Structural Strength Analysis and Mold Fan Blade Design Optimization using Finite Element Analysis

Azka Tristan Virsano Kusuma[1], Fakhri Tri Wibowo[2], Abdillah Sabari[2], Evan Agit Prianto[1], Stefanus Nathanael Mahesa[1], Dandun Mahesa Prabowoputra[1]*

[1]Program Studi Teknik Mesin, Fakultas Teknik, Univeristas Jenderal Soedirman Purbalingga, Jl. Raya Mayjen Sungkono No.KM 5, Dusun 2, Blater, Kec. Kalimanah, Kabupaten Purbalingga, 53371
[2]Program Studi Teknik Industri, Fakultas Teknik, Univeristas Jenderal Soedirman Purbalingga, Jl. Raya Mayjen Sungkono No.KM 5, Dusun 2, Blater, Kec. Kalimanah, Kabupaten Purbalingga, 53371
Corresponding author: dandun.mahesa@unsoed.ac.id

Abstract. This study aims to analyze the structural performance of a fan blade and its injection molding mold prior to the manufacturing stage using a Computer-Aided Design (CAD) and Computer-Aided Engineering (CAE) simulation approach. The scope of the study includes three-dimensional modeling of the fan blade and injection mold, a mesh study to obtain convergent simulation results, and mechanical response analysis under torsional and compressive loading conditions. Finite Element Analysis (FEA) was performed using SolidWorks Simulation with variations of fan blade materials, namely ABS, PA Type 6, and PEEK, as well as mold materials consisting of cast alloy steel, cast carbon steel, and cast stainless steel. The simulation results indicate that a mesh size of 6 mm provides stable von Mises stress values with an average error of 0.05%. An increase in applied torque on the fan blade results in a linear increase in stress and displacement, where PEEK exhibits the best structural stability with the smallest deformation. In the injection molding simulation, increasing compressive force leads to a linear increase in von Mises stress in both the cavity and core, with the maximum stress occurring in the core mold made of cast stainless steel. Furthermore, the simulation identifies several critical regions experiencing maximum stress, particularly in the fan blade shaft area.
Keywords: Fan blade, Injection molding, Finite element analysis, Mesh study, SolidWorks

How to cite (in APA style):
Kusuma, A.T.V., Wibowo, F.T., Sabari, A., Prianto, E.A., Mahesa, S.N., & Prabowoputra, D.M. (2026). Structural Strength Analysis and Mold Fan Blade Design Optimization using Finite Element Analysis. *Proceeding of IJC-MaSTEd*, 1(1), 147-161.

DOI: https://doi.org/10.31571/ijcmasted.v1i1.1034

1. Introduction

In the current era of technological advancement, work in the field of technical drawing requires the support of computer-based software to ensure the design process is carried out effectively and efficiently. The utilization of Computer-Aided Design (CAD), such as SolidWorks, enables the creation of technical drawings that are faster and more accurate. The application of CAD has been widely adopted across various industries because it can save design time and reduce production costs [1]. Manufacturing technology is advancing rapidly, particularly in plastic production through the plastic injection molding process. This development is supported by CAD technology, which allows for high-precision 3D product design before the manufacturing process begins. The use of CAD is essential to ensure that the shape, dimensions, and functionality of the product meet requirements while minimizing errors during the production stage. [2], [3], [4].

In the past, the design process was largely conducted manually, which was time-consuming and prone to design errors, but with technological advancements, CAD software such as SolidWorks has become a more efficient solution by enabling accurate 3D modeling and simplifying the design process. SolidWorks is also equipped with simulation and technical analysis features that support design evaluation before the production stage [5], serving as a user-friendly CAD tool for learning technical drawing that can be used to design, draft, and test materials before manufacturing [6]. Computer Aided Engineering (CAE) further assists this by using numerical engineering approaches to obtain analysis results that closely resemble real-world conditions, allowing potential design issues to be identified early and minimizing production risks [7]. Research by [7] utilized CAE simulations to predict product failure in front covers through injection molding process parameter analysis, yet it focused on optimal parameters and defect prediction without extending the results to advanced mechanical analysis. Consequently, the impact of molding process conditions on the mechanical response of the product, such as stress distribution or the ability to withstand force and torque, has not been quantitatively studied. Similarly, research by [8] utilized SolidWorks CAD and CFD simulations to analyze cooling uniformity in injection molds but focused on cooling system performance and temperature distribution without linking those results to the mechanical strength of the final product. Based on these studies, a research gap exists in the lack of integration between CAD-based 3D modeling and advanced mechanical analysis that considers mesh modeling, force or torque loading, and numerical stress distribution evaluation. Therefore, this study offers novelty by integrating 3D design using CAD, meshing stages tailored to loading conditions, and CAE simulations to analyze stress responses due to force and torque, providing a more comprehensive overview of structural performance before production. In product design, SolidWorks is often only used for 3D modeling without comprehensive simulation, and mechanical analysis is usually limited to a single mesh size, material type, and load. As a result, the influence of variations in mesh, material, force, and torque on stress distribution has not been deeply analyzed, including in molds for the injection molding process. This research analyzes the effects of these variations on stress distribution in 3D product and mold designs using SolidWorks, with the results expected to serve as a reference for the integrated use of SolidWorks in design, mechanical simulation, and mold analysis prior to manufacturing.

2. Method

This research employs a numerical simulation method based on Computer-Aided Design (CAD) and Computer-Aided Engineering (CAE) using SolidWorks software. The initial phase begins with the creation of a three-dimensional design of the fan blade and the injection mold used in its production, with the geometric models developed according to technical design principles to represent real-world operating conditions. Before conducting the primary simulation, a mesh study is performed to ensure that the Finite Element Analysis (FEA) results are convergent, stable, and not significantly influenced by variations in mesh size, aiming to determine the optimal element size for scientifically reliable simulation results. The fan blade materials analyzed include ABS, PEEK, and PA Type 6, referencing studies [9, 10, 11] which identify these as suitable materials for fan blades, while the mold materials consist of Cast Alloy Steel, Cast Carbon Steel, and Cast Stainless Steel. This selection allows for a comparison of the mechanical response of each material under specified loading conditions. Mechanical simulation of the fan blade is conducted by applying torque loads of 0.3 Nm, 3 Nm, and 5 Nm, based on research [12] which indicates these-torque values correspond to variations in the number of blades (3, 4, and 5) as determined through SolidWorks Flow Simulation using Computational Fluid Dynamics (CFD). Meanwhile, the mold simulation is carried out by applying force loads of 10,000 N, 15,000 N, and 20,000 N to represent the pressure during the injection molding process. All simulations are executed using SolidWorks Simulation with mesh settings validated through the mesh study, and the resulting stress distribution, strain, and deformation are analyzed comparatively to evaluate the influence of different materials on the structural performance of both the fan blade and the mold.

3. Results and Discussion

A mesh study was conducted as the initial stage of numerical analysis to ensure that the simulation results were not influenced by the size of the mesh elements used and could accurately represent the structural behavior of the fan blade, including the hub design which is a critical area for load distribution and stress concentration.

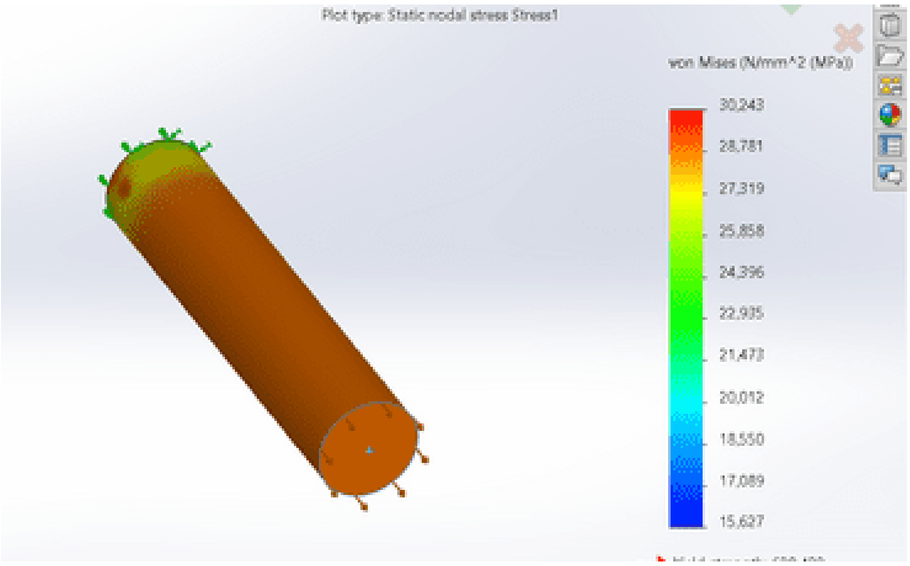

Figure 1. Mesh Study Results on SolidWorks Tube

An example of the mesh study simulation results on the hub design using SolidWorks is shown in Figure 1, illustrating the mesh distribution as a fundamental basis for evaluating mesh quality before proceeding with further mechanical analysis.

Table 1. Results of the Mesh Study in Solidworks

No	Mesh Size	Number of Elements	Stress Value N/mm^2	Error
1	6 mm	1222	28,878	0,03%
2	5 mm	1598	28,875	0,04%
3	4 mm	3170	28,873	0,05%
4	3 mm	7967	28,872	0,05%
5	2 mm	24670	28,866	0,07%
		Average		0,05%

Based on the mesh study results, a mesh size of 6 mm was selected as the most optimal configuration because it produced a stable von Mises stress value of 28.878 N/mm^2 with a low error rate of 0.03%. Although the use of a finer mesh significantly increased the number of elements, the change did not provide a significant impact on the resulting stress values. Therefore, the use of a 6 mm mesh is considered to provide an optimal balance between simulation accuracy and computational efficiency.

Table 2. Value Validation

No	Force Variation (N)	Stress Value Modeling (N/mm^2)	Anilitic Value (N/mm)	Error
1	5000	14,439	14,44314456	0,03%
2	10000	28,878	28,88628912	0,03%
3	15000	43,317	43,32943368	0,03%
4	20000	57,756	57,77257825	0,03%
5	25000	72,195	72,21572281	0,03%
		Average		0,03%

Simulation results indicate that increasing the force variation from 5,000 N to 25,000 N leads to a linear increase in stress values. The stress values obtained range from 14.439 N/mm² to 72.195 N/mm², with a relatively small and stable error rate of 0.03%. The consistency of the error values across each force variation demonstrates that the resulting structural response is stable, ensuring that the applied loading representatively describes the material behavior within the analyzed force range.

Torque Testing on Fan Blade

Torque testing on the fan blade was conducted to analyze the mechanical response of the material to the applied rotational load. The observed simulation parameters included equivalent stress (von Mises stress), displacement, and equivalent strain as the basis for evaluating material performance. The test results are presented in table format to facilitate a comparison between the various torque amounts and materials.

Table 3. Torque Comparison with a Value of 0.3

Material	Torque (Nm)	Stress Value (N/m^2)	Displacement (mm)	Strain
ABS		795.975	0,964	0,00
PA Type 6	0,3	811.371	0,777	0,00
PEEK		796.258	0,492	0,00

Based on the torque simulation results on the fan blade using SolidWorks with a torque load of 0.3 N·m, it was found that the ABS material experienced a maximum stress of 795,975.375 N/m² with the highest displacement of 0.964 mm, indicating the largest deformation. The PA Type 6 material produced the highest maximum stress of 811,371.063 N/m² with a displacement of 0.777 mm, which signifies better stiffness compared to ABS. Meanwhile, the PEEK material showed a maximum stress of 796,258.125 N/m² with the smallest displacement of 0.492 mm, thereby possessing the best structural stability against torque loading. The strain values for all three materials at 0.00 indicate that the deformation occurred remains within the elastic limit; therefore, it can be concluded that PEEK is the most optimal material for the fan blade based on the torque simulation results.

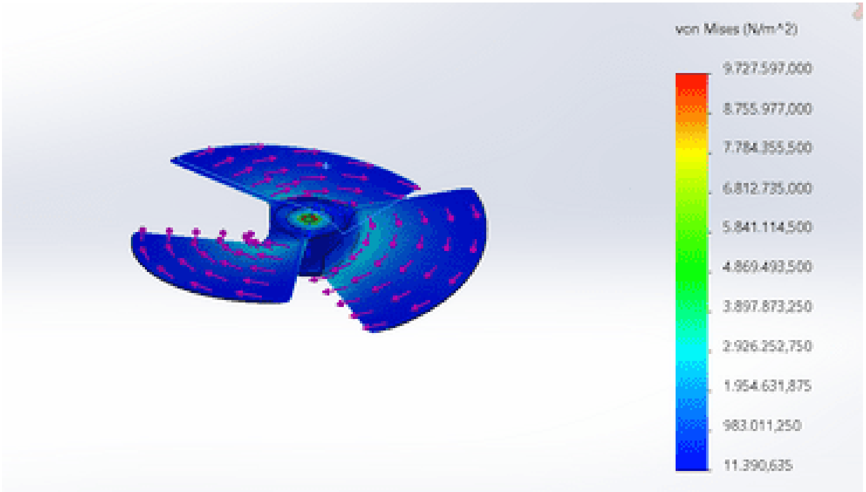

Figure 2. Visual Stress Material ABS

Figure 2 presents an example of the stress simulation on ABS material, illustrating the magnitude of internal forces acting on the material per unit area. The simulation results show that the highest stress concentration is centered on the fan blade shaft. The color indicators in the green area signify that this region experiences the highest internal load and could potentially become the starting point of material failure if the occurring stress exceeds the material's strength limits.

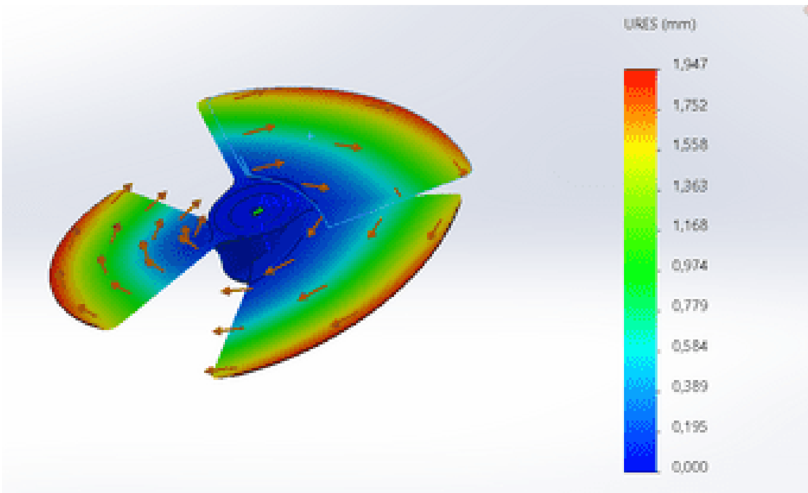

Figure 3. Visual Displacement Material ABS

Figure 3 shows an example of the displacement simulation on ABS material, representing the change in position of a point on the object that illustrates the global structural deformation under loading. The red area is located at the tips of the fan blades, which are furthest from the support point. This indicates that these sections experience the largest displacement due to the torque effect, while the blue areas are generally found around the restraint regions, experiencing minimum displacement.

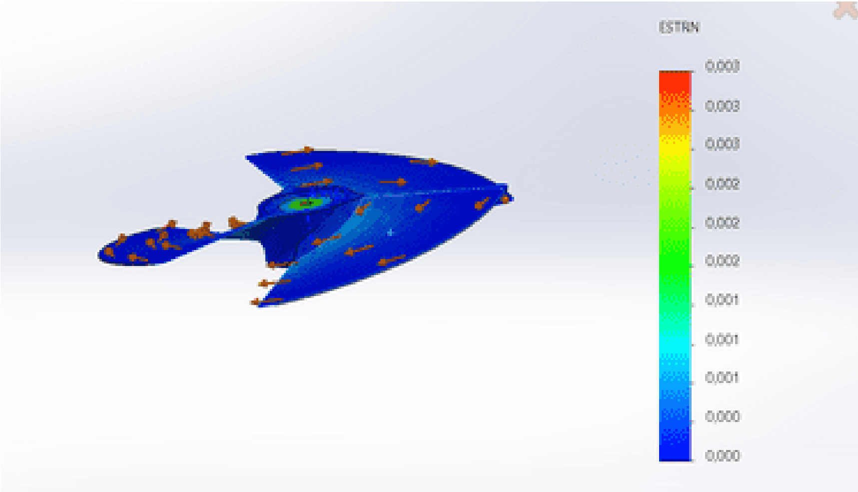

Figure 4. Visual Strain Material ABS

Figure 4 presents an example of the strain simulation on ABS material, which functions to measure relative deformation, defined as the ratio of the change in length to the original length. The simulation results show that the red area indicates the portion of the structure experiencing the greatest strain, occurring in regions with stress concentrations such as the fan blade shaft, cross-sectional transition areas, or regions with significant geometric changes. Conversely, the blue area indicates parts of the structure experiencing minimal strain, which are relatively non-critical regarding deformation failure.

Table 4 presents the results of a torque simulation of 3 N·m on the propeller fan for ABS, PA Type 6, and PEEK materials.

Tabel 4. Comparison of Torque with the Value 3

Material	Torque (N.m)	Value Stress (N/m^2)	Displacement Value (mm)	Strain Value
ABS		6.540.605	8,403	0,002
PA Type 6	3	6.928.086	6,922	0,001
PEEK		7.261.288	4,627	0,001

Based on the torque simulation results of the fan blade using SolidWorks with a torque load of 3 Nm, as shown in Table 4, the ABS material experienced the largest displacement of 8.403 mm with a maximum stress of 6,540,605.00 N/m². The PA Type 6 material showed a displacement of 6.922 mm with a stress of 6,928,086.00 N/m², while the PEEK material had the smallest displacement of 4.627 mm despite generating the highest maximum stress of 7,261,288.50 N/m². This indicates that PEEK possesses the best structural stability against torque loading.

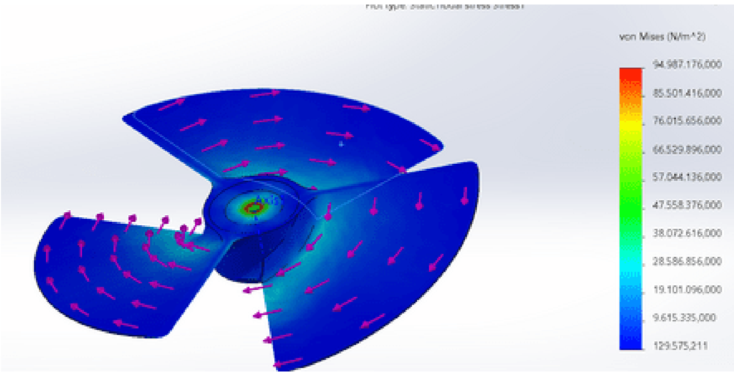

Figure 5. Visual Stress Material PA Type 6

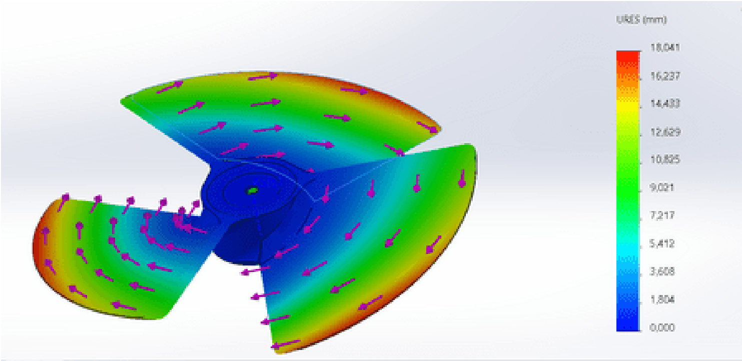

Figure 6. Visual Displacement Material PA Type 6

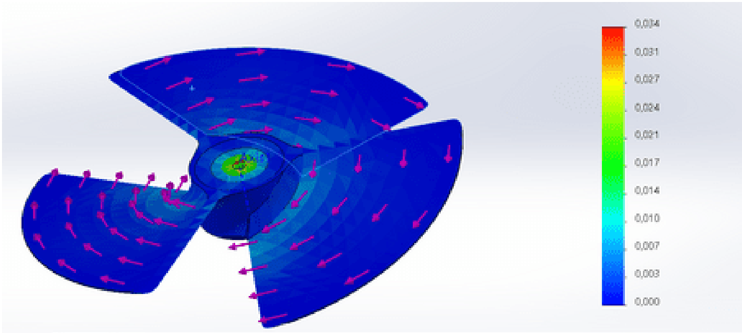

Figure 7. Visual Strain Material PA Type 6

Table 5 presents the torque simulation results at 5 Nm for the fan blade using ABS, PA Type 6, and PEEK materials.

Table 5. Comparison of Torque with a Value of 5

Material	Torque (N.m)	Stress Value (N/m^2)	Displacement Value (mm)	Strain Value
ABS		9.766.298	12,714	0,002
PA Type 6	5	10.586.026	10,648	0,002
PEEK		11.181.139	7,297	0,001

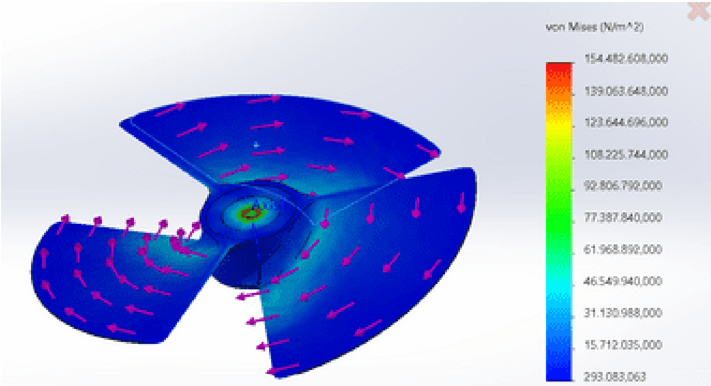

Figure 8. Visual Stress Material PEEK

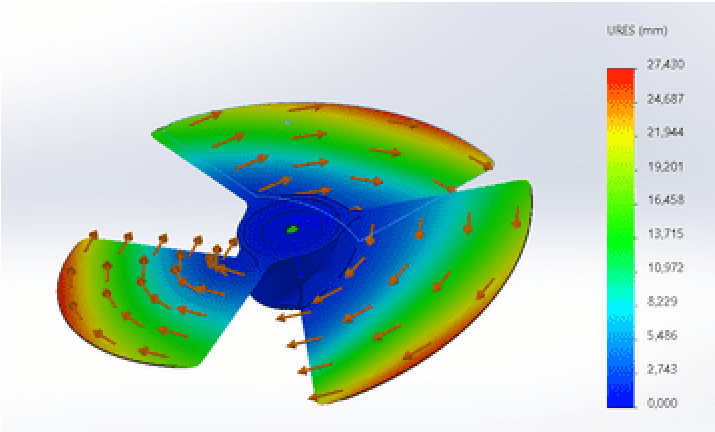

Figure 9. Visual Displacement Material PEEK

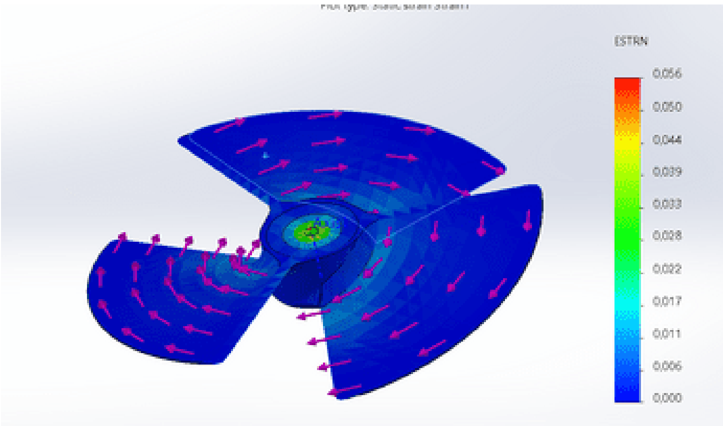

Figure 10. Visual Strain Material PEEK

Based on the torque simulation results for the fan blade using SolidWorks with a torque load of 5 Nm, as shown in Table 5, the ABS material produced a maximum stress value of 9,766,298.00 N/m² with a displacement of 12.714 mm. The PA Type 6 material showed an increase in stress to 10,586,026.00 N/m² with a displacement of 10.648 mm. Meanwhile, the PEEK material generated the highest maximum stress of 11,181,139.00 N/m² but with the smallest displacement of 7.297 mm, indicating the best structural stability against torque loading.

Force Testing on Molding Fan Blade

Force testing on the cavity section of the fan blade mold was conducted to analyze the mechanical response of the material to the applied force. The only parameter observed was the equivalent stress (von Mises stress) as a basis for evaluating material performance. The test results are presented in table format to facilitate a comparison of the effects of force variation and material type on stress distribution.

Table 6. Comparison of force cavity cast alloy steel

Force (N)	Material	Stress Value (N/m^2)
10000		6.298.406,000
15000	*Cast Alloy Steel*	9.447.609,000
20000		12.596.812,000

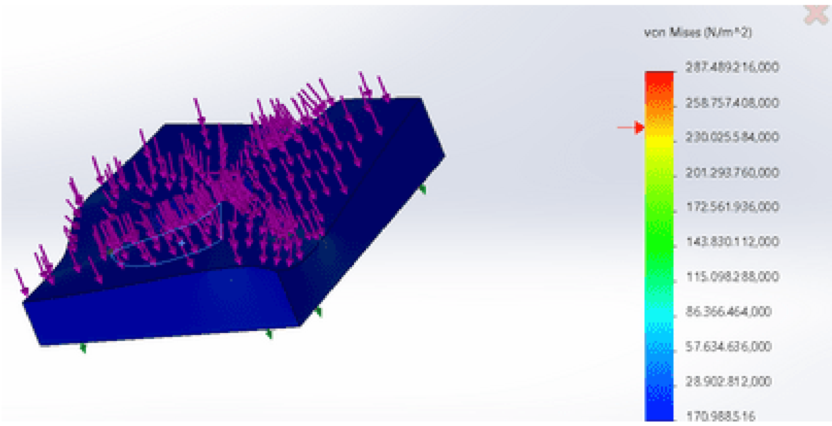

Figure 11. Stress simulation of cavity cast alloy steel

Based on the force loading simulation results using SolidWorks, the table illustrates the relationship between the magnitude of the applied force, the type of material, and the resulting stress. At a force of 10,000 N with cast alloy material, the maximum stress was 6,298,406 N/m². Increasing the force to 15,000 N resulted in a stress of 9,447,609 N/m², while at a force of 20,000 N with steel material, the maximum stress increased to 12,596,812 N/m². These results indicate that the increase in force is directly proportional to the increase in stress on the material, as demonstrated by the simulation outcomes.

Tabel 7. Stress simulation of cavity cast alloy steel

Force (N)	Material	Stress value (N/m^2)
10000		5.909.863,5
15000	*Cast Carbon Steel*	8.864.794
20000		11.819.727

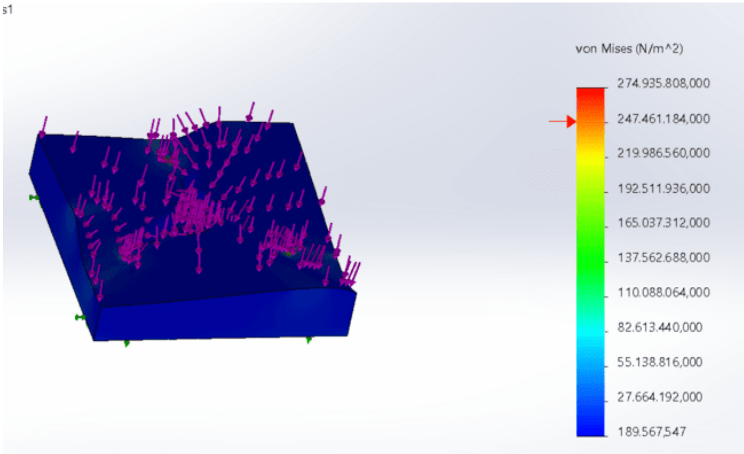

Figure 12. Stress simulation of cavity cast carbon steel

Based on the force loading simulation results, the table illustrates the influence of force variations on the stress values of the material. At a force of 10,000 N with cast carbon material, a stress of 5,909,863.50 N/m² was obtained. Increasing the force to 15,000 N resulted in a stress of 8,864,794.00 N/m², while at a force of 20,000 N with steel material, the stress value increased to 11,819,727.00 N/m². These results indicate that the greater the applied force, the higher the stress value occurring within the material.

Tabel 8. Comparison of force cavity cast stainless steel

Force (N)	Material	Stress Value (N/m^2)
10000		10.338.960
15000	*Cast Stainless Steel*	15.508.437
20000		20.677.920

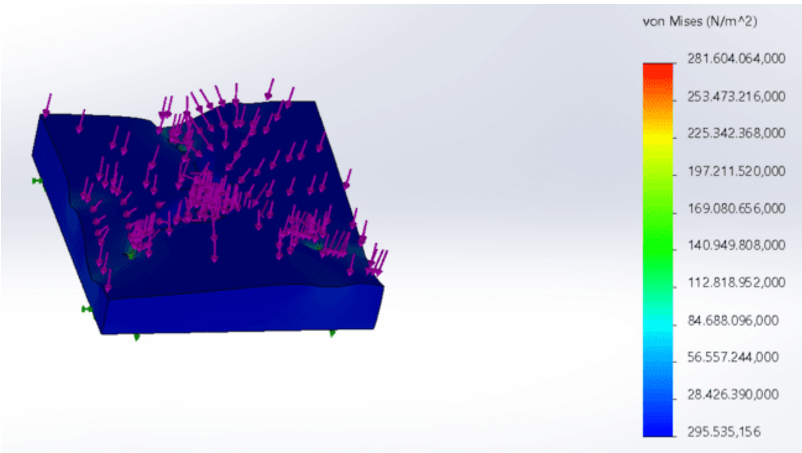

Figure 13. Stress simulation of stainless-steel cast cavity

Based on the force loading simulation results, the table shows the relationship between the magnitude of the force, the type of material, and the resulting stress value. At a force of 10,000 N with the mold material, a stress value of 10,338,960.00 N/m² was obtained. Increasing the force to 15,000 N for the stainless-steel material produced a stress of 15,508,437.00 N/m², while at a force of 20,000 N with steel material, the stress value increased to 20,677,920.00 N/m². These results indicate that an increase in force leads to a significant increase in stress within the material.

Furthermore, force load testing on the core section of the fan blade mold was conducted to analyze the mechanical response of the material to the applied force. The only parameter observed was the equivalent stress (von Mises stress) as a basis for evaluating material performance. The test results are presented in table format to facilitate a comparison of the effects of force variation and material type on the stress distribution.

Tabel 9. Comparison of force core cast alloy steel

Force (N)	Material	Stress Value (N/m^2)
10000		8.559.456
15000	*Cast Alloy Steel*	8.596.270
20000		8.789.744

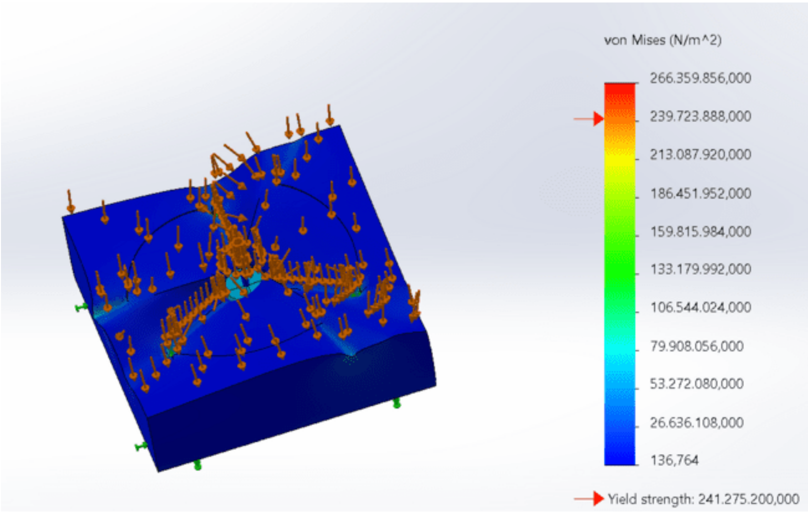

Figure 14. Stress simulation of core cast alloy steel

Based on the force loading simulation results, the table illustrates the relationship between the magnitude of the force, the type of material, and the resulting stress value. At a force of 10,000 N with the mold material in the cavity section, a stress value of 10,338,960.00 N/m² was obtained. Increasing the force to 15,000 N for the stainless-steel material produced a stress of 15,508,437.00 N/m², while at a force of 20,000 N with steel material, the stress value increased to 20,677,920.00 N/m². These results indicate that an increase in force leads to a significant increase in stress within the material.

Tabel 10. Comparison of force core cast carbon steel

Force (N)	Material	Stress Value (N/m^2)
10000		12.839.182
15000	*Cast Carbon Steel*	12.894.407
20000		13.184.608

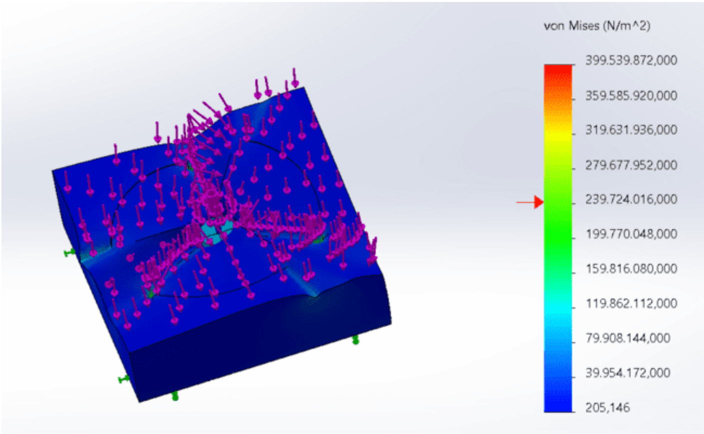

Figure 15. Stress simulation of carbon steel core cast

Based on the force loading simulation results, the table illustrates the relationship between the magnitude of the force, the material type, and the resulting stress values. At a force of 10,000 N with core cast carbon steel material, a stress value of 12,839,182 N/m² was obtained. Increasing the force to 15,000 N for the stainless-steel material produced a stress of 12,894,407 N/m², while at a force of 20,000 N with steel material, the stress value increased to 13,184,608 N/m². These results indicate that an increase in force leads to a significant rise in stress within the material.

Tabel 11. Comparison of force core cast stainless steel

Force (N)	Material	Stress Value (N/m^2)
10000		17.118.912
15000	*Cast Stainless Steel*	17.192.540
20000		17.579.488

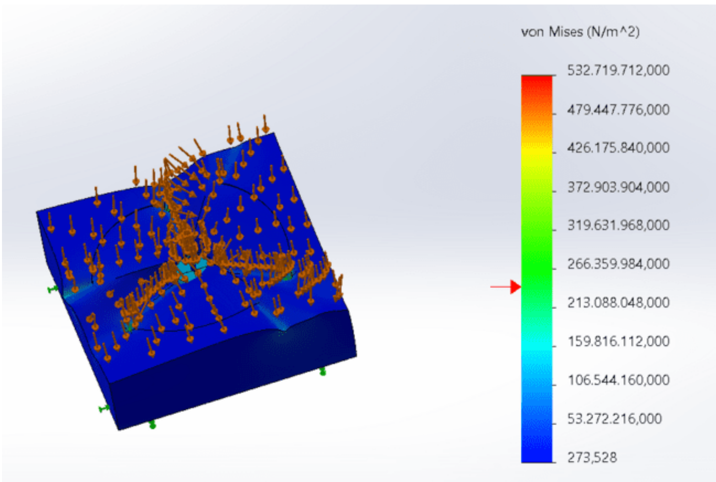

Figure 16. Stress simulation of stainless-steel core cast

Based on the force loading simulation results, the table illustrates the relationship between the magnitude of the force, the material type, and the resulting stress values. At a force of 10,000 N with core cast stainless steel material, a stress value of 17,118,912 N/m² was obtained. Increasing the force to 15,000 N for the stainless-steel material produced a stress of 17,192,540 N/m², while at a force of

20,000 N with steel material, the stress value increased to 17,759,488 N/m². These results indicate that an increase in force leads to a significant rise in stress within the material.

Overall, the simulation results shown in Figures 11 through 16 demonstrate that the color contour distribution in the load-affected areas does not undergo significant pattern changes despite the increase in compressive force. This consistency in color patterns indicates that the location and mechanism of stress concentration on the structure remain the same, specifically at the shaft and fan blade fin areas, such that the structural response to loading is geometrically stable. In other words, the increased load does not trigger new stress areas but only increases the magnitude of stress in the same regions. The linear relationship between the compressive force and the stress values indicates that the material still behaves within its elastic limits and does not experience material or geometric nonlinearity during the tested loading range. This condition reinforces the validity of the simulation results due to the consistently increasing stress response.

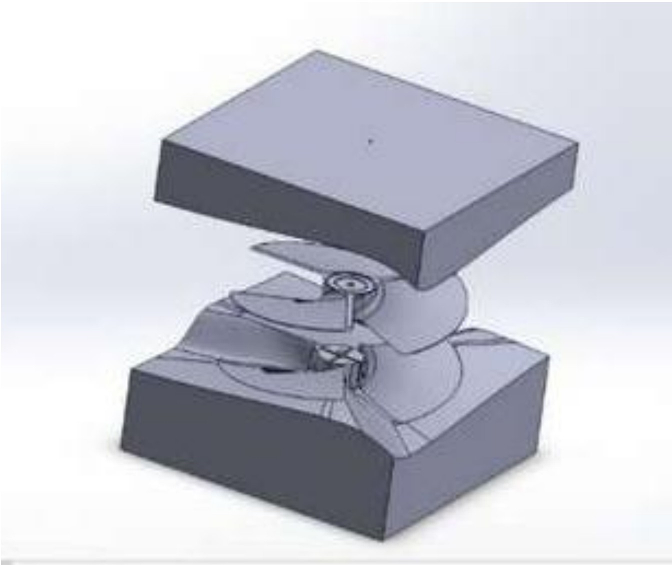

Figure 17. *Modelling mold fan blade*

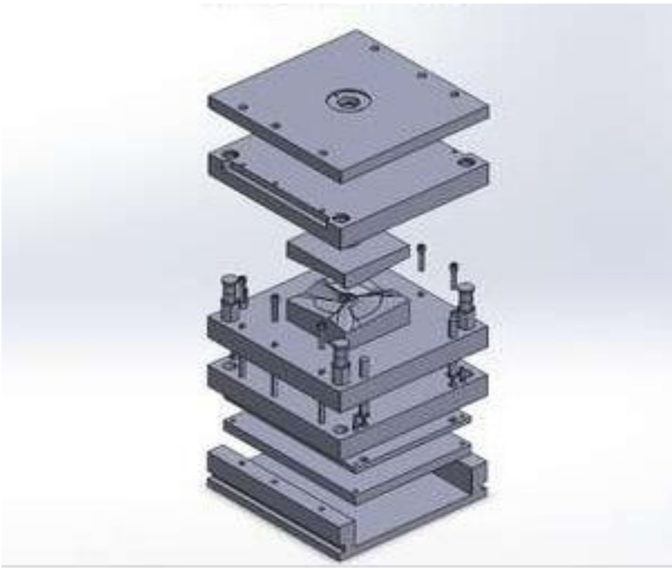

Figure 18. *Modelling injection molding fan blade*

The product design results in this research are visualized through 3D modeling in SolidWorks, displaying the entire product design along with its constituent components. This modeling also includes the mold structure for the injection molding process, providing a complete overview of the shape, dimensions, component arrangement, and critical areas of the mold. The final visualization of the design and mold is used as a guide for the subsequent testing and mechanical analysis stages.

4. Conclusion

This research aims to analyze the mechanical behavior of both the fan blade and the injection mold to evaluate their structural performance prior to the manufacturing stage using a Computer-Aided Design (CAD) and Computer-Aided Engineering (CAE) numerical analysis approach. Based on the mesh study results, an optimal mesh size of 6 mm was obtained, producing stable von Mises stress values with an average error rate of 0.05%. Consequently, it can be concluded that the simulation results are not significantly influenced by variations in mesh element size and exhibit a good level of convergence.

Simulation results for the fan blade indicate that increased torque loading leads to a consistent rise in stress, displacement, and strain values across all analyzed materials. ABS material experienced the largest displacement and strain compared to other materials at the same loading level. This condition indicates that ABS has relatively low structural stiffness, making it more prone to deformation when subjected to torque, whereas PEEK material demonstrated a high elastic modulus, allowing it to maintain its geometric shape much better than the other two materials.

For the fan blade molding simulations, a linear relationship was found between the increase in compressive force and the rise in von Mises stress for all analyzed mold materials: cast alloy steel, cast carbon steel, and cast stainless steel. In the cavity mold, the highest stress value was observed in cast stainless steel, while the lowest was in cast carbon steel, with clear sensitivity in stress changes for each material. In the core section, the highest stress was also found in stainless steel, while the lowest was in cast alloy steel, though the sensitivity to stress changes was not as significant as in the cavity. These results state that the maximum von Mises stress occurs in the core mold, while the cavity experiences lower maximum stress. The high stress in the core is caused by the concentration of the compressive load and its geometric characteristics, making the core the most critical component in the injection mold system.

The implications of this research show that the use of integrated CAD-CAE simulations can serve as an effective decision-making tool in the development of fan blades and injection molds. For future research, it is recommended to add thermal and fluid flow analysis (CFD) to the molding process, as well as experimental validation to enhance the accuracy and utility of the results on an industrial scale.

References

Algarni, M., & Ghazali, S. (2021). Comparative study of the sensitivity of PLA, ABS, PEEK, and PETG's mechanical properties to FDM printing process parameters. *Crystals (Basel)*, 11(8), 995. https://doi.org/10.3390/cryst11080995

Aswanto, T. H., Purwantono, P., Putra, R. P., & Afnison, W. (2023). Analisis variasi jumlah sudu terhadap torsi yang dihasilkan pada turbin vortex. *Asian Journal of Science, Technology, Engineering, and Art*, 1(2), 348–366. https://doi.org/10.58578/ajstea.v1i2.2070

Farotti, E., & Natalini, M. (2018). Injection molding: Influence of process parameters on mechanical properties of polypropylene polymer (A first study). *Procedia Structural Integrity*, 8, 256–264. https://doi.org/10.1016/j.prostr.2017.12.027

Kitayama, S., Tamada, K., Takano, M., & Aiba, S. (2018). Numerical optimization of process parameters in plastic injection molding for minimizing weldlines and clamping force using conformal cooling channel. *Journal of Manufacturing Processes*, 32, 782–790. https://doi.org/10.1016/j.jmapro.2018.04.007

Media, R. I., Adhiharto, R., Erlangga, Y. Y., & Friyadi, M. F. (2022). Analisis parameter cetakan injeksi plastik menggunakan simulasi CAE untuk memprediksi kegagalan produk front cover MiFUS®. *Jurnal Rekayasa Mesin*, 17(2), 241. https://doi.org/10.32497/jrm.v17i2.3210

Mohan, M., Ansari, M. N. M., & Shanks, R. A. (2017). Review on the effects of process parameters on strength, shrinkage, and warpage of injection molding plastic component. *Polymer-Plastics Technology and Engineering*, 56 (1), 1–12. https://doi.org/10.1080/03602559.2015.1132466

Prasetiyo, A. B. (2020). Cooling uniformity analysis of injection molding plastic products with cooling system variations using CFD method. *Jurnal Penelitian Saintek*, 25(2). https://doi.org/10.21831/jps.v25i2.34574

Prasetyo, A., Sihole, A. T. P., Khairunisa, M. D., Najibbulloh, M., & Atmayati, R. (2025). Implementasi software SolidWorks dalam perancangan produksi lemari untuk efisiensi waktu produksi. *Scientific Journal of Reflection: Economic, Accounting, Management and Business*, 8(1), 164-169.

Saefudin, S., et al. (2024). Peningkatan keterampilan menggambar teknik menggunakan software CAD untuk siswa SMK. *Jurnal Pengabdian Masyarakat Teknik*, 6(2), 91–97. https://doi.org/10.24853/jpmt.6.2.91-97

Tujmer, M., & Pilipović, A. (2025). Effect of annealing on tensile properties of carbon fiber reinforced PA 6 manufactured by fused deposition modeling. *European Mechanical Science*, 9(1), 8–15. https://doi.org/10.26701/ems.1595060

Ulaan, T., Poeng, R., Gede, I. N., Arungpadang, T., & Lumintang, R. (2025). Pelatihan computer aided design 3D menggunakan aplikasi SolidWorks untuk SMK Cokroaminoto Kotamobagu. *The Studies of Social Sciences*, 7(1), 78–83. https://doi.org/10.35801/tsss.v7i1.58822

van der Merwe, S. R., Desai, D. A., Snedden, G. C., & Okanigbe, D. O. (2024). Aircraft turbine fan casing and ballistic fan blade impact: Geometry and material. *Heliyon*, 10(2), e24157. https://doi.org/10.1016/j.heliyon.2024.e24157