International Journal of Engineering Sciences Paradigms and Researches (IJESPR)
Volume 55, Issue 02 and Publishing Date: 18th June, 2026
An Indexed, Referred and Peer Reviewed Journal
ISSN (Online): 2319-6564
www.ijesonline.com

Application of ANSYS in the Dynamic Analysis of a Hollow Turbine Blade

Balla Mohamed Ahmed Elimam[1] and Malaz Safieldin Bashir Altayp[2]

[1]Assistant Professor, Department of Mechanical Engineering, Karary University, Khartoum, Sudan
ballasd@yahoo.com
[2]Lec, Department of Engineering, National University, Khartoum, Sudan
malazSafieldin@nu.edu.sd

Abstract

This study presents a dynamic finite element analysis of a hollow turbine blade, conducted using the commercial software ANSYS. The methodology is grounded in the finite element method, wherein displacement fields are approximated via power series expansions, and element mass and stiffness matrices are evaluated through numerical integration.

The investigation was motivated by a failure incident observed in Unit 2 of the Khartoum North Gas Turbine Power Station. To accurately represent the complex aerofoil geometry of the blade, four-node structural shell elements were employed in the finite element model.

Under operational conditions corresponding to a rotational speed of 4980 rpm, the blade's stress distribution, strain response, and deflection profile were computed. Furthermore, a modal analysis was performed to determine the first five natural frequencies, and the associated mode shapes were visualized to characterize the blade's vibrational behavior.

Keywords: *Under steady-state operating conditions, the internal stresses in a turbine blade may be decomposed into steady*

1. Introduction

Turbomachinery blades constitute among the most critical—and costly—components in rotating machinery systems. A typical turbine blade may be idealized as a cantilevered beam with an asymmetric cross-section, fixed at the root and pre-twisted along its span from root to tip. These blades are mounted on a rotating disc at staggered angular positions such that the direction of pre-twist opposes the skew angle of the blade row. This geometric arrangement introduces coupling between bending modes in orthogonal planes [1]. Moreover, the asymmetry of the cross-section induces coupling between bending and torsional deformation modes.

Accurate prediction of the blade's deflected shape under operational loading is essential for effective design. Excessive untwisting can significantly alter the velocity triangles of the working fluid and modify leading- and trailing-edge tip clearances, both of which adversely affect turbomachine efficiency and performance. Under steady-state operating conditions, the internal stresses in a turbine blade may be decomposed into steady (mean) and vibratory components. The former are primarily induced by centrifugal forces due to rotation.

1.1 Traditional Design Criteria

Load-bearing structures in aircraft gas turbine engines operate under extreme thermomechanical environments and must satisfy stringent requirements for reliability, durability, lightweight construction, and high performance. Historically, the aerospace industry has adopted "failure-free lifetime" design philosophies for such critical components.

The conventional design process for turbine blades, illustrated schematically in Fig. 1, typically involves two primary analytical stages: (i) structural dynamic analysis to determine natural frequencies and mode shapes over the operational speed range, and (ii) stress analysis to evaluate the spatial distribution of dynamic stresses resulting from vibratory excitations. The objective is to identify locations of maximum vibratory stress under prescribed excitation conditions, thereby enabling estimation of peak stress amplitudes associated with each vibration mode [2].

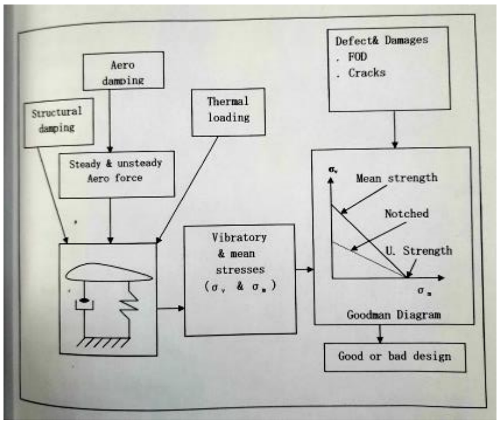

Fig. 1 Conventional turbine blade design process [2].

These design criteria are generally based on deterministic analytical methods that do not explicitly account for material degradation, statistical variability in experimental data, prior operational experience, or uncertainties inherent in real-world service conditions. Consequently, despite compliance with established standards, numerous in-service and developmental failures have been documented.

1.2 Special Features of Hollow Turbine Blades

Modern high-performance turbine blades—particularly those employed in aircraft gas turbines—are frequently fabricated with hollow internal cavities. This design facilitates internal cooling airflow, thereby enhancing thermal management while simultaneously reducing the rotational inertia of the rotor assembly.

Hot-section components in gas turbines are subjected to severe thermomechanical loading. For example, turbine blades routinely operate in gas streams exceeding 3000 °C. The progressive elevation of turbine inlet temperatures over recent decades has been enabled by advances in high-temperature materials and active cooling technologies [3]. Contemporary blades commonly employ convection or film cooling strategies to mitigate thermal gradients and preserve structural integrity.

Material selection for such applications often adheres to standards including ISO DO 4261 and ASTM D2880. Nickel-based superalloys—such as IN738 and IN739—are widely used due to their excellent high-temperature strength and oxidation resistance, attributed in part to their elevated chromium content. Mechanical properties may be further enhanced through post-fabrication processes such as hot isostatic pressing (HIP).

An alternative approach to accommodating higher turbine inlet temperatures involves the application of thermal barrier coatings (TBCs), typically deposited via plasma spray techniques. Nevertheless, turbine blades remain susceptible to thermally induced stresses and chemical degradation caused by fuel contaminants. Protective coatings—applied through galvanic deposition or plasma spraying—are employed to mitigate oxidation and corrosion [5].

1.3 Application of the Finite Element Method in Blade Design

Among the most influential computational methodologies developed for mechanical and structural analysis in recent decades is the finite element method (FEM), particularly as applied to plate and shell elements. Its versatility spans civil, naval, aeronautical, and aerospace engineering disciplines. Knowledge of the free vibration characteristics of plate-like structures enables engineers to optimize designs for enhanced performance and reduced mass. Consequently, the dynamic behavior of rectangular plates has remained a subject of sustained research for over a century [9].

The foundational theory of plate mechanics was first formulated by Euler in the 18th century. Lagrange later derived the first correct differential equation governing the free transverse vibration of thin plates [5].

The generality and adaptability of FEM render it a powerful tool for a wide range of solid and structural mechanics problems. Numerous commercial software packages implement this method, many of which are sufficiently generalized to address multidisciplinary engineering challenges with minimal modification. Two prominent examples—ANSYS and NASTRAN (National Aeronautics and Space Administration Structural Analysis)—

are extensively employed in industry and academia for complex structural analyses.

A comparative overview of selected capabilities of these software packages is provided in Table 1.

Table 1: Capabilities of selected finite element software packages

| Analysis Type	ANSYS	NASTRAN
Nonlinear continua	✓	✓
Ship structures	✓	✓
Plastic Analysis	✓	
Thermal stress and creep	✓	✓
Pipe system	✓	✓
Thick shells	✓	✓
Viscoelastic analysis	✓	
Transient analysis	✓	✓
structural members and mechanical elements	✓	✓
Thin shells		✓
Composite materials		✓
Fracture mechanics	✓	
Aero elasticity	✓	✓
Structural stability	✓	

Note: A checkmark (✓) denotes supported capability.

1.4 Objective of the Study

This work presents a dynamic finite element analysis of a hollow turbine blade using the commercial software ANSYS. The blade geometry is discretized using four-node shell elements. Governing equations for isotropic rectangular plate elements are formulated, and displacement fields are approximated via power series expansions. Based on these approximations, consistent mass and stiffness matrices are assembled, enabling the determination of the blade's free vibration characteristics—including the first five natural frequencies and corresponding mode shapes.

2. ANSYS modeling

2.1 The Finite Element Model

One of the most significant advances in solving mechanics problems in recent years has been the development of finite element computer programs. Among these, ANSYS stands out as one of the most powerful and widely recognized tools. It is particularly well-suited for turbine blade analysis due to its ability to handle dynamic problems with many degrees of freedom and its capacity to account for the stiffening effect of centrifugal forces.

This section presents the application of ANSYS for analyzing a hollow turbine blade. The analysis involves calculating stresses, displacements, and strains resulting from centrifugal force, vibration, gas pressure, and thermal effects. The detailed steps for applying the ANSYS program to this specific analysis are outlined in Appendix B.

The case under consideration is the first-row blades of the General Electric (GE) Model Series 5001 gas turbine (models N, M, & R), installed at the Khartoum North Gas Turbine Station. A photograph of these blades is shown in Figure 2. The airfoil geometry measures 95 mm in height, 63 mm in width, 33 mm in depth, and has a wall thickness of 1 mm. ANSYS was used to model this geometry, as depicted in Figure 3. The preprocessor's mesh generator was employed to create the finite element model using "Structural Shell Elastic 4 Node 63" elements, which are rectangular in shape. These elements were connected at 3,352 nodes.

Based on the information above, the final model, shown in Figure 4 & 5, consists of 3,276 elements and 3,352 nodes.

Fig. 2 First row turbine blade Khartoum North Gas Turbine

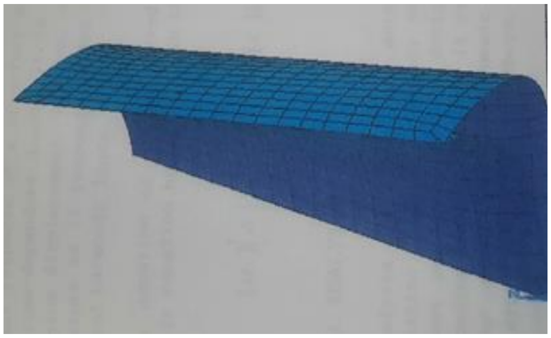

Fig. 3 Turbine blade geometry

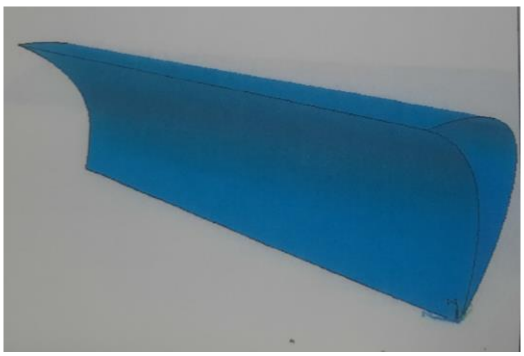

Fig. 4 Finite element model

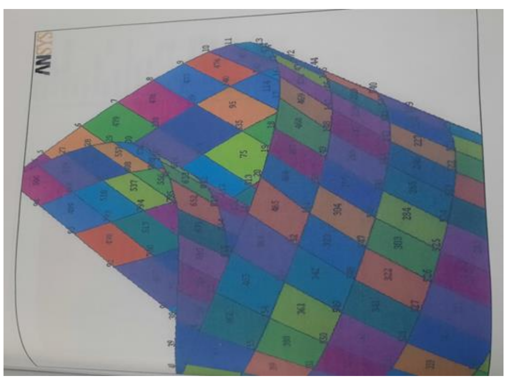

Fig. 5 Partial view of blade showing element and node of finite element model

2.2 Dynamic Analysis

Transient dynamic analysis is a technique used to determine the dynamic response of a structure subjected to general time-dependent loads. It is employed to calculate the time-varying displacements, strains, stresses, and forces within a structure as it responds to any combination of static, dynamic, or harmonic loads.

The fundamental equation of motion solved in transient dynamic analysis:

$$[M]\{\ddot{\delta}\} + [C]\{\dot{\delta}\} + [K]\{\delta\} = \{F_t\}$$

Where M, C, and K represent the mass, damping, and stiffness matrices, respectively; δ is the displacement vector; and F is the time-dependent force vector.

2.3 Modal Analysis

Modal analysis is used to determine the vibration characteristics of a structure—specifically, its natural frequencies and mode shapes. This information is a critical parameter in structural design, especially under dynamic loading conditions. Furthermore, modal analysis often serves as a foundational step for more detailed dynamic analyses, such as transient dynamic analysis.

2.4 Boundary Conditions

The boundary conditions for this model reflect the physical constraints of the turbine blade. Specifically, the nodes along the bottom row of the model (representing the blade root) are fixed, as the blade is clamped at its base. All other grid nodes in the model are permitted six degrees of freedom.

The following table summarizes the actual data input for the first-row blades of the gas turbine installed at the Khartoum North Gas Turbine Station, Unit 2, including the specified boundary conditions.

Table 2: Data Input for ANSYS Simulation

Description	Data Input
Preference discipline	Structural & h-method
Element type	Structural shell elastic 4 node 63
Thickness of plate	0.001 m
Material Properties	
| Young's modulus	207 GN/m²
Density	7850 kg/m³
Poisson's ratio	0.3
Key Points	

(Coordinates)			
No. Of key point	X	Y	Z
1	0.000	0.000	0.000
2	0.031	0.033	0.000
3	0.063	0.000	0.000
4	0.032	0.015	0.000
5	0.000	0.026	0.095
6	0.020	0.040	0.095
7	0.063	0.003	0.095
8	0.020	0.029	0.095
Mesh tool	Smart size 5 fine		
Fixed at node	2, 4, 41–51 and 62–74 free in x-axis		
Pressure on concave face	57,285 N/m²		
Temperature	980 °C		
Angular Speed	521.5 rad/s		
Diameter	0.3183 m		
Time step	Max. 36,000 s		

3. Result and Discussion

The following results were obtained:

Airfoil displacements resulting from the input data (as detailed in Table 2) are illustrated in Figure 6. The maximum displacement recorded is 1.569 mm at node 8, as shown in Figure 5 Additionally, the maximum untwisting angle at the airfoil tip is 2.054 degrees. This displacement and untwisting behavior represent critical design considerations for long, thin airfoils.

When the operating temperature increases to 1500°C, the displacement rises to 3.152 mm, as depicted in Figure 7. Although the blade tip clearance shown in Figure 8 is 3.232 mm, thermal expansion reduces this clearance to just 0.05 mm. Given that the turbine rotor expands more than the fixed casing and the recorded rotational speed exceeds 4980 rev/min, this condition may lead to blade tips rubbing against the casing.

A reported failure occurred at Khartoum North Gas Turbine Unit 2, involving blade tips rubbing against the fixed casing in the first row. This incident was attributed to overheating and necessitated blade replacement. The findings presented above may provide a plausible explanation for the cause of this failure.

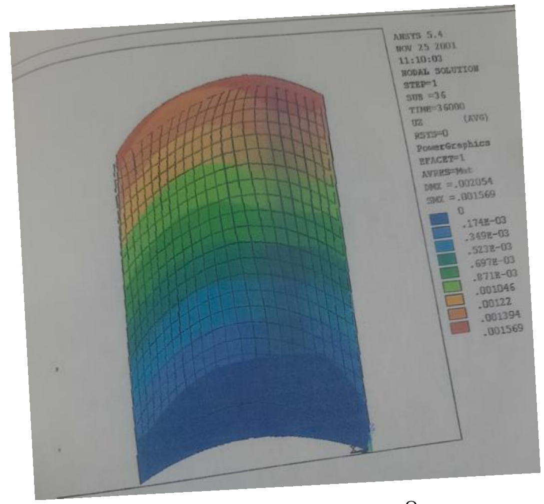

Fig. 6 Displacement at 980 ^{O}C

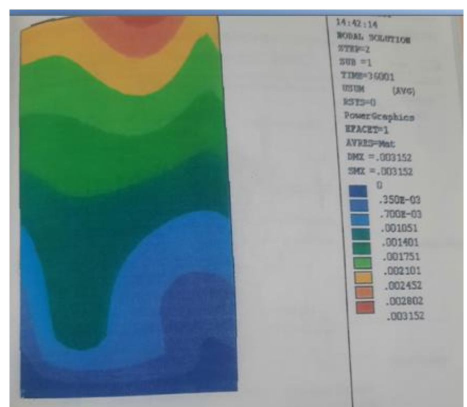

Fig. 7 Displacement at 1500 ^{O}C

International Journal of Engineering Sciences Paradigms and Researches (IJESPR)
Volume 55, Issue 02 and Publishing Date: 18th June, 2026
An Indexed, Referred and Peer Reviewed Journal
ISSN (Online): 2319-6564
www.ijesonline.com

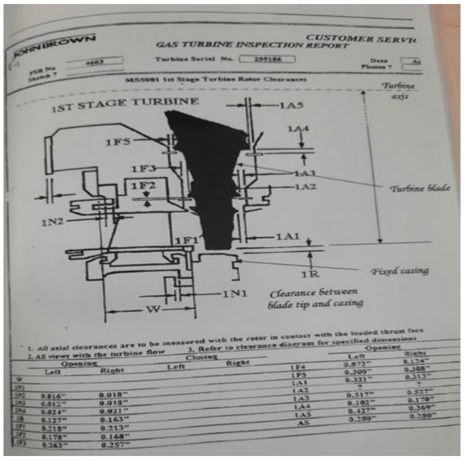

Fig. 8 First stage turbine blade clearance

3.1 Stress Analysis

The principal stress patterns on the concave and convex faces of the blade are shown in Figures 9 and 10. These stresses arise from the combined effects of centrifugal loading and lateral forces. It is important to note that these results should not be directly compared with predictions from simple beam theory, as bending stresses under transverse inertia loads during startup and shutdown are fundamentally different.

The temperature differential across the blade is significantly greater near the base, leading to higher localized stresses. Furthermore, the abrupt change in cross-sectional area near the base induces stress concentration. Overall, the centrifugal force is the dominant load component. ANSYS simulations predict that the maximum stress occurs at the blade base and is 128 MN/m². A short distance away from the base, the stress distribution changes sharply before becoming relatively uniform, decreasing to -664.859 kN/m² near the tip.

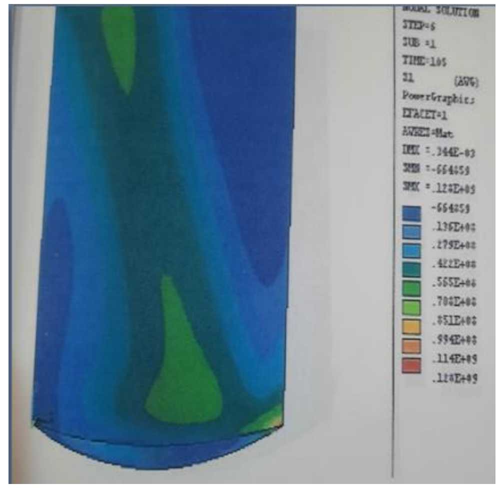

Fig. 9 Stress of concave face

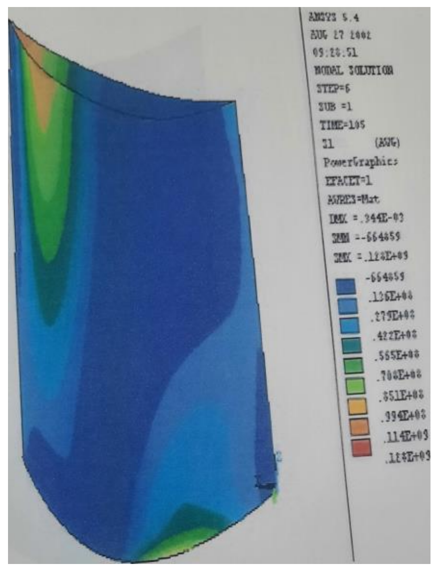

Fig. 10 Stress of convex face

3.2 Strain Analysis

The principal strains on the concave and convex faces of the blade are presented in Figures 11 and 12. The maximum strain, occurring at the blade base, is 620×10^{-6}. The minimum strain value, located near the tip, is 4.5×10^{-6}.

Fig. 11 Strain of convex face

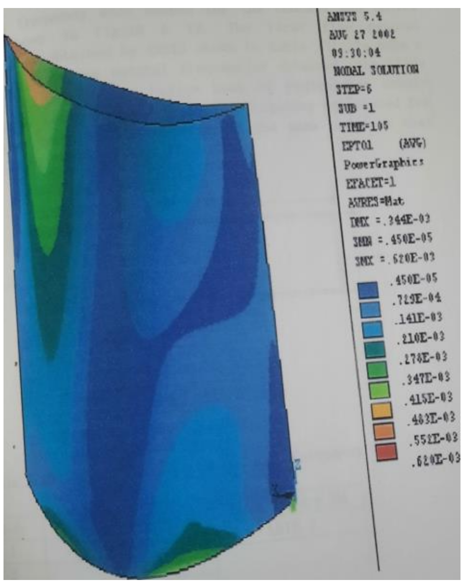

Fig. 12 Strain of convex face

3.3 Natural Frequency Analysis

The natural frequency mode shapes for the concave and convex faces are shown in Figures 18 and 19. The first five natural frequencies calculated using ANSYS are summarized in Table 3.

To validate the ANSYS results, a simplified analytical model was employed: the blade was approximated as a hollow rectangular beam 60X12 mm and 1 mm thickness, as illustrated in Figure 13. For this model, the first natural frequency was calculated using classical beam theory and found to be of the same order of magnitude as the ANSYS result.

Fig. 13 Hollow beam

Table 3: ANSYS and Calculated Natural Frequencies

Mode	ANSYS Solution (Hz)	Beam Theory (Hz)
1	1246	1515.7
2	1614	
3	2543	
4	2705	
5	2914	

Note: Beam theory calculations were performed only for the first mode for validation purposes.

4. Conclusion and Recommendation

4.1 Conclusions

This study demonstrates that ANSYS® provides a reliable computational framework for evaluating elastic stresses, displacements, strains, and natural frequencies in gas turbine blades under representative operating loads.

The failure of first-stage turbine blades in Khartoum North Gas Turbine Unit 2 is attributed to thermal overload. This conclusion is supported by finite element predictions of excessive tip displacement and corroborated by post-failure inspection revealing localized oxidation and burn marks on the removed blades—indicative of prolonged exposure to elevated temperatures beyond design limits.

Finite element analysis further indicates that maximum von Mises stress and equivalent plastic strain occur at the airfoil base, consistent with the expected stress concentration in cantilevered blade configurations. Additionally, stress and strain levels on the concave (pressure) surface exceed those on the convex (suction) surface, which is consistent with the direction and magnitude of the applied aerodynamic load (57,285 N) acting normal to the pressure side.

A simplified beam model, used to approximate the blade's fundamental vibrational characteristics, yielded a first-mode natural frequency within the same order of magnitude as the three-dimensional ANSYS model. This agreement validates the fidelity of the numerical approach and confirms the suitability of the finite element model for dynamic assessment of similar blade geometries.

4.2 Recommendations for Future Work

The present analysis considered only steady-state conditions—constant rotational speed, uniform temperature distribution, and static gas pressure. For improved predictive capability, future work should incorporate transient thermal-mechanical coupling during critical operational phases such as startup, shutdown, and rapid load changes, which are known to induce significant thermal gradients and cyclic stresses.

Furthermore, the influence of unsteady aerodynamic forces—particularly those arising from combustor-driven pressure pulsations—should be investigated through time-domain or harmonic response analyses. Such excitations may excite structural resonances near operational speeds, potentially leading to high-cycle fatigue (HCF), a common failure mode in gas turbine blades.

Integration of fluid–structure interaction (FSI) modeling and probabilistic life assessment methods is also recommended to enhance reliability predictions under realistic service conditions.

References

[1] Ramamurti, V., and Sreenivasamurthy, S., 1980, "Dynamic Stress Analysis of Rotating Twisted and Tapered Blades," *Proc. Inst. Mech. Eng.*, January.

[2] Shen, M. H. H., 1999, "Reliability Assessment of High Cycle Fatigue Design of Gas Turbine Blades Using the Probabilistic Goodman Diagram," *Elsevier*.

[3] Abdul-Aziz, A., and Kalluri, S., 1996, "Thermal and Structural Analysis Conducted on Hollow-Core Turbine Blade of the Space Shuttle Main Engine," NASA Technical Report, April.

[4] Elliott, T. C., 1989, *Standard Handbook of Power Plant Engineering*, McGraw-Hill, New York.

[5] Cohen, H., Rogers, G. F. C., and Saravanamuttoo, H. I. H., 1987, *Gas Turbine Theory*, 3rd ed., Longman Scientific & Technical, Harlow, UK.

[6] Rao, S. S., 1982, *The Finite Element Method in Engineering*, Pergamon Press, Oxford.

[7] Allen, J. M., and Erickson, I. B., 1975, "NASTRAN Analysis of a Turbine Blade and Comparison With Test and Field Data," *ASME Paper No. 75-DET-23*, March.

[8] Zuladzinki, G., 1977, "Effect of Centrifugal Forces on Lateral Deflection and Fundamental Frequency of Turbine Blade," *ASME Paper No. 77-DET-25*, September.

[9] Charbonneau, E., and Lakis, A. A., 2000, "Semi-Analytical Shape Functions in the Finite Element Analysis of Rectangular Plates," *Int. J. Mech. Sci.*, 242, pp. [page range if available], August.

[10] Ahmad, S., Anderson, R. G., and Zienkiewicz, O. C., 1970, "Vibration of Thick Curved Shells with Particular Reference to Turbine Blades," *Proc. Inst. Mech. Eng.*, April.

[11] Zilard, R., 1974, *Theory and Analysis of Plate*, Prentice-Hall, Englewood Cliffs, NJ.

APPENDIX B: APPLICATION OF ANSYS FOR DYNAMIC ANALYSIS OF HOLLOW TURBINE BLADE

The following steps outline the procedure for creating the model and performing a modal analysis based on the input data shown in Table (1).

A. CREATE MODEL

This section details how to set up the geometry and material properties for the hollow turbine blade.

Step 1: Specify the Title and Set Preferences

1. Navigate to: `Utility Menu > File > Change Title`.
2. Enter the text **“Hollow turbine blade”** and click **“OK”**.
3. Navigate to: `Main Menu > Preferences`.
4. Select the **Structural** option and click **“OK”**.

Step 2: Define Element Types

1. Navigate to: `Main Menu >Preprocessor> Element Type > Add/Edit/Delete`. The *Element Types* dialog box appears.
2. Click **“Add”**. The *Library of Element Types* dialog box appears.
3. In the left scroll box, select **“Structural Shell”**.
4. In the right scroll box, select **“Elastic Quad 4node 63”**.
5. Click **“Apply”**.
6. Click **“OK”**.
7. Click **“Close”** in the *Element Types* dialog box.

Step 3: Define Real Constants

1. Navigate to: `Main Menu >Preprocessor> Real Constants > Add`. The *Real Constants* dialog box appears.
2. Click **“Add”**. The *Element Types for Real Constant* dialog box appears.
3. Click **“OK”**. The *Real Constant for Shell163* dialog box appears.
4. Enter **`1`** for **Real Constant Set No.**.
5. Enter **`0.001`** for **Shell thickness at node I TK(I)**.
6. Click **“Apply”**.
7. Click **“OK”**.
8. Click **“Close”** in the *Element Types* dialog box.

Step 4: Define Material Properties

1. Navigate to: `Main Menu >Preprocessor> Material Props > -Constant- Isotropic`. The *Isotropic Material Properties* dialog box appears.
2. Click **“OK”** to specify material number 1. A second dialog box appears.
3. Enter **`207000000000`** for **EX** (Young’s Modulus).
4. Enter **`7850`** for **DENS** (Density).
5. Enter **`0.3`** for **NUXY** (Poisson’s Ratio).
6. Click **“OK”**.

Step 5: Create Keypoints (KPs) at Given Locations

Keypoints are primary points defining the surface geometry of the blade.

1. Navigate to: `Main Menu >Preprocessor> -Modeling- Create >Keypoints> In Active CS`. The *Create Keypoints in Active Coordinate System* dialog box appears.
2. Enter **`1`** for the keypoint number and **`0, 0, 0`** for the X, Y, and Z coordinates. Use the **TAB** key to move between fields.
3. Click **“Apply”**.
4. Repeat this procedure for the following keypoints:
   - **Keypoint 2:** `0.031, 0.033, 0.000`
   - **Keypoint 3:** `0.063, 0.000, 0.000`
   - **Keypoint 4:** `0.032, 0.015, 0.000`
   - **Keypoint 5:** `0.000, 0.026, 0.095`

- **Keypoint 6:** `0.020, 0.040, 0.095`
- **Keypoint 7:** `0.063, 0.003, 0.095`
- **Keypoint 8:** `0.020, 0.029, 0.095`

5. After entering the last keypoint, click **“OK”**.
6. Navigate to: `Utility Menu >PlotCtrls> Window Controls > Window Options`.
7. In the scroll box for *Location of triad*, scroll to **“Not shown”** and select it.
8. Click **“OK”**.
9. Navigate to: `Utility Menu >PlotCtrls> Numbering`.
10. Enable **Keypoint numbering** and click **“OK”**. The numbered keypoints will now be visible in the ANSYS Graphics window.

Step 6: Create Lines and Splines between Keypoints

A spline is a curved line connecting three keypoints.

1. Navigate to: `Main Menu >Preprocessor> -Modeling- Create > -Lines- Splines > Spline thru KPs`. The *Create B-Spline* picking menu appears.
2. Click once on keypoints **1, 2, and 3** in that order. A line appears between them.
3. Click once on keypoints **1, 4, and 3** in that order. A line appears between them.
4. Click once on keypoints **5, 6, and 7** in that order. A line appears between them.
5. Click once on keypoints **5, 8, and 7** in that order. A line appears between them.
6. Click once on keypoints **1 and 5** in that order. A line appears between them.
7. Click once on keypoints **3 and 7** in that order. A line appears between them.
8. Click **“OK”** in the picking menu.
9. Click **“SAVE_DB”** on the ANSYS Toolbar.

Step 7: Create Surface Area

The surface area defines the 2D geometry of the turbine blade.

1. Navigate to: `Main Menu >Preprocessor> -Modeling- Create > -Areas- Arbitrary > By Lines`. The *Create Area by Lines* picking menu appears.
2. Click once on lines **1, 3, 5, and 7**.
3. Click **“OK”**. The area defined by these lines is highlighted.
4. Click once on lines **2, 4, 5, and 7**.
5. Click **“OK”**. The area defined by these lines is highlighted.
6. Click **“SAVE_DB”** on the ANSYS Toolbar.

Step 8: Define the Mesh Density and Mesh the Area

1. Navigate to: `Main Menu >Preprocessor>MeshTool`. The *Mesh Tool* dialog box appears.
2. Click on **“Smart Size”**.
3. Set the size to **`5`** for fine meshing.
4. Under *Mesh*, select **“Area”**.
5. Under *Shape*, select **“Quad”**.
6. Under *Mesher*, select **“Free”**.
7. Click **“Mesh”**. The *Mesh Areas* picking menu appears.
8. Click **“Pick All”**.
9. Click **“OK”**.
10. In the *Mesh Tool* dialog box, click **“Refine”**. The *Refine Mesh At Elements* menu appears.
11. Click **“Pick All”**.
12. Click **“OK”**.
13. Click **“SAVE_DB”** on the ANSYS Toolbar.

B. MODAL ANALYSIS

Modal analysis determines the vibration characteristics (natural frequencies and mode shapes) of the structure.

Step 1: Enter Solution and Specify Analysis Type and Options

1. Navigate to: `Main Menu > Solution > -Analysis Type- New Analysis`. The *New Analysis* dialog box appears.
2. Select the **“modal analysis”** option and click **“OK”**.

3. Navigate to: `Main Menu > Solution > -Analysis Options`. The *Modal Analysis* dialog box appears.
4. Select the **“subspace”** option.
5. Enter **`5`** for the number of modes to extract.
6. Click **“OK”**. The *Subspace Modal Analysis* dialog box appears.
7. Click **“OK”** to accept the default values.

Step 2: Deselect PLANE42 Elements

Unselect any PLANE42 elements used for 2-D area meshing, as they are not needed for this analysis.

1. Navigate to: `Utility Menu > Select > Entities`. The *Select Entities* dialog box appears.
2. In the top two scroll boxes, select **“Elements”** and **“By Attributes”**.
3. Enable the **“Elem type num”** option.
4. Enter **`1`** in the *Min. Max. Inc.* field for the element type number.
5. Enable the **“Unselect”** option.
6. Click **“Apply”**.

Step 3: Apply Constraints to the Model

1. Navigate to: `Utility Menu > Select > Entities`. The *Select Entities* dialog box appears.
2. In the top two scroll boxes, select **“Nodes”** and **“By Location”**.
3. Enable the **“X coordinates”** option.
4. Enter **`0`** in the *Min. Max.* field for the X-coordinate location.
5. Enable the **“From Full”** option.
6. Click **“Apply”**.
7. Navigate to: `Main Menu > Solution > -Loads- Apply > -Structural- Displacement > On Nodes`. The *Apply U, ROT on Nodes* picking menu appears.
8. Click on nodes **`2, 4, 41-51, and 62-74`**. Click **“Apply”**.
9. In the *Apply U, ROT on Nodes* dialog box, select **“All DOF”** and click **“Apply”**.
10. Enter **`0`** for the *Displacement value* and click **“OK”**.
11. Return to the *Select Entities* dialog box. In the second scroll box, select **“By NumPick”**.
12. Click **“Sele All”**.
13. Click **“Cancel”** to close the dialog box.

Step 4: Specify the Number of Modes to be Expanded and Solve

1. Navigate to: `Main Menu > Solution > -Load Step Opts- ExpansionPass> Expand Modes`. The *Expand Modes* dialog box appears.
2. Enter **`5`** for the number of modes to expand.
3. Click **“OK”**.
4. Navigate to: `Main Menu > Solution > -Solve- Current LS`. The *STAT Command* dialog box appears.
5. Review the information in the dialog box, then close it using `File > Close`.
6. Click **“OK”** to begin the solution.
7. Click **“Close”** when the solution is complete.

Step 5: List the Natural Frequencies

1. Navigate to: `Main Menu > General Postproc> Results Summary`. Review the information in the dialog box, then close it using `File > Close`.

Step 6: View the Five Modes

1. Navigate to: `Main Menu > General Postproc> -Read Results- First Set`.
2. Navigate to: `Utility Menu >PlotCtrls> Animate > Mode Shape`. The *Animate Mode Shape* dialog box appears.
3. Enter **`5`** for the time delay in seconds.
4. Click **“OK”**. The *Animation Controller* dialog box appears, and the animation begins.
5. Click **“Stop”** to halt the animation.
6. Navigate to: `Main Menu > General Postproc> -Read Results- Next Set`.
7. Navigate to: `Utility Menu >PlotCtrls> Animate > Mode Shape`. The *Animate Mode Shape* dialog box appears.

8. Click **“OK”** to accept the previous settings. The animation begins.
9. Click **“Stop”** to halt the animation.
10. Repeat Steps 6–9 for the remaining three modes.

Step 7: Exit ANSYS

1. Choose **“QUIT”** from the ANSYS toolbar.
2. Select **“Quit”**, **No Save**.
3. Click **“OK”**.

C. DYNAMIC ANALYSIS

Dynamic analysis determines displacement, strain, stress, and other response quantities over time.

Step 1: Define Analysis Type and Analysis Options

1. Navigate to: `Main Menu > Solution > -Analysis Type- New Analysis`.
2. Select **“Transient”** and click **“OK”**.
3. Navigate to: `Main Menu > Solution > -Analysis Options`. The *Transient Analysis* dialog box appears.
4. Select **“Full”** and click **“OK”**. The *Full Transient Analysis* dialog box appears.
5. In the dropdown menu for *Stress-Stiff effects*, select **“On”**.
6. Click **“OK”**.

Step 2: Set Load Step Options

1. Navigate to: `Main Menu > Solution > -Load Step Opts- Time/Frequenc> Time – Time Step`. The *Time and Time Step Options* dialog box appears.
2. Enter **`36000`** for *Time at end of load step*.
3. Enter **`1000`** for *Time step size* and click **“OK”**.

Step 3: Apply Loads for the First Load Step

1. Navigate to: `Main Menu > Solution > -Loads- Apply > -Structural- Displacement > On Nodes`. The *Apply U, ROT on Nodes* picking menu appears.
2. Click on nodes **`2, 4, 41-51, and 62-74`**. Click **“Apply”**.
3. In the *Apply U, ROT on Nodes* dialog box, select **“UY, UZ, ROTX, ROTY, ROTZ”** and click **“Apply”**.
4. Enter **`0`** for the *Displacement value* and click **“OK”**.
5. Click **“SAVE_DB”** on the ANSYS Toolbar.

Step 4: Solve the First Load Step

1. Navigate to: `Main Menu > Solution > -Solve- Current LS`.
2. Review the information in the status window, then click **“Close”**.
3. Click **“OK”** on the *Solve Current Load Step* dialog box to begin the solution.
4. Click **“Close”** when the solution is complete.

Step 5: View the Results

1. Navigate to: `Main Menu > General Postproc> Plot Results > Deformed Shape-Nodal solution`. The *Contour Nodal Solution Data* dialog box appears.
2. Under *Item to be contoured*, select **“Stress – 1st principal”**.
3. Under *Item to be plotted*, select **“Def shape only”**.
4. Click **“OK”**. The plotted contour for stress appears. Review the information and close the dialog box.
5. Navigate to: `Main Menu > General Postproc> Plot Results > Deformed Shape-Nodal solution`. The *Contour Nodal Solution Data* dialog box appears.
6. Under *Item to be contoured*, select **“Strain-total – 1stprin”**.
7. Under *Item to be plotted*, select **“Def shape only”**.

8. Click **“OK”**. The plotted contour for strain appears. Review the information and close the dialog box.
9. Navigate to: `Main Menu > General Postproc> Plot Results > Deformed Shape-Nodal solution`. The *Contour Nodal Solution Data* dialog box appears.
10. Under *Item to be contoured*, select **“DOF Solution – USUM”**.
11. Under *Item to be plotted*, select **“Def + undeformed”**.
12. Click **“OK”**. The plotted contour for displacement appears. Review the information and close the dialog box.

Step 6: Print or Save the Results

1. Navigate to: `Utility Menu >PlotCtrls> Capture Image`. The *Image 1* dialog box appears.
2. Print the image or save the dialog box using `File > Print` or `File > Save`.

Step 7: Exit ANSYS

1. Choose **“QUIT”** from the ANSYS toolbar.
2. Click on the **“Save”** option you want, and click **“OK”**.