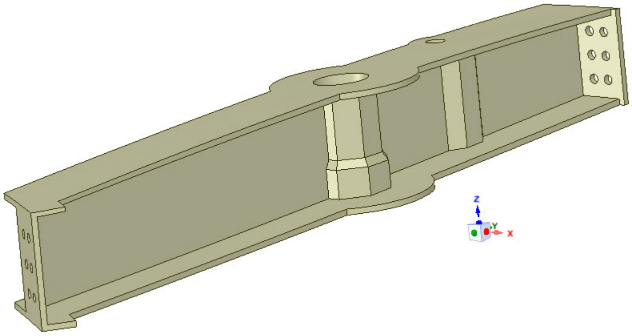

Figure 3: CAD Model of Wing Rib

5.2 Geometry

Dimensional details in the form of diagrams or sketches can also be included. The present trend is to use computer-aided design (CAD) software like CATIAⓇ, SolidWorksⓇ, AutoCADⓇ, etc. to develop models. The CAD model then forms a template for downstream FE modeling after its translation from the CAD environment to FE software. In FE software, geometry preparation involves cleaning, simplification, decimation, and symmetric exploitation. Therefore, these preceding steps may also be covered to establish the reproducibility and fidelity of the model.

5.3 Material Properties

Relevant material properties such as Modulus of Elasticity, Poisson Ratio, Shear Modulus, etc., should be thoroughly reported. Many Finite Element (FE) codes now include an extensive library of material models, which can be referenced in the report when utilized for analysis.

5.4 Environmental Conditions

Environmental conditions for the FE model consist of imposed boundary conditions and applied loads. It is crucial to report necessary details, including magnitude, time history, direction, and application region of these conditions. Furthermore, explaining their physical interpretation and corroborating them with reality is essential to avoid misinterpretation resulting from differences in syntax among various FE codes [26].

In many cases, applied boundary conditions serve as modest representative approximations. A set of boundary conditions offers a range for variation in Finite Element (FE) results. Such scenarios necessitate a comprehensive consideration of boundary condition selection alongside the predicted variation in results [27]. However, there are instances where a specific FE code does not permit the direct application of certain environmental conditions. For example, in cases where torque cannot be directly applied, an equivalent force multiplied by the moment arm is applied through a rigid connection. The FE report must include such workarounds [3].

Finite Element Analysis (FEA) of a mechanical assembly, comprising various interacting components, is indeed relatively intricate. Interactions between these components are typically idealized; for instance, fasteners may be replaced with 1D beam elements, while mating surfaces are often fixed or replaced with frictional contact [28]. It is crucial to document a comprehensive list of interacting components, specifying the type of interaction (e.g., gap, friction, no separation, no penetration) and the attributes of interaction (such as friction coefficient, stiffness, etc.) [29].

5.5 Model Calibration

In determining certain simulation parameters, heuristics can be employed in addition to actual experimentation [30]. Therefore, validating these assumptions becomes necessary, and this is achieved by comparing the output from the simulation model with the actual experimental output. Based on the correlation or deviation between these two outputs, the model is calibrated to generate pseudo-true values. If model calibration has been employed, it should be reported for result interpretation.

5.6 Case Study

The model of the wing rib is developed in a native CAD environment called Ansys Design ModelerⓇ. Development of CAD model in Design Modeler eliminates translation errors during import of model from CAD to FE environment. Figure 3 presents a CAD model of the wing rib.The linear elastic isentropic material model is used for analysis. Wing rib is assigned material properties of Aluminum Alloy 7050-T7 Yield strength=427 MPa, Modulus of Elasticity=70 GPa and Poisson Ratio=0.33.

Imposed boundary conditions and applied loads are shown in Figure 4. Global coordinates and length scale are presented for visualization. Elastic support represents the stiffness of the attached wing structure. Bolt holes are given free displacement with

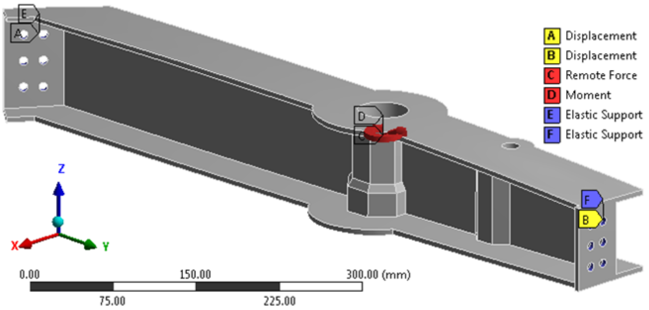

Figure 4: Loads and Boundary Conditions

Table 1: Applied Loads.

Force Component	Magnitude (kN)	Moment Componet	Magnitude (kN.m)
Fx	-3.86	Mx	87.1
Fy	6.9	My	5880
Fz	2.43	Mz	-27.9

linear and rotational stiffness of 99 KN/m and 50 KN.m/rad respectively. Aerodynamic and inertial loads acting on the wing rib are applied as force (8.27 KN) and moment (5881 KN.m) at the spindle of the wing rib. Table 1 presents the components of applied force and moment.

6 Discretization Scheme

Mesh characteristics have a direct impact on the accuracy of results [31]. Therefore, it is imperative to report the discretization scheme to assess its adequacy.

6.1 Element Name

Names of finite elements used for meshing should be provided, for example, CQUAD4, etc.

6.2 Element Attributes

Attributes of finite elements dictate their capability or limitation to capture physical phenomena [32]and should be reported. For example, specifying the type of elements used, such as 1D line (Beam) elements or 2D shell (Triangular or quadrilateral) elements, is essential. Additionally, reporting the number of nodes, shape function, and total degrees of freedom (DOFs) per node is crucial for transparency and accuracy.

6.3 Mesh density

In the FE report, it is important to include details about total degrees of freedom (DOFs), nodes, and elements of the mesh, along with illustrations highlighting mesh pattern (e.g., free or mapped) and density (coarse or fine). Variations in mesh density, which may be adjusted to capture stress gradients and geometric discontinuities [33], should be reported. Additionally, the suitability of local and global mesh densities for the current analysis should be discussed.

6.4 Mesh Convergence

Finite Element Analysis is performed with progressively finer mesh until an appropriate mesh density is identified, which results in no significant change (less than 5%) in the primary FE output [34]. Once this mesh density is determined, the solution is considered independent of mesh, and this mesh is later used for all subsequent analyses. A mesh convergence study should be reported to establish the independence of FE results from mesh density. The maximum equivalent stress is typically used as an appropriate FE output for mesh convergence study. If a secondary output is used for this purpose, the relationship between the primary and secondary outputs should be reported. Additionally, the difference between averaged and un-averaged stress values, typically less than 3 % [35], serves as a good indicator of mesh convergence and may also be reported.

6.5 Mesh Quality

Mesh quality can be evaluated and presented in the FE report [36] using various mesh metrics such as Element Quality, Aspect Ratio, Skewness, Jacobian, Warping Factor, Maximum corner angle, etc. Additionally, various criteria in terms of average values and allowable percentage of bad elements have been developed to assess mesh quality. Locations of bad elements which have failed assessment criteria should also be reported. It is important to note that a few ill-shaped elements in critical regions of the model can significantly affect the accuracy of results [37]

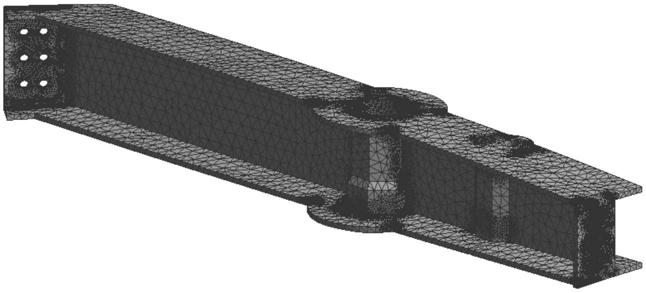

Figure 5: Mesh Model of Wing Rib

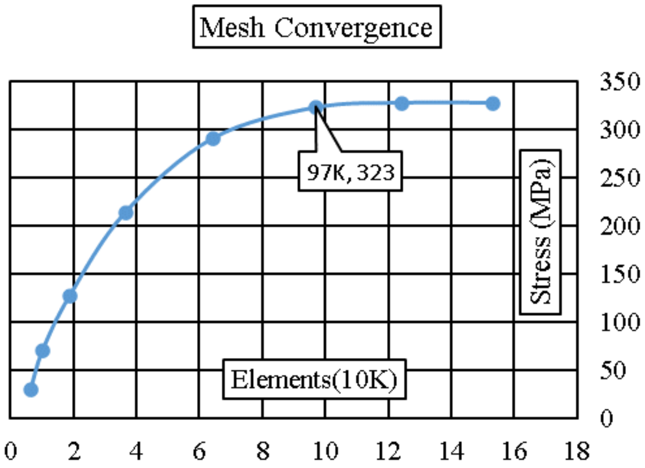

Figure 6: Mesh Convergence Study

6.6 Case Study

An unstructured free mesh is employed for the discretization of the problem domain, with tetrahedral-shaped 10-node elements (named Tet10) with quadratic shape functions. The mesh model of the wing rib is illustrated in Figure 5. Both local and global mesh controls have been employed to enhance mesh quality. Mesh refinement has been implemented at regions of load application, constraints, and geometric transitions.

The mesh convergence study is depicted in Figure 6. It is observed that the mesh-independent solution is obtained with a mesh count of 97,000 elements. The average skewness of this mesh is 0.24, where 0 represents the best quality and 1 is the worst. The value of equivalent stress is recorded as 323 MPa. No bad or ill-shaped elements have been observed in the area of stress gradients.

7 Analysis Setting

A pre-processor generates an input file for the FE solver, which, based on defined analysis settings, performs computations of the numerical model. If an analyst has opted for default settings, this should be explicitly stated. Otherwise, any relevant changes made to these settings should be reported, including: 1. The type of solver used, whether direct or iterative, should be specified. 2. Numerical algorithms employed, such as Newton's method, quasi-Newton methods, or Newton-Raphson method, should be mentioned. 3. Convergence Criteria, including the field variable (e.g., displacement, rotation, force, etc.) and associated tolerances, should be clearly defined. During the solution phase, the solver may generate errors and warning messages, which require critical review by the analyst. If certain measures are implemented to address these warnings, they should be documented in the FE report.

8 Output Parameters

FE outputs for downstream design analysis must be reported, with commonly reported results including contour plots for deformation, strain, and stress. It is also advisable to report the maximum and minimum magnitude of the field variable on these contour plots. While these plots provide useful visualization of field parameters, care should be taken to avoid misrepresentation through graphical artifacts.

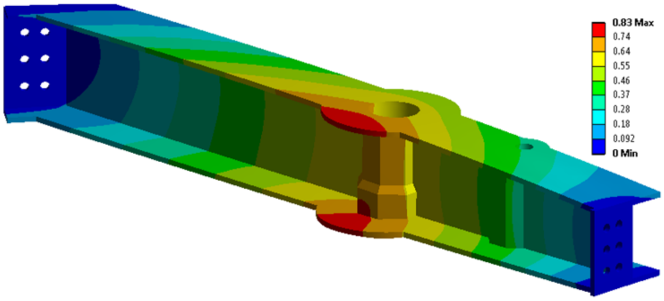

Figure 7: Deformation Field

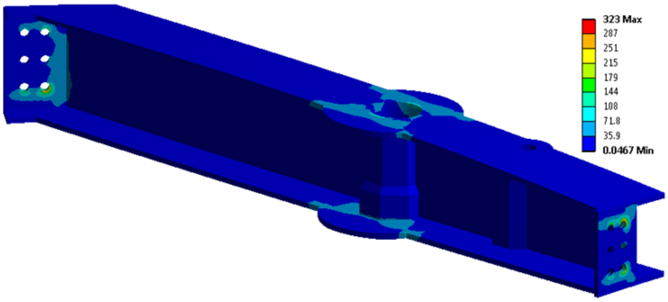

Figure 8: Maximum Stress

8.1 Case Study

Default settings in Ansys solver have been utilized for static structural analysis. Figure 7 illustrates the deformation of the wing rib under design load, with maximum deformation (0.83 mm) observed at the spindle. Figure 8 presents von Mises stress distribution for the rib under design load, with the maximum stress (323 MPa) observed at the bolt holes for the attachment of the rib to the wing.

9 Sensitivity Analysis

Uncertainty in modeling parameters can significantly impact simulation results. Therefore, sensitivity analysis should be conducted and the influence of such uncertainties on Finite Element (FE) results may be reported. In this approach, input parameters are generated based on probability distributions, and a large number of simulations (generally several thousand) are performed to generate a probability distribution of simulation results. This comprehensive sensitivity analysis should be documented to establish the predictive performance of the model under uncertainty in modeling parameters [38].

9.1 Case Study

Max stress under applied load is observed at bolt holes. The original bolt hole diameter is 4 mm. A sensitivity analysis can be performed to determine the effect of the bolt hole on maximum stress. Figure 9 presents the variation in maximum stress with changes in the diameter of bolt holes. The diameter of bolt holes at each side is varied from 3 mm to 5 mm. Minimum stress (286 MPa) is observed when the diameter of the bolt hole is 4.5 mm. Maximum stress is decreased from 323 MPa to 286 MPa (11.4% decrease) by enlarging the diameter of bolt holes from 4 mm to 4.5 mm.

10 Verification

Verification and Validation of the model guarantee accurate results with the right balance between computational cost and accuracy. Verification determines that the model has been solved while validation determines correct model has been solved. The following information should be documented [39]; 1. A comparison of inertial properties such as mass, center of gravity, and moment of inertia between the Finite Element (FE) model and the physical/CAD model should be conducted. 2. Reasonable deformation of the body under its weight (i.e., 1 g condition without applied load) 3. Verification of applied boundary conditions under external loads should be conducted to ensure their observance. 4. Equilibrium conditions can be assessed by comparing reaction forces and moments with the applied loads. 5. Code verification of user-developed subroutines

11 Validation

The validation process determines that correct equations governing a physical phenomenon have been solved. The following validation steps may be reported [38]; 1. When numerically representing complex physical phenomena, Finite Element (FE)

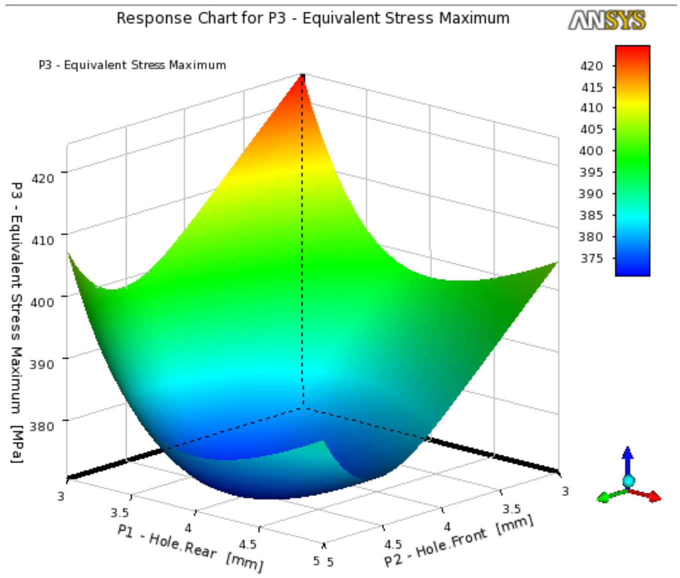

Figure 9: Response Chart

analysts inevitably rely on modeling assumptions. It is essential to report and substantiate these assumptions to assess the efficacy of FE models. 2. Correlation with analytical or empirical solutions of a simplified model is essential to validate the accuracy and reliability of numerical solutions. 3. Correlating the current analysis with similar accredited Finite Element (FE) analyses helps validate the reliability and accuracy of the results. 4. Correlating experimental output with simulation results is crucial to validate and verify the accuracy of the Finite Element (FE) analysis. 5. In the Finite Element (FE) report, it is imperative to provide appropriate references for modeling parameters to ensure credibility. 6. The repeatability of FE results on different hardware/operating systems and FE software demonstrated the robustness of digital twins. 7. Error i.e., difference between simulation result and experimental output, should be reported in terms of relevant statistical metrics.

12 Results Section

Physical interpretation of FE results should be thoroughly reported. The report should deliberate on how and to what extent these results have answered the posed research question. If the results have not conclusively addressed the question, a new approach or methodology can be recommended. Singularities in Finite Element (FE) outputs, often indicated by values approaching infinity, may arise at locations of constraints, applied loads, or zero radii corners. While it is crucial to report these singularities, they may be excluded from discussion with appropriate justification.

12.1 Case Study

By using an elastic support, the maximum stress observed is 323 MPa. This elastic support introduces the stiffness of the global wing model into the solution of the isolated local model. When the stiffness of the global model is made infinite, i.e., a fixed support is used as a boundary condition, the maximum stress increases to 388 MPa. This represents an increase of 20.1% in stress. Therefore, it is concluded that the application of a fixed boundary condition for the local model isolated from a global model under predicts the strength by 20%.

13 Availability

Granting public access to the model enables prospective users to evaluate and enhance its capabilities. When access to the model is provided under appropriate distribution conditions and contractual binding, it fosters improvements in the model. 1. Clear licensing terms should be established for the distribution, reuse, and modification of FE models. 2. An online link or web address may be provided for model download, developmental updates, user reviews, and address of queries by the model owner.

14 Conclusion

This article elucidates essential considerations for reporting in the context of finite element analysis (FEA). Adherence to these guidelines is poised to yield succinct documentation. FEA report fashioned by these guidelines is designed to encapsulate all pertinent details, enabling a comprehensive understanding of the simulation process and facilitating accurate result reproduction. The application of these guidelines serves to instill confidence in the reproducibility, repeatability, and accountability of FEA studies.

These guidelines offer a valuable resource for researchers, journal editors, and funding agencies, equipping them with a framework to evaluate and ascertain the capabilities, limitations, and usability of simulation models. While it may be impractical for a journal manuscript to comprehensively present and discuss all reporting parameters, the identification of omitted sections can serve as a compass, guiding prospective research directions.

Moreover, these guidelines have the potential to evolve into a foundational reporting standard for FEA studies. Over time, the systematic application of these guidelines may contribute to the establishment of standardized protocols for the exchange of FEA models. Notably, there is a contemporary trend among software vendors and seasoned FEA users who are actively developing subroutines for automated reports within FEA software. Incorporation of these guidelines stands to enhance the fidelity of such subroutines.

It is imperative to acknowledge that these guidelines, while comprehensive, may not be exhaustive and are susceptible to evolution concurrent with ongoing advancements in computer modeling and simulation. Furthermore, their adaptability for multi-domain and multi-physics analyses underscores their relevance in the dynamic landscape of simulation methodologies.

Acknowledgments

The author acknowledges the facilitation of his department at Air University for providing all the resources for this publication.

Author Contributions

100 % contribution by AH.

Competing Interests

The author declares that he has no competing interests.

References

[1] S. David Müzel, E. P. Bonhin, N. M. Guimarães, and E. S. Guidi, "Application of the finite element method in the analysis of composite materials: A review," *Polymers*, vol. 12, no. 4, p. 818, 2020.

[2] L. Sabat and C. K. Kundu, "History of finite element method: a review," *Recent Developments in Sustainable Infrastructure: Select Proceedings of ICRDSI 2019*, vol. 3, no. 121, p. 395, 2020.

[3] H.-H. Lee, *Finite element simulations with ANSYS Workbench 21*. SDC publications, 2021.

[4] T. Chandrupatla and A. Belegundu, *Introduction to finite elements in engineering*. Cambridge University Press, 2021.

[5] K. N. B. R. U. A. Mastoi S., Mugheri A. B. M. and M. R. B., "Solution accuracy of finite element grids," *International Journal of Advanced Research in Engineering and Technology*, vol. 4, no. 17, p. 1127, 2021.

[6] A. M. Law, "How to build valid and credible simulation models," in *2022 Winter Simulation Conference (WSC)*, p. 128, IEEE, 2022.

[7] D. Arndt, W. Bangerth, D. Davydov, T. Heister, L. Heltai, M. Kronbichler, M. Maier, J.-P. Pelteret, B. Turcksin, and D. Wells, "The deal. ii finite element library: Design, features, and insights," *Computers & Mathematics with Applications*, vol. 3, no. 128, p. 128, 2021.

[8] K. Srinivas and S. Motera, "Verifications and validations in finite element analysis (fea)," *Ahmedabad: Advanced Scientific and Engineering Services (AdvanSES)*, vol. 3, no. 128, p. 407, 2020.

[9] P. M. Kurowski, *Finite element analysis for design engineers*. SAE International, 2022.

[10] M. Zaheer, P. Lindh, L. Aarniovuori, and J. Pyrhönen, "Comparison of commercial and open-source fem software: A case study," *IEEE Transactions on Industry Applications*, vol. 56, no. 6, p. 6411, 2020.

[11] I. Magomedov and Z. Sebaeva, "Comparative study of finite element analysis software packages," in *Journal of Physics: Conference Series*, vol. 1515, p. 73, IOP Publishing, 2020.

[12] D. Marinkovic and M. Zehn, "Survey of finite element method-based real-time simulations," *Applied Sciences*, vol. 9, no. 14, p. 2775, 2019.

[13] S. Chakraverty, N. Mahato, P. Karunakar, and T. D. Rao, *Advanced numerical and semi-analytical methods for differential equations*. John Wiley & Sons, 2019.

[14] I. Erhunmwun and U. Ikponmwosa, "Review on finite element method," *Journal of Applied Sciences and Environmental Management*, vol. 21, no. 5, p. 999, 2018.

[15] T. Stolarski, Y. Nakasone, and S. Yoshimoto, *Engineering analysis with ANSYS software*. Butterworth-Heinemann, 2018.

[16] I. Koutromanos, *Fundamentals of finite element analysis: Linear finite element analysis*. John Wiley & Sons, 2018.

[17] R. Basu, K. Kirkhope, and J. Srinivasan, "Guideline for evaluation of finite elements and results.," *Miitary Systems Engineering*, p. 64, 2018.

[18] M. Kuzin, P. Vovk, and O. Kuzin, "Mathematical modelling and mechanics approaches in investigation of structural failure causes," in *MATEC Web of Conferences*, vol. 390, p. 04010, EDP Sciences, 2024.

[19] S. Georgescu, P. Chow, and H. Okuda, "Gpu acceleration for fem-based structural analysis," *Archives of Computational Methods in Engineering*, vol. 20, no. 11, p. 111, 2019.

[20] J. Schröder, T. Wick, S. Reese, P. Wriggers, R. Müller, S. Kollmannsberger, M. Kästner, A. Schwarz, M. Igelbüscher, N. Viebahn, *et al.*, "A selection of benchmark problems in solid mechanics and applied mathematics," *Archives of Computational Methods in Engineering*, vol. 28, p. 713, 2021.

[21] A. Mar and M. Hicks, "A benchmark computational study of finite element error estimation," *International journal for numerical methods in engineering*, vol. 39, no. 23, p. 3969, 2018.

[22] V. Saravanan, M. Ramachandran, and C. Raja, "A study on aircraft structure and application of static force," *REST Journal on Advances in Mechanical Engineering*, vol. 1, no. 1, p. 1, 2022.

[23] C. Zhao, A. Alimardani Lavasan, and T. Schanz, "Application of submodeling technique in numerical modeling of mechanized tunnel excavation," *International Journal of Civil Engineering*, vol. 17, p. 75, 2019.

[24] M. W. Sracic and W. J. Elke, "Effect of boundary conditions on finite element submodeling," in *Nonlinear Dynamics, Volume 1: Proceedings of the 36th IMAC, A Conference and Exposition on Structural Dynamics 2018*, p. 163, Springer, 2019.

[25] A. H. Bhutta, "Appropriate boundary condition for finite element analysis of structural members isolated from global model.," *NED University Journal of Research*, vol. 18, no. 3, 2021.

[26] D. Venkatkumar and D. Ravindran, "Effect of boundary conditions on residual stresses and distortion in 316 stainless steel butt welded plate," *High Temperature Materials and Processes*, vol. 38, no. 2019, p. 827, 2019.

[27] F. Guarracino, A. Walker, and A. Giordano, ‘‘Effects of boundary conditions on testing of pipes and finite element modelling,’’ *International Journal of Pressure Vessels and Piping*, vol. 86, no. 2-3, p. 196, 2019.

[28] R. Omar, M. A. Rani, and M. Yunus, ‘‘Representation of bolted joints in a structure using finite element modelling and model updating,’’ *Journal of Mechanical Engineering and Sciences*, vol. 14, no. 3, p. 7141, 2020.

[29] T. Liu, Q. Zhao, Y. Cao, and J. Yang, ‘‘A generic approach for analysis of mechanical assembly,’’ *Precision Engineering*, vol. 54, p. 361, 2020.

[30] S. Ereiz, I. Duvnjak, and J. F. Jiménez-Alonso, ‘‘Review of finite element model updating methods for structural applications,’’ in *Structures*, vol. 41, p. 684, Elsevier, 2022.

[31] A. Ruggiero, R. D’Amato, and S. Affatato, ‘‘Comparison of meshing strategies in thr finite element modelling,’’ *Materials*, vol. 12, no. 14, p. 2332, 2020.

[32] M. S. Petrov and T. D. Todorov, ‘‘Properties of the multidimensional finite elements,’’ *Applied Mathematics and Computation*, vol. 391, no. 125, p. 695, 2021.

[33] A. Nemade and A. Shikalgar, ‘‘The mesh quality significance in finite element analysis,’’ *J. Mech. Civ. Eng*, vol. 17, p. 44, 2020.

[34] K. Jalammanavar, N. Pujar, and R. V. Raj, ‘‘Finite element study on mesh discretization error estimation for ansys workbench,’’ in *2018 International conference on computational techniques, electronics and mechanical systems (CTEMS)*, p. 344, IEEE, 2021.

[35] H. A. Carson, A. C. Huang, M. C. Galbraith, S. R. Allmaras, and D. L. Darmofal, ‘‘Mesh optimization via error sampling and synthesis: An update,’’ in *AIAA Scitech 2020 Forum*, p. 87, 2020.

[36] W. Kwok and Z. Chen, ‘‘A simple and effective mesh quality metric for hexahedral and wedge elements.,’’ in *IMR*, p. 325, Citeseer, 2020.

[37] J. Svetlichny, ‘‘Overview of ansys meshing preprocessor capabilities to create high quality meshes,’’ *Open Information and Computer Integrated Technologies*, no. 95, p. 83, 2022.

[38] C. Oefner, S. Herrmann, M. Kebbach, H.-E. Lange, D. Kluess, and M. Woiczinski, ‘‘Reporting checklist for verification and validation of finite element analysis in orthopedic and trauma biomechanics,’’ *Medical Engineering & Physics*, vol. 92, p. 25, 2021.

[39] M. G. Faes, M. Daub, S. Marelli, E. Patelli, and M. Beer, ‘‘Engineering analysis with probability boxes: A review on computational methods,’’ *Structural Safety*, vol. 93, no. 102, p. 92, 2021.