



Investigating and Calculating Aerodynamic Characteristics on Aviation Engine Turbines by Using Ansys CFX

MSc Thanh Cuong Nguyen^a, Ph. D Truong Thanh Nguyen^b, MSc Van Huong Ngo^c, MSc The Son Nguyen^d, MSc Thanh Chung Le^e.

^{a,b,c,d,e} Air Force Officer's College, Nha Trang City, Khanh Hoa, 650000, Vietnam

ABSTRACT

Numerical simulation is a field that has been strongly developed in many countries around the world. Especially, the use of numerical methods in combination with computer programs to solve related problems helps to reduce time and costs; can study systems where experimentation is difficult or impossible; Calculation of structures, physical and chemical interactions give detailed solutions to warn of possible dangers. This paper presents application of CFX software in study and calculation of aerodynamic characteristics on turbines of aviation engines. Specifically, the influence of temperature and pressure on the combustion air stream affects the turbine blades during the operation of the engine. The result can provide the basis for determining the dangerous load area on the turbine blades.

Keywords: Uhs; turbine blades; CFX; temperature; pressure; jet turbine engines.

1. Problem formulation

Gas turbines are widely used in jet turbine engines for aviation applications. Turbine blades are always subjected to a very high temperature and high pressure environment, especially the first stage of the turbine, the greater the influence. Along with that, in the process of using the engine, the damage appearing on the turbine blades appears more and more over time, leading to damage and reduced reliability.

Failures in turbines occur due to long-term accumulation and are difficult to observe and detect. Therefore, when the damage occurs, it is to a large extent and difficult to overcome and repair.

In order to complete the calculation method of aerodynamic characteristics on the turbine blades and have a scientific basis for surveying, assessing the status and diagnosing failures, it is necessary to identify the areas temperature distribution and stress concentration on the blades during engine operation.

2. Solving the problem

2.1. The main equations used for calculations in CFD

- Continuity equation (equation of conservation of mass) for a moving gas stream:

$$\frac{D\rho}{Dt} + \rho \nabla \cdot \vec{V} = 0 \quad (1)$$

- Momentum equation:

$$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x \quad (2a)$$

$$\rho \frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y \quad (2b)$$

$$\rho \frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z \quad (2c)$$

2.2. Turbulent model

For the problem of thermodynamic simulation on engine turbines, to obtain results with high accuracy and flow separation under high temperature and pressure, the author chooses the Shear Stress Transport (SST). The SST model is the most complete physical model, having a combination of the k- ϵ model in the free-flow and the k- ω model near the wall.

The turbulent flow model is determined by the formula:

$$\mu_t = \rho \frac{\bar{k}}{\omega} \mu$$

Combining the SST entanglement model with two basic equations, the continuity equation and the momentum equation, we get a closed system of equations sufficient to solve the temperature and pressure distribution fields on the turbine blade. In addition, additional models of heat transfer and heat transfer are added to increase accuracy in the simulation process.

2.3. Element model

The process of solving a simulation problem by the finite volume method is first to subdivide the computational domain (mesh), next is to integrate the different forms of differential equations on each domain and perform the operation. interpolate to describe the relevant variables in the center of the grid cell. As a result, we get equations in discrete form, allowing us to preserve parameters such as mass, momentum, and energy.

Through the use of tools on CFX software, the author chooses an automatic meshing method with a Hexahedral grid structure, and at the same time sets up the meshing to be taken according to the parameter Y^+ and Reynolds number to high accuracy.

2.4. Steps of Computational Simulation

From the theoretical basis of simulation, the process of solving the CFD problem in general and the simulation problem of thermodynamic calculation on the turbine by Ansys CFX software in particular is performed in the following order:

- Step 1: Rebuild the model
- Step 2: Determine input parameters and boundary conditions
- Step 3: Set the solution parameters for the problem
- Step 4: Solve the simulation problem
- Step 5: Read and process the calculation results.

3. Results and Discussion

3.1. Simulated input data

The input parameters of the simulation process:

- Directional leaves of the 1st floor: 51
- Turbine blades of the 1st floor: 71
- Directional leaves in the 2nd floor: 47
- Turbine blades in the 2nd floor: 61
- Distance between directional blades and turbine blades: 9mm
- Opening angle of turbine air duct: 17 degrees
- Air channel inlet height: 31mm
- Air channel outlet height: 61mm
- Rotor disc inner diameter: 240 mm
- Maximum flow of combustible gas at inlet: 8.2 kg/s
- Maximum rotation speed: 21000 rpm
- Air flow rate at turbine inlet: 190 m/s

- Turbine inlet temperature : 1153 K
- Turbine inlet pressure: $6,589,10^5$ N/m²
- Combustible gas density: 1,205 kg/m³
- Dynamic Viscosity: $1.8375 \cdot 10^{-5}$ kg/ms

3.2. Computational simulation on CFX

Based on the typical geometric dimensions of the engine turbine blades, the author redesigned the simulation model with the Design Modeler tool in Ansys Workbench.

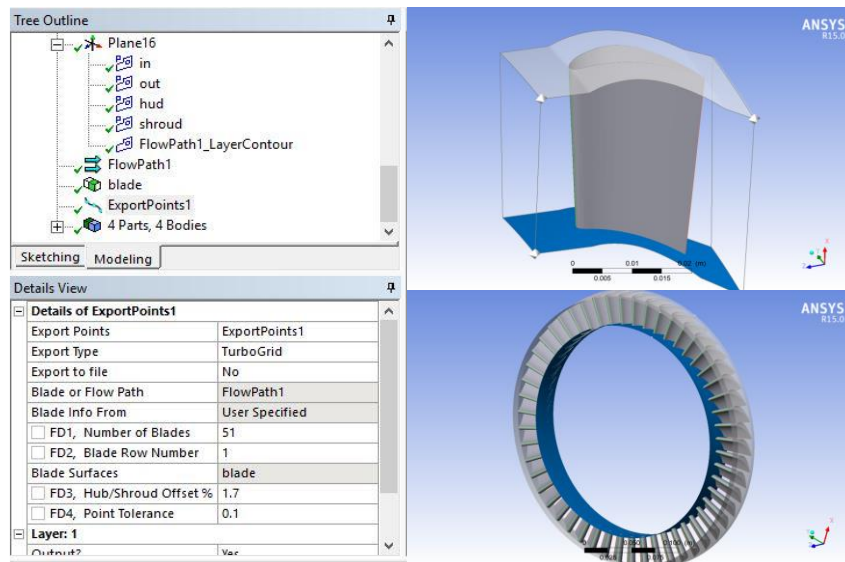


Figure 1. 3D model of directional leaf and distribution on engine turbine

The model mesh is divided using the TurboGrid tool, and the edges and edges of the profile are optimized to increase calculation accuracy (Figure 2).

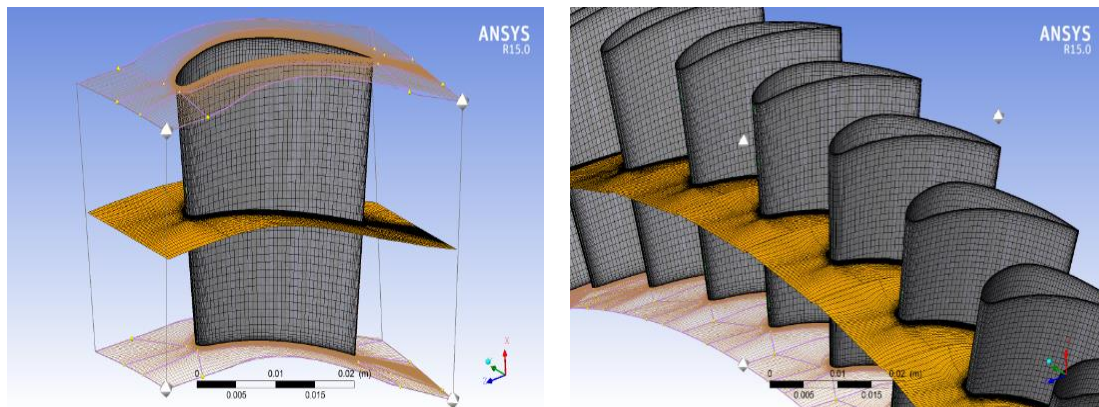


Figure 2. Grid pattern of directional leaves

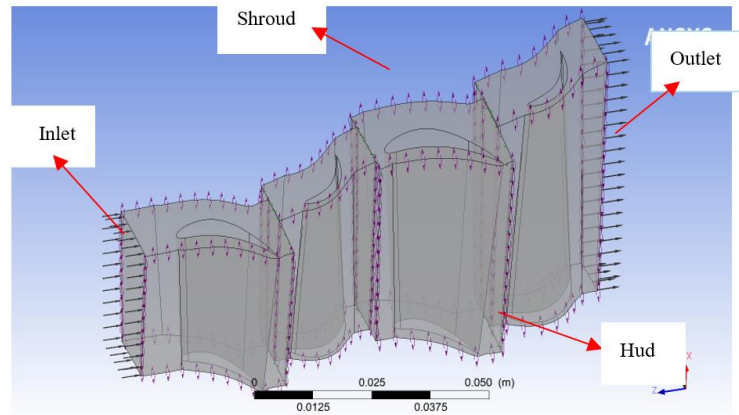


Figure 3. Computational domain and boundary conditions after model setting

3.3. Survey results in 4 operating modes

The process of surveying turbine blades is based on 4 basic operating modes of the engine. Each mode has a specific working parameter defined in Table 1.

Table 1. Input parameter table according to theoretical condition of the motor

Mode	Rotation (%)	Speed (rpm)	Temperature T_3 (K)	Inlet pressure (N/m ²)	Air flow (kg/s)
Low Throttle	64	13440	873	4,985.10 ⁵	5,25
Long Distance	94,5	19845	1023	5,836.10 ⁵	7,75
Normal	96	20160	1063	6,079.10 ⁵	7,87
Max	100	21000	1153	6,589.10 ⁵	8,2

The temperature and pressure distribution fields on the turbine blades according to the operating modes of the engine are shown in Figures 4 and 5.

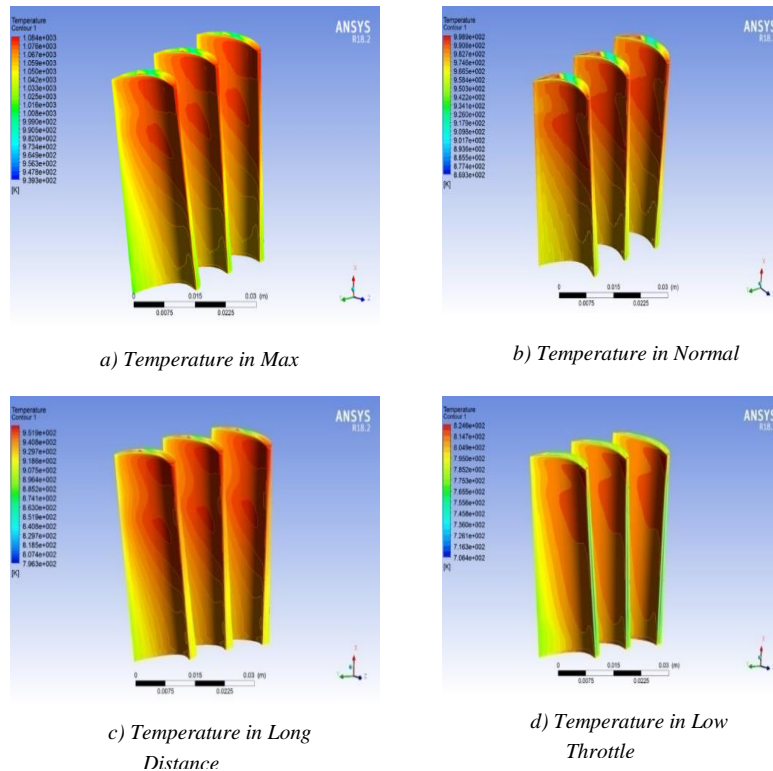
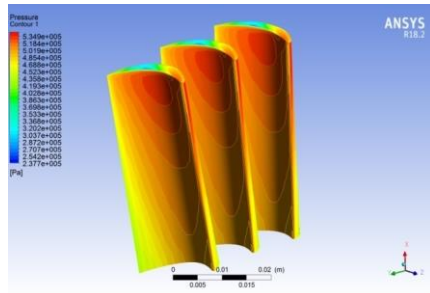
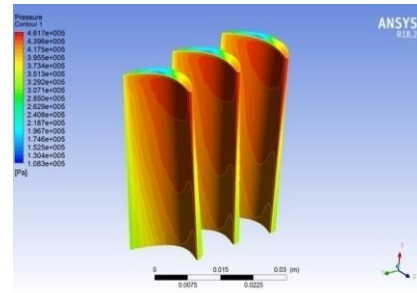


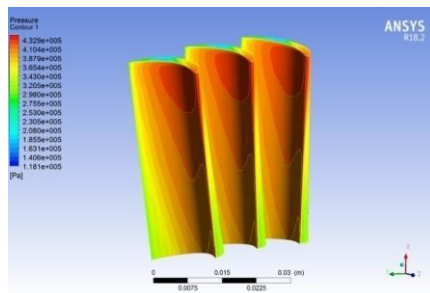
Figure 4. The temperature distribution field on the turbine blade according to the operating modes



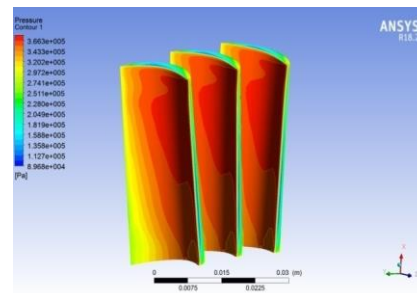
a) Pressure in Max



b) Pressure in Normal



c) Pressure in Long Distance



d) Low throttle pressure

Figure 5. Pressure distribution field on turbine blades according to operating modes

In summary:

1. Through surveying the movement of gas flow when passing through turbine stages in different working modes, values such as braking temperature, braking pressure change in accordance with the working mode of the engine and the law of change of gas flow when moving through the engine.
2. Based on the analysis results of the field of temperature and pressure distribution on the turbine blades, it is possible to see the change of temperature and pressure along the length and surface of blades. The locations where the greatest stress and temperature are concentrated are those that are easily damaged, causing damage to the components.

CONCLUSION

With powerful tools from CFX simulation software, it helps a lot in the process of studying and calculating aerodynamic characteristics on engine turbines. Based on the obtained research results, it is helpful in identifying dangerous load areas on turbine leaves from which effective exploitation and use measures can be taken. In addition, software can be applied to conduct surveys for many other types of engines being used today.

References

- [1] Fluent inc, *History of Fluent*, <http://www.fluent.com/about/history.htm>
- [2] Fluent inc, *Fluent 6.1 documentation*, <http://myrtle.kc.umkc.edu/classes/mcFluentHelp/Fluent/help/index.htm>.
- [3] J.K Matingly, *Elements of Gas Turbine Propulsion*, Mc Graw- Hill international 1996.
- [4] WANG Yi-wei, (2008), *Combination of CFD and CSD Packages for Fluid-Structure Interaction*