

The Impact of Material Variations and Loading on the Structural Response of Wind Turbine Blades and CAD-CAE-Based Mold Injection Molding

Muhammad Pasha¹, Mohammad Khosyi², Ryan Febrian Nur Halim², Gunawan Prasetyo¹, Stefanus Nathanael Mahesa¹, Dandun Mahesa Prabowoputra^{1*}

¹ Program Studi Teknik Mesin, Fakultas Teknik, Universitas Jenderal Soedirman, Jl. Raya Mayjen Sungkono, Purbalingga, Jawa Tengah, Indonesia

² Program Studi Teknik Industri, Fakultas Teknik, Universitas Jenderal Soedirman, Jl. Raya Mayjen Sungkono, Purbalingga, Jawa Tengah, Indonesia

*Corresponding author: dandun.mahesa@unsoed.ac.id

Abstract. Advances in plastic manufacturing technology necessitate precise design techniques to reduce product failure rates. Nonetheless, a disparity often exists between 3D model creation and a comprehensive mechanical study prior to the production phase. This study seeks to examine the structural response, including stress distribution, displacement, and strain, in wind turbine blade designs and injection-molding dies by integrating CAD and CAE systems. This study employed Finite Element Analysis (FEA) numerical simulations in SolidWorks to evaluate variations in blade materials (ABS, High Viscosity Polycarbonate, Nylon 6/10) and mold materials (Cast Alloy Steel, Gray Cast Iron, C355 Aluminum Alloy). Boundary conditions were implemented with a torque load of 12 Nm on the blade and a compressive force of 10,000 N on the mold to simulate actual operational conditions. Simulation findings indicated that Nylon 6/10 was the most advantageous material for the blade, exhibiting the lowest deformation among the materials. Simultaneously, all casting materials exhibited safe performance, with stress distributions remaining beneath the material's elastic threshold. This CAD-CAE integration demonstrated efficacy as a design validation instrument, capable of improving manufacturing process efficiency by mitigating prototype error risks.

Keywords: computer aided design, injection molding, finite element analysis.

How to cite (in APA style):

Pasha, M., Kosyi, M., Halim, R.F.N., Prasetyo, G., Mahesa, S.N., Prabowoputra, D.M. (2026). The Impact of Material Variations and Loading on the Structural Response of Wind Turbine Blades and CAD-CAE-Based Mold Injection Molding. *Proceeding of IJC-MaSTEd*, 1(1), 298-304

Copyright © 2026 Authors.

DOI: <https://doi.org/10.31571/ijcmasted.v1i1.1030>

1. Introduction

With today's rapidly advancing technology, work in the field of technical drawing relies heavily on computer-based software to ensure that the design process runs effectively and efficiently. The use of Computer-Aided Design (CAD) software, such as SolidWorks, enables the creation of technical drawings with greater speed and precision. Consequently, CAD technology has been widely adopted across various industrial sectors due to its ability to accelerate design time while reducing production costs (Fuchs et al., 2022). Advances in manufacturing technology have also seen significant progress, particularly in the production of plastics through plastic injection molding. These advancements are supported by the use of CAD, which enables the design of products in the form of three-dimensional models with a high degree of accuracy before the manufacturing process begins (Hrechanyi et al., 2025). Consequently, the use of CAD is crucial for ensuring the correctness of a product's shape, dimensions, and function, as well as for minimizing potential errors during the production stage.

In the past, the design process was largely done manually, which took a relatively long time and carried a significant risk of design errors. With technological advancements, CAD software like SolidWorks has emerged as a more efficient solution, capable of accurately generating 3D models and streamlining the design process. Additionally, SolidWorks is equipped with simulation and technical analysis features that allow for design evaluation before entering the production phase (Hunde & Woldeyohannes, 2022). SolidWorks is also known as a relatively user-friendly CAD software, making it widely utilized as a learning tool for technical drawing. This software is not only used for drafting and designing but also for testing materials before the production process begins. Computer-Aided Engineering (CAE) serves as an analytical tool using a numerical approach to generate simulations that closely approximate real-world conditions, thereby enabling potential design issues to be detected earlier and minimizing the risk of failure during the production stage (Ikubanni et.al., , 2022).

Previous research conducted by Kong et.al (2003), utilized CAE simulation to predict product failure through the analysis of injection molding process parameters. That study focused on determining optimal process parameters and identifying potential product defects based on simulation results, but did not proceed to the stage of advanced mechanical analysis. Consequently, the influence of molding process conditions on the product's mechanical response—such as stress distribution or the ability to withstand force and torque loads—has not been quantitatively analyzed. Another study used SolidWorks CAD and simulation to evaluate the performance of cooling systems in injection molds, but did not correlate these cooling results with the mechanical strength analysis of the final product (Nguyet, 2023). This has led to the relationship between the results of the mold process simulation and the structural performance of the product after being subjected to working loads not being comprehensively explained. Based on a review of these studies, there remains a research gap in the form of the lack of integration between CAD-based 3D model design and advanced mechanical analysis, which includes mesh modeling, the application of force or torque loads, and the numerical evaluation of stress distribution. Therefore, this study offers a novel approach by integrating 3D design using CAD, a meshing process tailored to loading conditions, and CAE simulation to analyze stress responses resulting from force and torque loads, thereby providing a more comprehensive understanding of the product's structural performance prior to the production stage.

In product design and analysis practice, the use of SolidWorks is often limited to creating 3D designs without fully utilizing its simulation features. Furthermore, mechanical analysis is typically performed using only a single mesh size, a single material type, and a single loading condition; consequently, the effects of variations in mesh size, force, and torque on simulation results have not been thoroughly investigated. Another common issue is that mold analysis in the injection molding process has not been integrated with the overall product analysis. Therefore, the research problem in this study is to determine how variations in mesh size, material type, and force and torque loading affect stress distribution in 3D product and mold designs analyzed using SolidWorks. The research methodology began with the creation of a three-dimensional design using SolidWorks, followed by mechanical simulation through mesh sizing and the application of force and torque loads on various material variations. The simulation was conducted using the SolidWorks Simulation feature to obtain results in the form of stress, strain, and deformation distributions. Subsequently, injection mold design and analysis were performed with mesh sizing adjusted to the force loads during the molding process, to identify stress distributions and critical areas in the mold.

The objective of this study is to analyze the effects of mesh size, material variations, and the application of force and torque loads on stress distributions in 3D product and mold designs using SolidWorks. It is hoped that the contributions of this research can serve as a reference for the integrated use of SolidWorks in the design, mechanical simulation, and mold analysis processes,

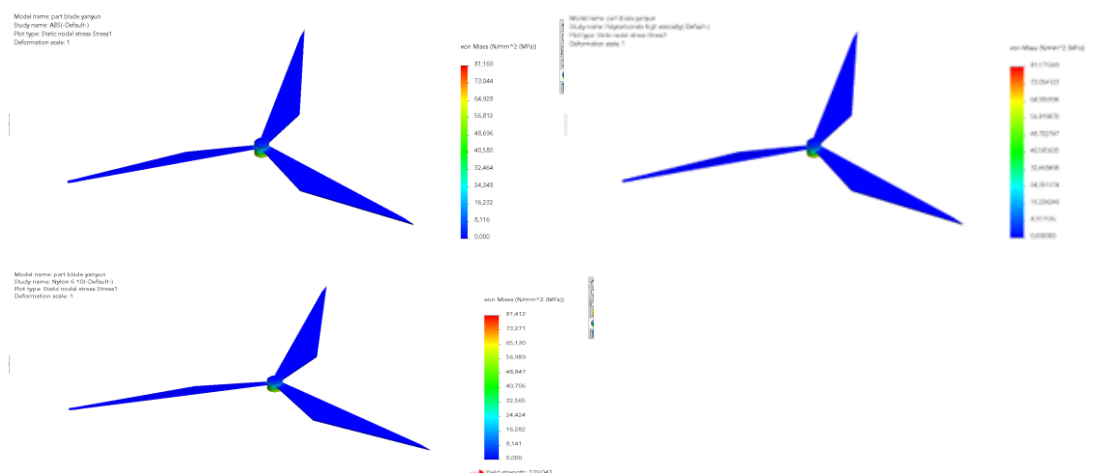
thereby assisting in the evaluation of design strength and the determination of appropriate simulation parameters prior to the manufacturing process.

2. Method

This study was conducted using a numerical simulation method based on Finite Element Analysis (FEA) with the aid of SolidWorks Simulation software. The research subjects included a three-bladed plastic wind turbine blade, as well as the upper and lower molds used in the blade manufacturing process. The geometric models were created as three-dimensional solids, with the blades arranged in a circular pattern. The structural analysis was performed under static conditions. In the blade simulation, a torsional load of 12 Nm was applied to the cylindrical surface of the hub where the blade is connected, with the direction of the torque determined based on the axis of the shaft cylinder. Boundary conditions were applied using fixed geometry on the hub surface connected to the shaft, thereby restricting displacement and rotation in that region. The blade material is varied using ABS, High Viscosity Polycarbonate, and Nylon 6/10, which are available in the SolidWorks material library. In the die simulation, the model is divided into an upper die and a lower die and analyzed separately to account for differences in load distribution and boundary conditions. A compressive load of 10,000 N is applied to the mold contact surface, while the outer surface of the mold is given a fixed-geometry boundary condition. The mold materials used include Cast Alloy Steel, Gray Cast Iron, and C355 Aluminum Alloy (Permanent Mold Cast). The mesh used in all simulations is a solid mesh with a mesh level of 6. The evaluated result parameters include maximum von Mises stress, maximum displacement, and equivalent strain (ESTRN), which are extracted from the entire component volume to determine the structural response of each material.

3. Results and Discussion

The results of the structural simulation on the turbine blades are shown in Figures 1, 2, and 3. The structural behavior of the turbine blades shows clear differences among the plastic materials used. Under the same torsional loading conditions, the ABS material experienced a maximum stress of 35.484 MPa with a displacement of 6.597 mm and an equivalent strain of 0.0137 mm/mm. The relatively large displacement value indicates that ABS tends to undergo significant deformation when used as a blade material. High-Viscosity Polycarbonate produces a slightly lower maximum stress of 33.325 MPa, with a displacement of 5.694 mm and a strain of 0.0109 mm/mm. Although there is a reduction in deformation compared to ABS, the structural response of polycarbonate still exhibits a tendency to flex under the applied torsional load.



Gambar 1 Stress simulation results for each material

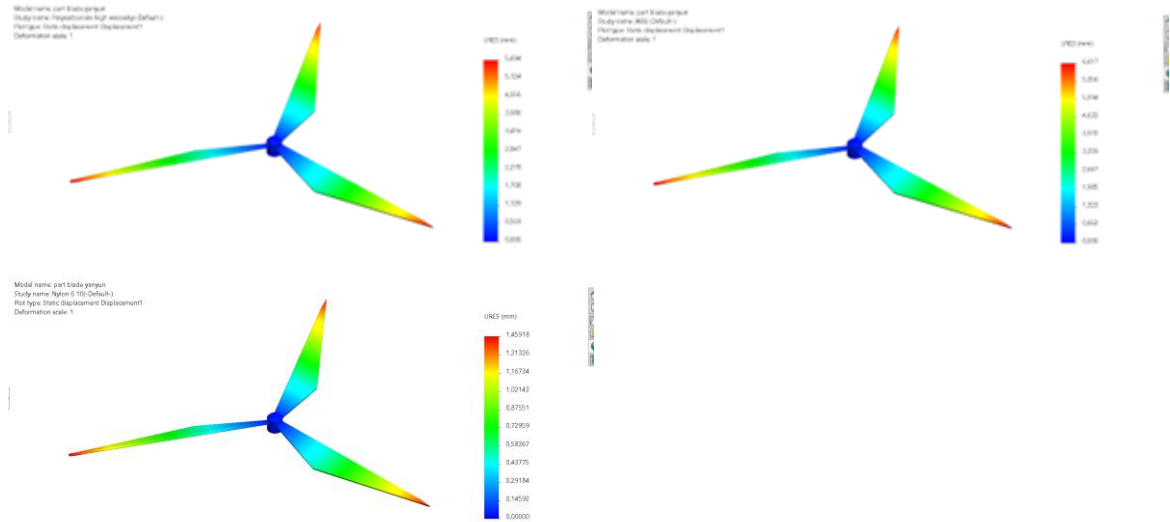


Figure 2 Displacement simulation results for each material

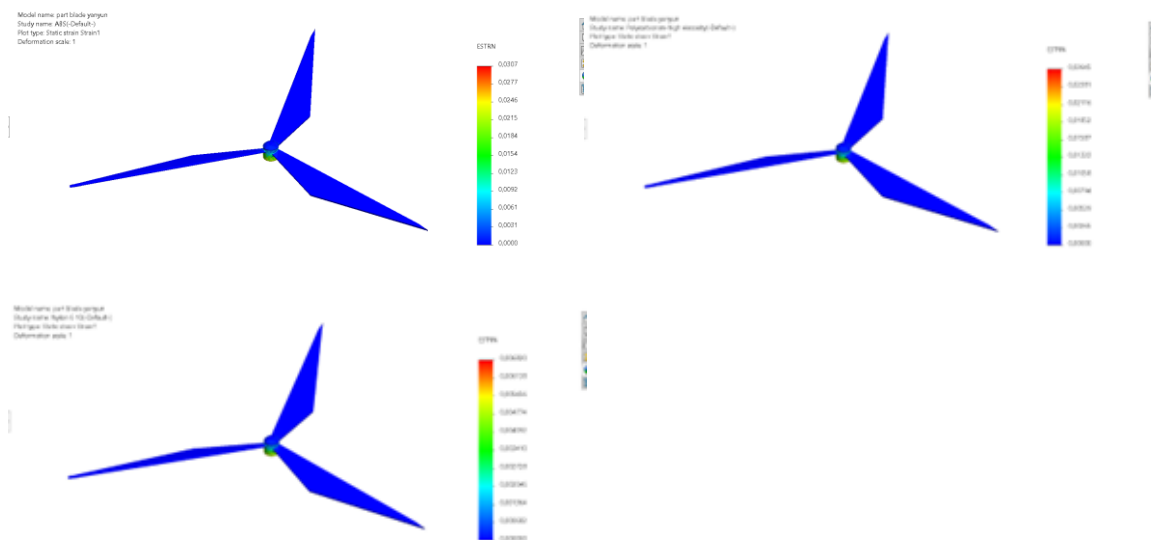


Figure 3 Strain simulation results for each material

The maximum stress, displacement, and strain values for the turbine blade for each material variation are shown in Table 1.

Table 1. Blade simulation results

Plastic Materials	Stress (Mpa)	Displacement (mm)	Strain (mm/mm)
ABS	35,484	6,597	0,0137
PC High Viscosity	33,325	5,694	0,0109
Nylon 6/10	33,215	1,454	0,0029

Nylon 6/10 exhibited the most stable response among the three materials analyzed. The maximum stress reached 33.215 MPa, while the displacement and strain were only 1.454 mm and 0.0029 mm/mm, respectively. This significant difference in displacement indicates that Nylon 6/10 has higher stiffness, allowing it to maintain its geometric shape more effectively. Across all material variations, the maximum stress tends to be concentrated in the transition zone between the blade and the hub, which is a critical point due to changes in geometry and the transfer of torque from the shaft to the blade. This pattern aligns with the fundamental principles of structural mechanics and finite element analysis, where stress concentrations typically occur at areas of geometric discontinuity.

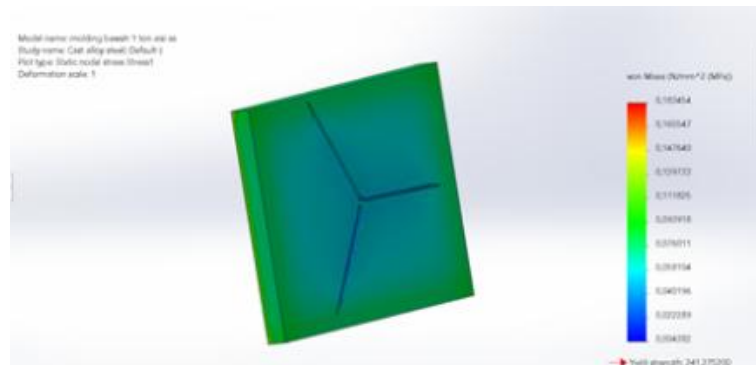


Figure 4 Stress results for the lower mold of a cast alloy steel

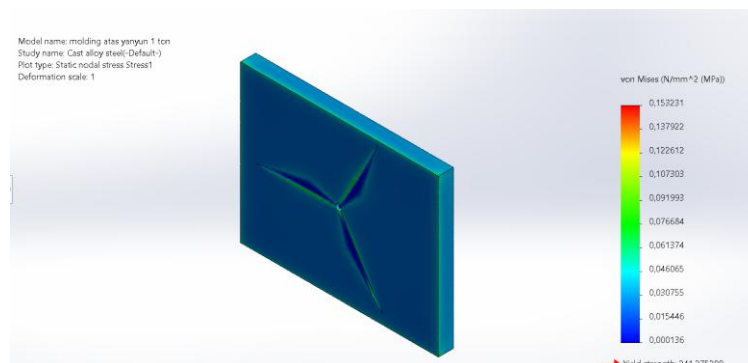


Figure 5 Stress results for the upper mold of the cast alloy steel

The results of the stress distribution simulation for the lower and upper molds made of cast alloy steel are shown in Figures 4 and 5. The lower mold exhibits higher maximum stress values due to a more dominant load distribution. Nevertheless, the resulting displacement and strain values remain very small and do not indicate a potential for structural failure. In the upper mold, the stress distribution shows a relatively uniform pattern with low maximum values. This indicates that the upper mold does not undergo significant deformation during the loading process, regardless of the material variation used.

The structural response of the die exhibits different characteristics compared to the blade. In the upper die, the maximum stress ranges from 0.037 to 0.038 MPa, with very small displacements and strains. A summary of the upper die simulation results for each material is presented in Table 2. The differences between materials are relatively insignificant, although Cast Alloy Steel exhibits the smallest tendency for deformation. This indicates that the stiffness of the die material is sufficient to withstand the applied compressive load without significant deformation.

Table 2. Results of the structural simulation of the upper mold

Top Molding Simulation			
Molding Materials	Stress (MPa)	Displacement (mm)	Strain (mm/mm)
Cast Alloy Steel	0,03765	0,0000115	0,00000017
Gray Cast Iron	0,03766	0,0000335	0,00000048
C355 Alumunium	0,03812	0,0000346	0,00000048

In the lower mold, the maximum stress value increased to approximately 0.096–0.097 MPa. This increase is related to the position of the lower mold, which bears the load more directly during the compression process. Nevertheless, the resulting displacement and strain values remained very small and showed no indication of structural failure in any of the analyzed mold materials. The difference in response between the upper and lower dies reflects the asymmetric load distribution, as is common in two-part die systems. The simulation results for the lower die with material variations are presented in Table 3.

Table 3 Results of the structural simulation of the lower mold

Bottom Molding Simulation			
Molding Materials	Stress (MPa)	Displacement (mm)	Strain (mm/mm)
Cast Alloy Steel	0,09733	0,0000176	0,00000039
Gray Cast Iron	0,09727	0,0000513	0,0000011
C355 Alumunium	0,09697	0,000053	0,0000011

Conceptually, these results confirm that material selection plays a key role in determining the deformation response of rotating structures. In practical terms, the simulation results indicate that Nylon 6/10 is the most suitable material candidate for small-scale wind turbine blades because it can withstand torsional loads with minimal deformation, while all the molded materials analyzed demonstrate adequate structural performance for use in the molding process.

4. Conclusion

The simulation results show that ABS material exhibits the greatest deformation, with a displacement of 6.597 mm and a strain of 0.0137 mm/mm, while High Viscosity Polycarbonate exhibits a displacement of 5.694 mm and a strain of 0.0109 mm/mm. Nylon 6/10 material provided the best performance with the smallest displacement, namely 1.454 mm, and a strain of 0.0029 mm/mm, making it the most structurally stable. The maximum stress on the blade was in the range of 33–35 MPa and was concentrated in the transition area between the blade and the hub. On the mold simulation, the maximum stress on the upper mold was in the range of 0.037–0.038 MPa, while on the lower mold it was approximately 0.096–0.097 MPa, with very small displacement and strain. These results indicate that all mold materials are safe for use in the molding process.

References

- Fuchs, D., Bartz, R., Kuschmitz, S., & Vietor, T. (2022). Necessary advances in computer-aided design to leverage on additive manufacturing design freedom. *International Journal on Interactive Design and Manufacturing (IJIDeM)*, 16(4), 1633-1651. <https://doi.org/10.1007/s12008-022-00888-z>
- Hrechanyi, O. M., Vasilchenko, T. O., Vlasov, A. O., Ivakhnenko, O. P., & Vernydub, M. V. (2025). The Role of Cad in the Development of Engineering Solutions for the Metallurgical Industry: Experience in Using ANSYS, Autocad, Solidworks and MatlaB. *Системні технології*, 3(158), 10-20. <https://doi.org/10.34185/1562-9945-3-158-2025-02>
- Hunde, B. R., & Woldeyohannes, A. D. (2022). Future prospects of computer-aided design (CAD)—A review from the perspective of artificial intelligence (AI), extended reality, and 3D printing. *Results in Engineering*, 14, 100478. <https://doi.org/10.1016/j.rineng.2022.100478>
- Ikubanni, P. P., Adeleke, A. A., Agboola, O. O., Christopher, C. T., Ademola, B. S., Okonkwo, J., Adesina, O. S., Omoniyi, P. O., & Akinlabi, E. T. (2022). Present and future impacts of computer-aided design/computer-aided manufacturing (CAD/CAM). *Journal Européen des Systèmes Automatisés*. 55(3), 349. <https://doi.org/10.18280/jesa.550307>
- Kong, L., Fuh, J. Y. H., Lee, K. S., Liu, X. L., Ling, L. S., Zhang, Y. F., & Nee, A. Y. C. (2003). A Windows-native 3D plastic injection mold design system. *Journal of Materials processing technology*, 139(1-3), 81-89. [https://doi.org/10.1016/S0924-0136\(03\)00186-9](https://doi.org/10.1016/S0924-0136(03)00186-9)
- Nguyet, V.N. (2023). Design plastic injection mold with the help of modular software mold wizard. *European Modern Studies Journal*, 7(1), 239-253. [https://doi.org/10.59573/emsj.7\(1\).2023.22](https://doi.org/10.59573/emsj.7(1).2023.22)