Turbine Blade Design of a Micro Gas Turbine

Bhagawat Yedla
Vellore Institute of Technlogy, Vellore – 632014, India

Sanchit Nawal
Vellore Institute of Technlogy, Vellore – 632014, India

Shreehari Murali
Vellore Institute of Technlogy, Vellore – 632014, India

Abstract- The introduction of small drones, missiles and small, made micro gas turbines fairly ubiquitous. The project refers to the design of a micro gas turbine blade. Micro gas turbines are smaller versions of Jet Engines typical used to propel aircrafts of medium to high sizes and capacity. The Blade is theoretically designed and further rendered in Solidworks 2014. CFD analysis has been carried out using ANSYS Fluent and ANSYS ICEM CFX, meshing done by ANSYS Mesh and results shown by CFX Post.

Index Terms - Aero Foil, Blade, Camber, Turbine

I. INTRODUCTION

A turbine is a mechanical shaft energy producing turbomachine. They are divided into two types namely radial inflow turbine and axial flow turbines. Generally axial turbines are used for their efficiency and power to weight ratio.

Micro gas turbines are smaller versions of larger jet engines and are typically used to power small scale turbo generators or even in the transmission axles of electric or hybrid vehicles. These turbines, along with heat and power cogeneration components, advancement in electronic supply and stability, can achieve efficiencies of above 80%.

II. METHODOLOGY

Design Constraints

Every turbine is designed with respect to a certain preceding compressor design values or assumption. The blade is hence designed according to these certain initial values.

Power demanded by the compressor	$P_w = 230000$ J/kg
Blade speed	$U = 440$ m/s
Flow Coefficient	$\Phi = 0.7$
Reaction ratio	$R = 0.4$
Axial velocity component	$C_x = 307$ m/s
Inlet stagnation temperature	$T_{01} = 1300K$
Inlet stagnation pressure	$P_{01} = 9.8$ bar

Free vortex design

Specific heat ratio	$\gamma = 1.32$
Specific heat constant	$C_p = 1.178$ kJ/kg
Target efficiency	$\eta = 0.95$

Final Calculated Values

For Blade Angle

Tanα_2 =ΔW/U
C_x =307 m/s α_2 = 59.492° C_2 = 606.85 m/s

M_2 = 0.894 Mach   Since α2 = 0°

tanβ_3 = U/C_x= 1/Φ =1/0.7 β_3 =55.00°
tanβ_2= tanβ_3-2R/Φ = 0.28 β_2= 15.945°

W_3 = 307/cos55 =535.238 m/s
C_2= 307/cos 59.492 =604.73 m/s
T_2= T01-C_2 /2C_p
= 1300- 6042/2x1200 =1149.99K

M_2 = 0.9185 @1150K

For Nozzle Area

$A_2 = m/(\rho_2 C_X)$
$P_2/P_{01} = (T_{2s} / T_{01})/\eta_n$
P_2 = 584.549 kPa

C_X =307m/s
Since α_3=0
P_3 =4.3013 bar
A =0.02684m^2
Mass flow rate=A*ρC_x=14.5 kg/m^2

Exit Mach number
M=$C_3 / \sqrt{(\gamma R T_3)}$ =0.4847

α_1 =0° α_2 =59.492° α_3 =0°
β_2 =15.945 ° β_3 =55.09°

α_{2h} =67.350 ° α_{2m} =59.492° α_{2t} =51.945°
β_{2h} =33.498° β_{2m} =15.945° β_{2t} =-0.4349°
β_{3h} =52.108° β_{3m} =55.09° β_{3t} =57.693°

H_{st1} = 21mm H_{st2} =27 mm H_{st3} = 32 mm

b_{stator} = 25mm b_{rotor} =30mm
N_{stator} = 50 N_{rotor} = 45 blades

Mean diameter = 292.86 mm
Torque = Power/angular velocity = 230000/ 2034.48 = 113.05 Nm
Net tangential force = Torque/Mean Radius = 113.05/ 0.14 = 807.5 N
Force on one Blade = Net force/Number of blades = 807.5/50 = **16.15 N**

Figures

P_w	Power
U	Blade speed
Φ	Flow Coefficient

R	Reaction ratio
C_x	Axial velocity component
T_{01}	Inlet stagnation temperature
P_{01}	Inlet stagnation pressure
γ	Specific heat ratio
C_p	Specific heat constant
η	Efficiency
α	Air angle
β	Blade angle
M	Mach Number
A	Area

CAD

The blade was further designed using the aerofoil CH10 as a benchmark. It was modified according to the fluid inlet and exit values according to the above calculations.

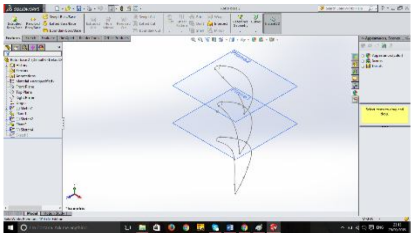

Fig 1

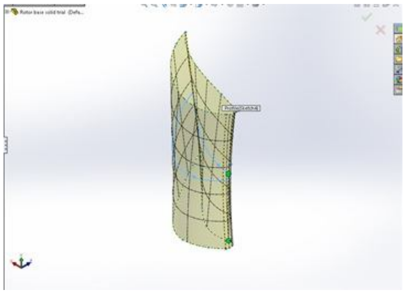

Fig 2

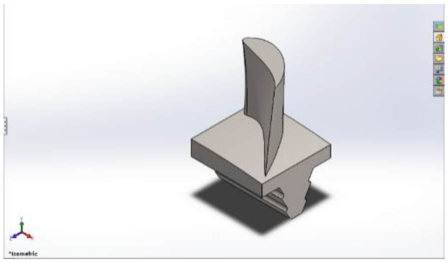

Fig 3

ICEM CFX Analysis

The material used for the blade is Titanium Aluminium Vanadium alloy due to its high thermal resistance and Modulus of elasticity – 104800.31 N/mm^2.

The blade is analysed in a compact imaginary fluid domain called as a Cascade. ANSYS BladeGen is used to generate the cascade.

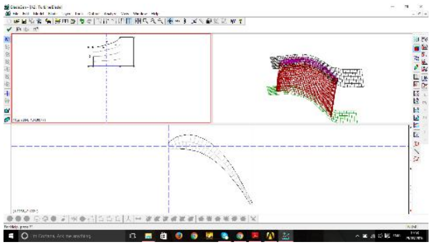

Fig 4

The cascade is the imported to a CFX Simulation geometry block in ANSYS Workbench called ICEM CFX and meshed.

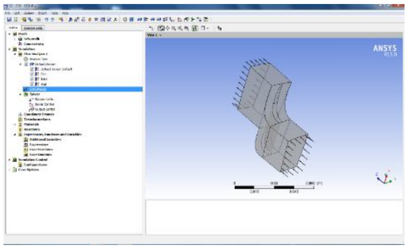

Fig 5

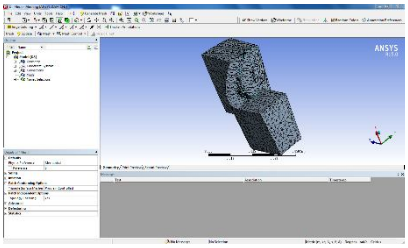

Fig 6

Nodes – 174862
Elements – 936496
Analysis Method – k-epsilon 2 equation model
Inlet velocity – 307 m/s

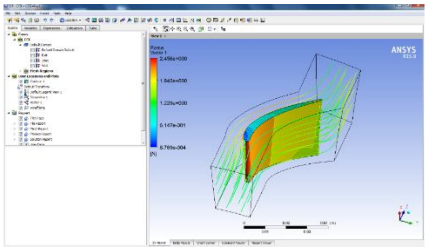

Fig 7

Flow separation is seen at the hub blade profile trailing edge due to shock losses which result in a lower generated force than the calculated values.

The force values given by Post Processor are:

F_x = 6.66 N
F_y = 0.632 N
F_z = -14.672 N (Negative due to blade lower camber facing +ve Z direction)

Resultant Force – 16.031 N
Error from calculation – 0.737 %

Static Structural Analysis

Further, with the force values calculated, a static structural analysis is carried out using ANSYS Static Structural.

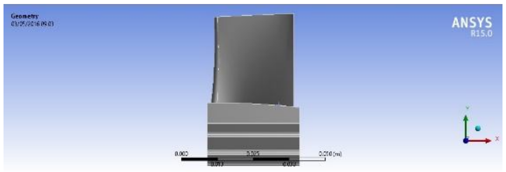

Fig 8

The blade is then meshed and analysed.

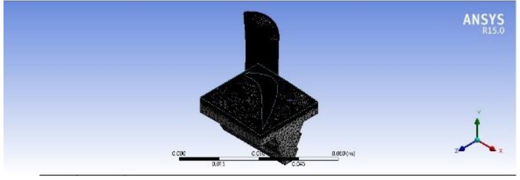

Fig 9

Nodes – 572383
Elements – 394617
Forces – 20 N lower camber, 10 N on Leading edge.

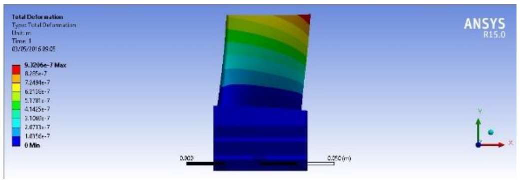

Fig 10

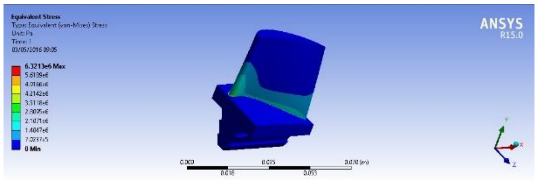

Fig 11

A total deformation of 9.32e-7 m is seen. A maximum stress value of 6.321e6 is calculated which falls within the modulus value of the material.

III. LITERATURE REVIEW

Turbines are designed specifically in relation to the output power required or decided. After the initial assumptions are laid out regarding flow coefficients, compressor power and desired efficiency, turbomachine concepts of velocity triangles defining the fluid entry and exit angles are calculated [1]. Further, for blade designs, aero foils are required. This is generally designed or modified from an existing aero foil [2]. Aero foils can be of varied shapes and can be designed with different camber and attack angles for maximum output power.

The literature gap found was the design methodology which changes due to the velocity triangles used and the blade aero foil which varies from engine to engine [3]. This enabled us to further reach design constants and carry out further blade calculations. The objectives of this project is to reach an efficient and power delivering blade design which can be further used on rotors. Modelling the same is a challenge in itself and analysing it further more. The design aspect is higher than the scope of the Turbomachinery field for bachelors study and hence this was a motivational challenge in itself.

IV. RESULTS AND DISCUSSION

The blade is designed using a base CH10 aero foil and further modified according to the calculated angles. CFD Analysis on the blade concludes an error of 0.737% is obtained through ANSYS simulation. This is due to the turbulence and shock losses suffered by the blade. Stress analysis on the blade concludes the maximum stress value to lie within the Modulus of Elasticity of the selected material.

V. CONCLUSION

The blade of a micro gas turbine is hence designed as per turbomachinery concepts of velocity triangles along with the input of a new modified aero foil. The CAD is made using Solidworks 2014 and the analyses are performed in ANSYS Workbench.

REFERENCES

[1] S. L. Dixon, Fluid Mechanics and Thermodynamics of Turbomachines
[2] Himanshu Singh, A Study Paper on Jet Engines
[3] General Electricals Global Research, Turbine Blade Design
[4] Rolls Royce Archives, The Jet Engine
[5] Philip G Hill, Mechanics of Propulsion
[6] John D Couli, Blade Loading and Meanline Design
[7] James Borg Bartolo, Design and Modelling of a 45KW Jet Engine
[8] David Winstanley, HTF7000 Engine Design
[9] Baumschule Mailwald, Research on Jet Engines ASME, Engineering of Gas Turbines