International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 08 Issue: 04 | April - 2024 SJIF Rating: 8.448 ISSN: 2582-3930

THERMAL ANALYSIS OF A GAS TURBINE ROTAR BLADE

K. Anoosha[1], Ch. Akhil Kumar[2], L. Kesava[3], V. Bharath[4], N. Akhil[5]

[1]Assistant Professor, Department of Mechanical Engineering, GNIT, Hyderabad, Telangana.

[2] Assistant Professor, Department of Mechanical Engineering, GNITC, Hyderabad, Telangana.

[3,4,5,6]UG Scholars Department of Mechanical Engineering, GNIT, Hyderabad, Telangana.

---***---

Abstract - Gas turbines have an important role in electric power generation gasterband technology is used in a variety of configurations for electric power generation. Turbine blades are mainly affected due to static loads also the temperature has significant effect on the gas turbine rotor blades. This paper summarises the design and steady state of thermal analysis of gas turbine router blade on which Solid Works 2020 of software is used for design of solid model of the turbine blade. Ansys 16 software is used for analysis of finite element model generated by meshing of the Blade and design calculation is computed by using MAT lab software .The materials of the gas turbine rotter blade are chosen as copper titanium and nickel.

Key Words : Gas turbine, Turbine rotor blades, SolidWorks 2020, Ansys Workbench

1. INTRODUCTION

At the heart of a gas turbine engine lies a vital workhorse: the rotor blade. These wing-shaped airfoils lining the edge of a spinning disc play a critical role. They are meticulously designed to capture the energy from the engine's scorching hot, high-pressure gas. As this gas streams past the blades, it exerts a pushing force, causing the disc and shaft to spin. This rotation is the engine's output power, used for propulsion in aircraft or electricity generation. The efficiency of the entire gas turbine hinges on the effectiveness of these blades in extracting energy. Operating in an extreme environment, rotor blades endure intense heat, centrifugal forces from high speeds, and constant stress from the gas flow. These demanding conditions necessitate the use of advanced materials and sophisticated cooling technologies to ensure the blades can withstand this harsh environment.

2. Introduction to Solid Works

This software suite empowers engineers and designers across industries to bring their ideas to life. It offers a user-friendly interface packed with robust features, encompassing all aspects of product development from creation to analysis and management. At its core, SOLIDWORKS excels in building detailed 3D models, fostering a virtual environment to explore form and functionality before physical prototypes are built. Beyond modeling, it allows for analysis of a design's behavior under various conditions, aiding in optimization for performance and efficiency. Furthermore, SOLIDWORKS facilitates seamless collaboration throughout the design cycle, enabling clear communication and streamlined decision-making between designers, engineers, and other stakeholders involved in the process.

SOLIDWORKS is a computer-aided design (CAD) software program that allows you to create 3D models of physical objects. It is used by engineers, designers, and other professionals to create everything from simple parts to complex assemblies

Figure 1: SOLID WORKS

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 08 Issue: 04 | April - 2024 SJIF Rating: 8.448 ISSN: 2582-3930

3. Introduction to ANSYS Work Bench

Ansys, a software suite synonymous with engineering simulation, empowers engineers across disciplines to virtually analyze products before they're built. Imagine a digital testing lab to examine designs under extreme conditions, simulating stress on buildings or airflow around wings. Ansys enables virtual prototyping and testing, optimizing performance by identifying areas for improvement, ultimately reducing risk and cost by mitigating potential problems early in the design phase.

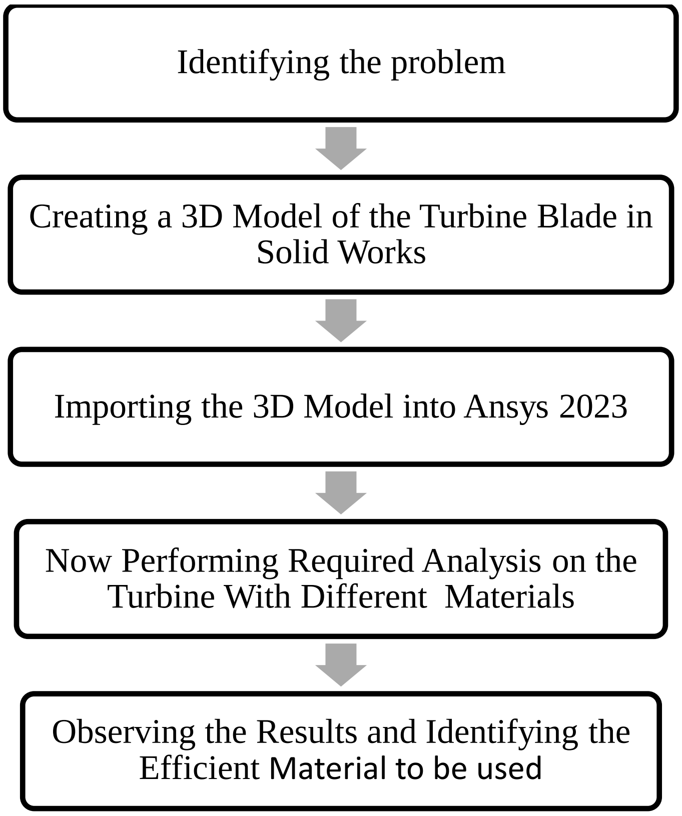
ANSYS WORKBENCH, developed by ANSYS INC., USA, is a computer aided finite element modelling and finite element analysis tool (CAFEM AND CAFEA). In the graphical user interface GUI of ansys workbench the user can generate 3-dimensional and FEA models, perform analysis and generate results of analysis. We can perform a variety of tasks ranging from design assessment to finite element analysis to complete product optimisation analysis by using ANSYS WORKBENCH. ANSYS also enable the combination of standalone analysis system into a project and to manage the project workflow.

In ansys workbench this are the list of analysis can be determined:

- Modal analysis
- Static structural analysis
- Transient structural analysis

Figure 2: ANSYS Work Bench

4. Methodology



5. Process

5.1 Create a new part and select the Front plane.

- To create a new part, click the File menu and select New. Then, select Part and click OK.
- To select the Front plane, click the Planes menu and select Front.

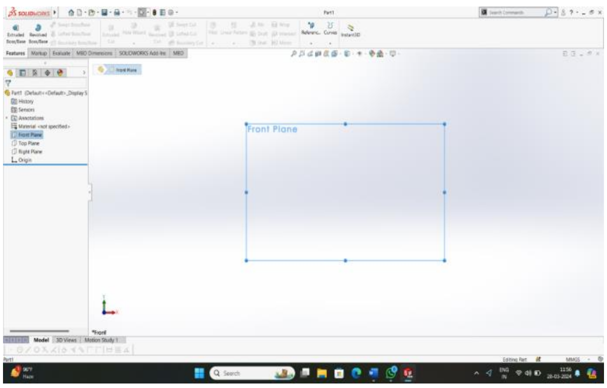


Figure 3.1 Selecting a Plane in SolidWorks

5.2 Creating a Rectangular block

- Select the rectangle tool.
- Give the dimensions as per the picture given below.

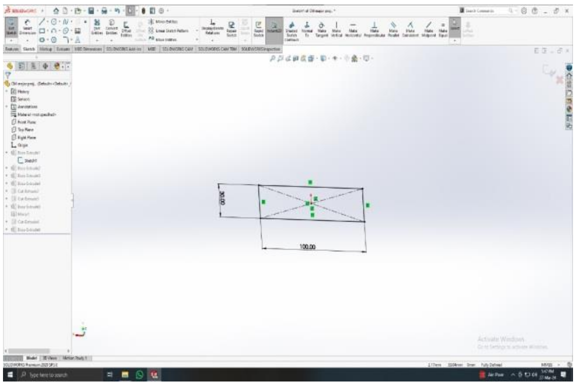


Figure 3.2 Creating a Rectangular block

5.3 Creating the rectangle on top plane

- Now, select the top plane
- Create the rectangle on the top surface of the extruded rectangular block.
- After creating on the top surface of rectangular block click on enter.

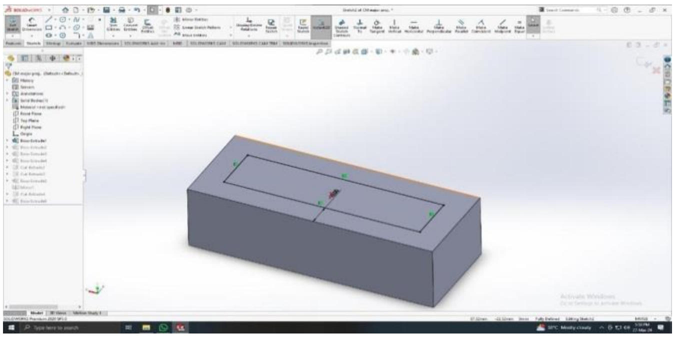


Figure 3.3 Creating the rectangle on top plane

5.4 Creating the rectangle on the top surface of the rectangular block

- Now, select the top plane
- Create the rectangle on the top surface of the extruded rectangular block.
- After creating on the top surface of rectangular block click on enter.
- Rectangle block has been created on the top surface of the rectangular block

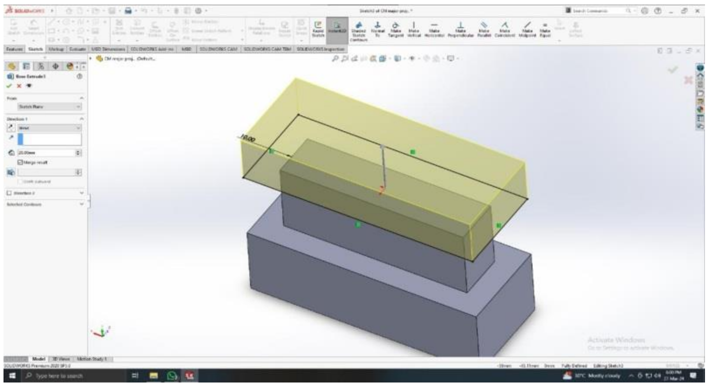


Figure 3.4 Creating the rectangle on the top surface of the rectangular block

5.5 Creating a hilt

- Create a small rectangle at the right side of the bottom rectangular block.
- Then select lofted extrude command from the features.
- Now select the rectangle to be extruded and give it’s dimensions as required.

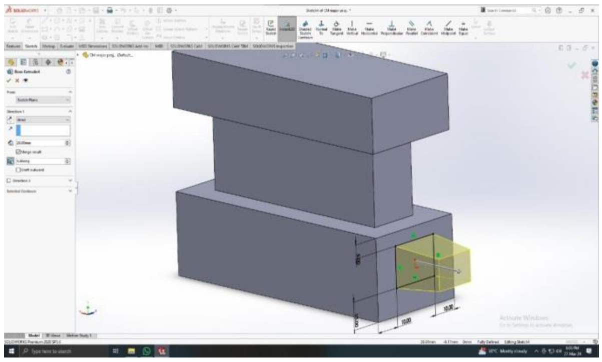


Figure 3.5 Creating a hilt

5.6 Creating of the fins

- Select the front plane and click on enter
- Create a small rectangular blocks on the bottom rectangular block
- Select the features and select the cut extrude command
- Extrude the small rectangular blocks as per the given dimensions
- After that select the mirror command and select the object to be mirror

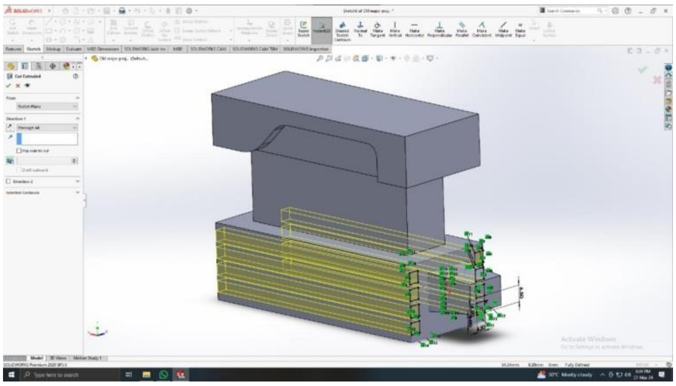


Figure 3.6 Creating of the fins

5.7 Creating the blade profile

- Select the top plane
- Now on the top surface of the rectangular block
- Select the arc command and draw the arc as per the given dimensions
- Select the extrude command click on enter
- Select the blade profile and extrude the blade profile

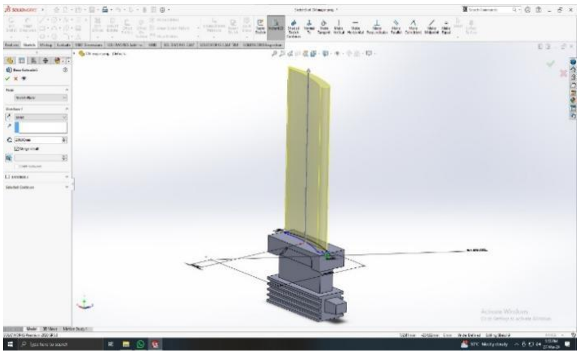


Figure 3.7 Creating the blade profile

6 Process of ANSYS Work Bench

6.1 Main windows of a Ansys Workbench

- Open the ansys software work bench student version
- After opening of ansys work bench click on new file
- In the new file it will open ansys systems
- After that click on linear budding
- Click on geometry

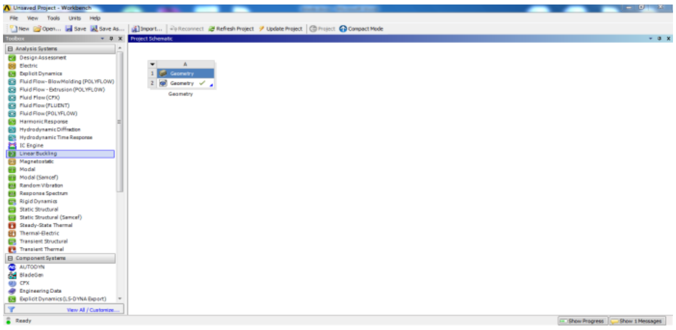


Figure 3.8 Main windows of ansys workbench

6.2 Selection of a Material Properties

- Click on the tool box
- Now we can see the properties of an ansys
- In the properties select the physical properties
- It shows the engineering data the material application will be done by the add symbol in the general materials
- In this we are using the aluminium alloy

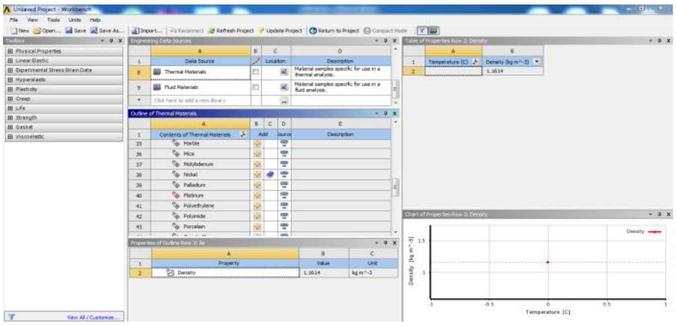


Figure 3.8 Selection of a Material Properties

6.3 Generation of mesh Geometry

- From the outline tab select the geometry part
- From the bottom window material
- Select the required model
- To generate the mesh in the fixed support right click on the analysis settings and insert the fixed support

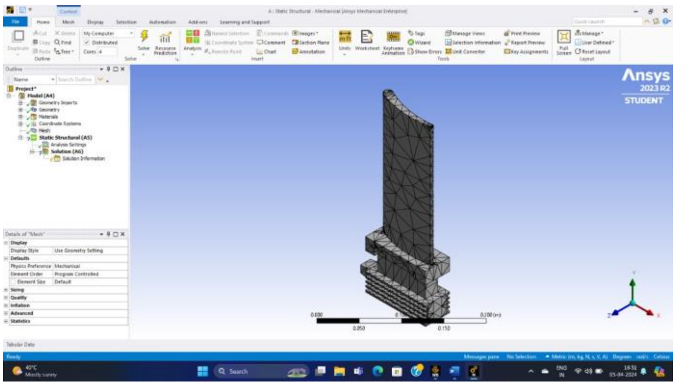


Figure 3.9 Generation of mesh Geometry

Volume: 08 Issue: 04 | April - 2024 SJIF Rating: 8.448 ISSN: 2582-3930

6.4 applying and fixing of the Component

- To apply the moment right click on the analysis settings and insert the moment
- Select the required locations and apply

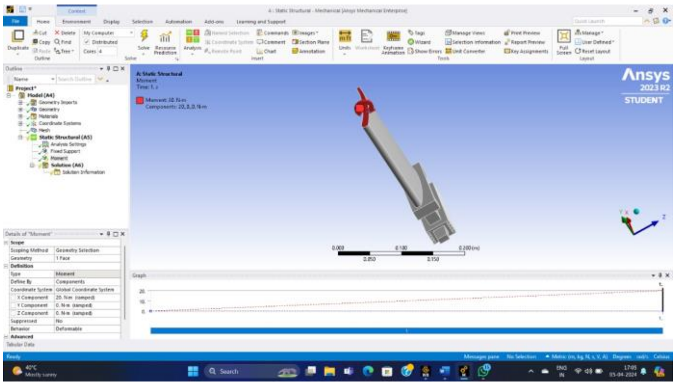


Figure 3.10 Moment

6.5 Applying of the Different Moments

- **AT THE MOMENT 10Nm**

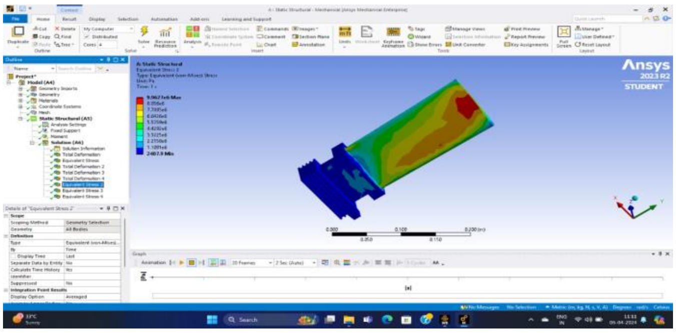


Figure 3.11 Equivalent stress at the 10Nm

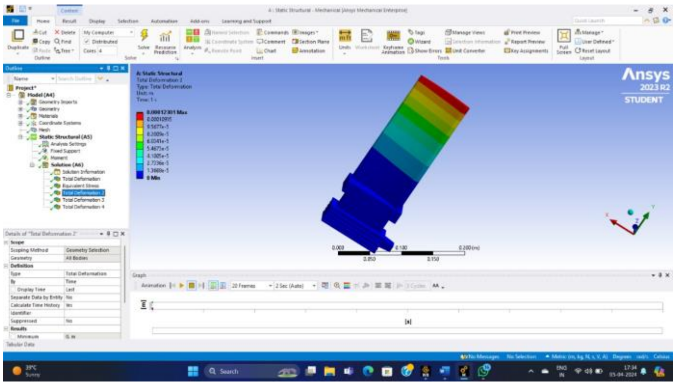


Figure 3.12 Total Deformation at the 10Nm

Equivalent stress

The stress obtained is $9.9627*10^7$pa for Al alloy which is very low as compared to the yield strength of material

Total deformation

The deformation of the body is very low as compared to the body size, so it will not be taken into the consideration

- **AT THE MOMENT 20Nm**

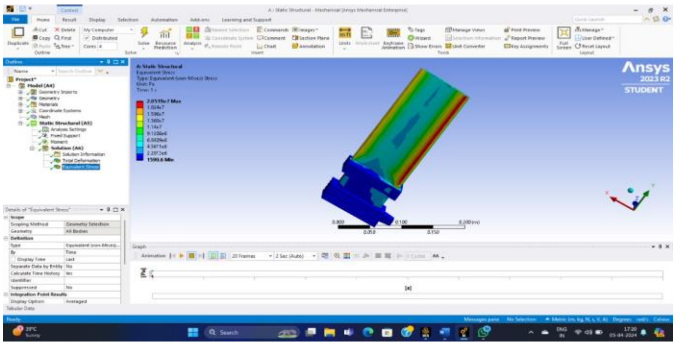


Figure 3.13 Equivalent Stress at the 20Nm

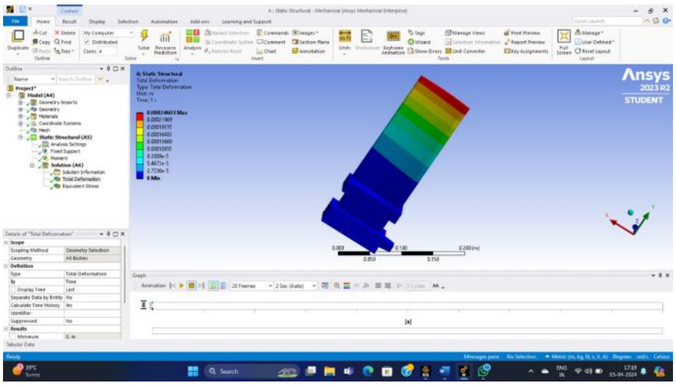


Figure 3.14 Total Deformation at the 20Nm

STEPS IN THE TRANSCIENT THERMAL ANALYSIS

6.6 Main windows of a Ansys Workbench

- Open the ansys software work bench student version
- After opening of ansys work bench click on new file
- In the new file it will open ansys systems
- After that click on linear budding
- Click on geometry

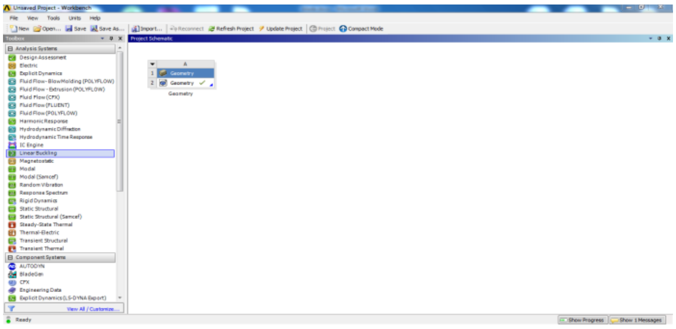


Figure 3.15 Main windows of ansys workbench

6.7 Selection of a Material Properties

- Click on the tool box
- Now we can see the properties of an ansys
- In the properties select the physical properties
- It shows the engineering data the material application will be done by the add symbol in the general materials
- In this we are using the aluminium alloy

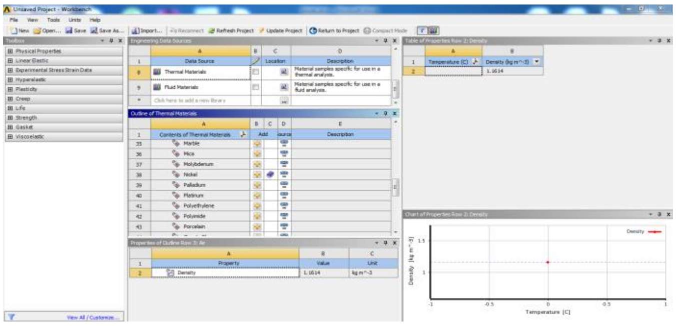


Figure 3.16 Selection of a Material Properties

6.8 Generation of mesh Geometry

- From the outline tab select the geometry part
- From the bottom window material
- Select the required model
- To generate the mesh in the fixed support right click on the analysis settings and insert the fixed support

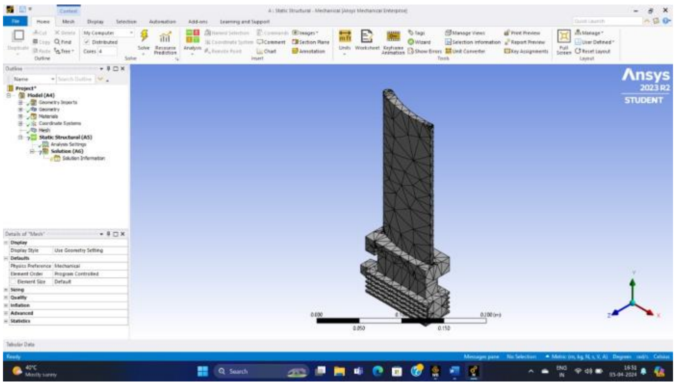


Figure 3.17 Generation of mesh Geometry

6.9 Applying the Temperatures and HEAT FLOW

To apply Temperature the component: Right click on the Analysis Settings – Insert – temperature Support – Select all the bushes or bearings location and apply, Now enter the Temperature value's ºC.

Definition	
Type	Temperature
Magnitude	Tabular Data
Suppressed	No
Tabular Data	
Independent Variable	Time

	Steps	Time [s]	Temperature [°C]
1	1	0.	0.
2	1	1.	100.
3	2	2.	200.
4	3	3.	300.
5	4	4.	400.
6	5	5.	500.
*			

Figure 3.18 Temperatures

To Apply the heat flow: Right click on Analysis settings – Insert – heat flow – Select the required locations as shown below and apply – Now enter the heat flow value:10 w/mm^2 ºC.

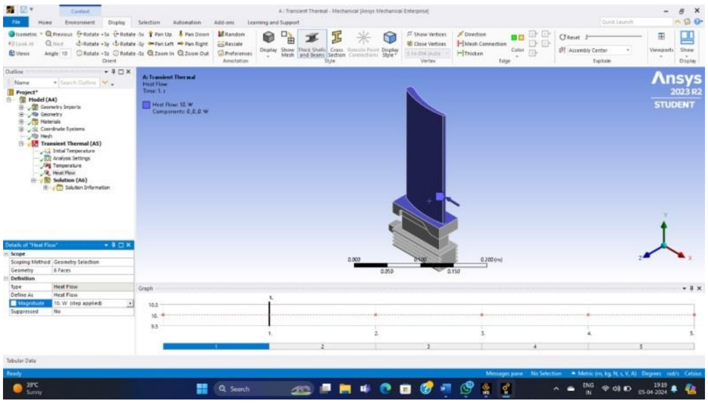


Figure 3.19 Heat Flow

- **AT THE TEMPERATURE 100 ºC**

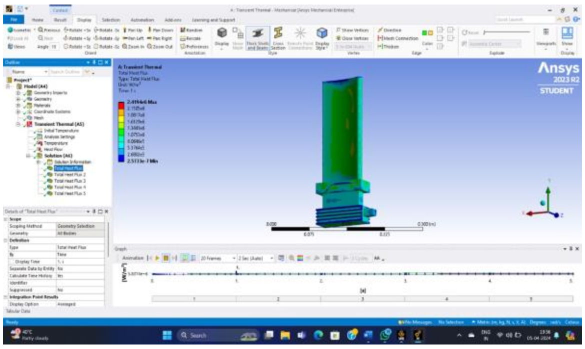


Figure 3.20 AT THE TEMPERATURE 100 ºC

The Heat flux obtained is 2.419*10^{6}w/m^2 for Al alloy

- **AT THE TEMPERATURE 200 ºC**

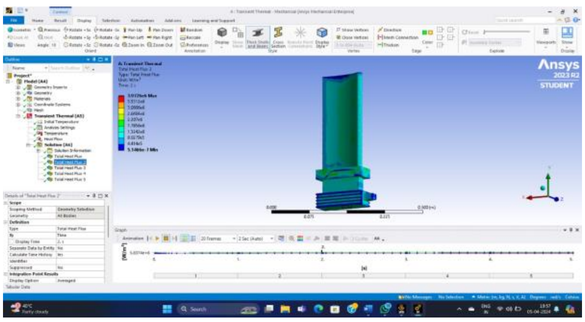


Figure 3.21 AT THE TEMPERATURE 200 ºC

The Heat flux obtained is 3.9726*10^{6}w/m^2 for Al alloy.

7. Results

GRAPH FORMAT :

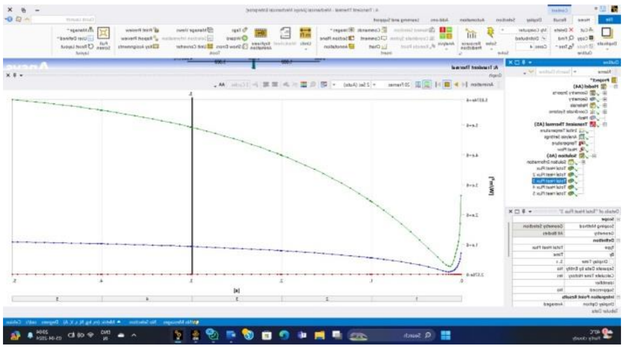


Figure 3.21 Graph Format

STATIC STRUCTURAL ANALYSIS

S.NO	MOMENT (N-m)	STRESS(N/m2)	DEFORMATION(m)
1	10	$9.9627*10^6$	0.00012301
2	20	$2.0519*10^7$	0.00024603
3	30	$3.0779*10^7$	0.00036904
4	40	$3.985*10^7$	0.00028697

Table 1: Statistical structural analysis

The table appears to be a sample of a structural analysis output, possibly for a beam subjected to increasing moments. There are moment (N-m), stress (N/m²), and deformation (m). Looking at the data, we can see that the moment, stress, and deformation all increase as the serial number increases. This suggests that the structure is likely experiencing increasing load. The stress values are in the millions of N/m² which is typical for structural analysis results. Without additional information about the specific material and geometry of the structure, basic statistical analysis on the data you provided.

Here's a summary of the statistical analysis of the data:

The average moment applied is 25 N-m.

The average stress is 2.53 x 10^7 N/m².

The average deformation is 2.56 x 10^-4 m.

TRANSIENT THERMAL ANALYSIS

S.NO	time (s)	Temperature [°C]	heat flow w/mm² °C	heat flux
1	0	0	10	0
2	1	100	10	$2.419*10^6$W/m²
3	2	200	10	$3.9726*10^6$w/m²
4	3	300	10	$4.9053*10^6$w/m²
5	4	400	10	$5.4799*10^6$w/m²
6	5	500	10	$58374*10^6$w/m²

Table 2: Transient thermal analysis

In the conducted transient thermal analysis, the temperature of the system varied over time. Initially, at t = 0 seconds, the temperature was 0°C. As time progressed, the temperature increased, reaching 100°C at t = 1 second, 200°C at t = 2 seconds, 300°C at t = 3 seconds, 400°C at t = 4 seconds, and finally, 500°C at t = 5 seconds. The heat flow remained constant at 10 W/mm²°C throughout the analysis. Notably, the heat flux exhibited a substantial increase, reaching 58,374 x 10^6 W/m² at the final time point. These findings provide valuable insights into the transient behavior of the system under study

8. Conclusion

In this project, i here create or design or modeling of turbine blade block by using one of the most advanced 3dimensional software mostly known as Solid Works software. By catia software I designed v12 engine block by using different type of tools and feature can be seen in modeling of turbine blade . Later the file is saved in the format as a STP or IGES file to do analysis on the component. The analysis is done by ansys software one of the most practical meshing accurate analysis software to find out the results over the component. By using ansys software I here declared the rate of increase the temperature in turbine will increase the heat flux in turbine blade.

10. REFERENCES

1. N. K. Sahu and J. Kale, “A Review of Gas Turbine Rotor Blade Design,” IJARIIE, vol. 2, no. 4, pp. 964-966, 2018.

2. S.W. Chang, P.-S. Wu, T.-Y. Wan, and W.-L. Cai, “A Review of Cooling Studies on Gas Turbine Rotor Blades with Rotation,” Inventions, vol. 8, no. 1, Art. no. 21, Jan. 2023

3. T. Puja Priyanka, P. Poorna Mohan, K. Subba Rao, et al., “Gas Turbine Blade Design and Analysis,” IJSRD - International Journal for Scientific Research & Development, vol. 5, no. 07, pp. 53, Jan. 2017.

4. T. S. Chowdhury, F. T. Mohsin, M. M. Tonni, M. N. Hasan Mita, M. M. Ehsan, “A critical review on gas turbine cooling performance and failure analysis of turbine blades,” International Journal of Thermofluids, vol. 18, May 2023

5. A. A. Jabbar, A. K. Rai, P. R. Reddy, and M. H. Dakhil, “Design and Analysis of Gas Turbine Rotor Blade Using

Finite Element Method," Journal of Mechanical Engineering, vol. 4, no. 1, pp. 91-112, Feb. 2014.

6. Singh and A. C. Tiwari, "A revisit to different techniques for gas turbine blade cooling," Mater1. Today: Proc., in press, corrected proof, Available online 28 January 2023, doi: 10.1016/j.matpr.2023.01.2172.

7. A. Dong, P. Yan, X. Qian, W. Han, and Q. Wang, "Rotation Effect on Flow and Heat Transfer for HighTemperature Rotor Blade in a Heavy Gas Turbine," J. Therm. Sci., vol. 30, pp. 707-715, Mar. 2021.

8. W. Wiberg and N. Anton, "Aerodynamic design of a gas turbine rotor blade for the KTH test turbine," Division of Thermal Power Engineering, Department of Energy Sciences, Lund Institute of Technology, Lund, Sweden, 2013.

9. J. Town, D. Straub, J. Black, K. A. Thole, and T. I-P. Shih, "State-of-the-Art Cooling Technology for a Turbine Rotor Blade," J. Turbomach., vol. 140, no. 7, Art. no. 071007, Jul. 2018

10. G. Karadimas, K. Georgarakis, and K. Salonitis, "Mechanical and Thermal Performance of Ceramic and Nickel Superalloys Composites for Gas Turbine Blade Applications," in Advances in Manufacturing Technology XXXVI, Nov. 2023

11. A. S. M. Ashique Mahmood, "Literature Survey on Topic Modeling," 2013.

12. G. Gilardi, I. Sharf, "Literature survey of contact dynamics modelling," *Mechanism and Machine Theory*, vol. 37, no. 10, pp. 1213-1239, 2002, ISSN 0094-114X.

13. K. Teffah, Y. Zhang, X.-L. Mou, "Modeling and experimentation of new thermoelectric cooler– thermoelectric generator module," *Energies*, vol. 11, no. 3, p. 576, 2018.

14. H. Mickoski, I. Mickoski, M. Djidrov, "Dynamic modeling and simulation of three-member robot manipulator," *Mathematical Models in Engineering*, vol. 4, 2018, doi: 10.21595/mme.2018.20319.

15. R. Geng, C. Liang, Z. Yao, J. Huang, "Mechanical modeling and analysis of V-Shaped LUM mechanical drift," *Journal of Physics: Conference Series*, vol. 1654, no. 1, p. 01208, 2020.

16. F. D. Gurmesa, H. G. Lemu, "Literature Review on Thermomechanical Modelling and Analysis of Residual Stress Effects in Wire Arc Additive Manufacturing," *Metals*, vol. 13, no. 3, p. 526, 2023.

17. N. Gardner, H. Khan, C.-C. Hung, "Definition modeling: literature review and dataset analysis," *Applied Computing and Intelligence*, vol. 2, no. 1, pp. 83-98, 2022.

18. L. Kwasniewski, B. A. Izzuddin, M. Pereira, C. Bucur, M. Gizejowski, "Modelling and analysis," in: *Joint Workshop of COST Actions TU0601 and E55*, Ljubljana, Slovenia, 2009, pp. 91-102.

19. G. Nagpal, U. Chanda, H. Seth, "The First Half-Century of Decision Modelling for Substitutable Products: A Literature Review and Bibliographic Analysis," *Operations and Supply Chain Management*, vol. 14, no. 3, pp. 261-276, 2021.

20. M. Carraturo et al., "Two-level method part-scale thermal analysis of laser powder bed fusion additive manufacturing," *Engineering with Computers*, vol. 38, no. 6, pp. 4815-4828, 2022.

21. N. Kumar, M. Thrivendra Mamidi, "Design Optimization of Propeller Blade Subjected to Different Loading Conditions," *International Journal of Research & Technology*, vol. 3, no. 11, pp. 26-37, Nov. 2019.

22. M. Aleshin, "On Structural Optimization of the Propeller Blade," *International Journal of Engineering & Technology*, vol. 7, no. 4.36, pp. 1203-1207, 2018