COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF A GAS TURBINE COMBUSTION CHAMBER USING ANSYS CFX

Hafiz Abdul Hadi [a], Muhammad Sajid Kamran [a], Naseem Abbas [b*]

[a] Department of Mechanical Engineering, University of Engineering and Technology Lahore, Pakistan

[b] Department of Mechanical Engineering, University of Central Punjab Lahore, Pakistan

*Corresponding author: naseem.abbas@ucp.edu.pk

ABSTRACT: *The Can-combustor is one of the types of burner in gas turbine. The burner is a chamber where fuel is burned in the presence of excess air. The basic can-combustor was modeled using Solid Edge ST6. A CAD model of already existing combustor chamber is developed and under the application of certain numerical boundary conditions CFD results are validated with the numerical results. The result validation provides an extension to the optimization of same combustion chamber. The results were compiled on the basis of temperature at the wall of combustion chamber, temperature at the exhaust, pressure loss during combustion, recirculation of primary air and the cooling effect created by the secondary air to minimize temperature at the walls of combustion chamber. The validation of the CFD results was done by theoretical calculations for the combustor. The comparison was done on the basis of mass flow rates and temperature ratio.*

1 INTRODUCTION

The gas turbine engines play an important role in the new technological world. The gas turbine converts chemical energy into the mechanical energy by the combustion of fuel in the presence of air. The combustion happens in the specific area of the gas turbine called the combustion chamber. The combustion chamber ignites the fuel and converts the chemical energy into mechanical energy. The exhaust gases interact directly to the turbine blades. The gases transfer the heat energy to the turbine blades and thus the turbine runs and gives the output in the form of torque or power [1]. It is used nowadays in petroleum rigs, electricity manufacturing and aircrafts industries [2]. The research mainly focuses upon the CFD results extracted from the CAD model of already existing combustion chamber, which later can become an extension for the optimization of same chamber on the basis of certain optimization parameters and result validation theory [3].

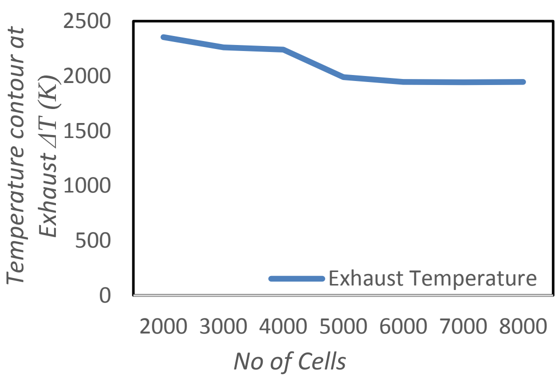
The gas turbine has many parts working together as a unit to convert the chemical energy into mechanical energy. The gas turbine includes propeller, compressor, combustion chamber and turbine. The combustion chamber has many types annular, can and can-annular. The research is directly based on the combustion of can type of combustion chamber. The can-combustor is used in certain aircrafts and in many stationery gas turbine engines. Design of combustion chamber is responsible for the overall efficiency of the gas turbine [4]. So far research mainly focuses upon the phenomenon through which bearing temperature of the exhaust and wall of the combustion chamber can be reduced and also the efficient methodology to reduce the exhaust harmful gases mainly includes emission of NOx, SOx and CO. In this regard Swirler is optimized by introducing new technology called as colorless distributed combustion technology. It also ensures high level of performance if fuel mixing is improved and proper combustion is accomplished with in the combustor [5].The modern gas turbine can now bear a remarkable temperature at the turbine blade and the combustion wall because of introduction of composite materials. For most of ceramics based combustor chambers can bear temperature up to 1300 K though the turbine entry can go as far as to 1800Kor beyond in the new modern era, because of presence of certain composites used in turbine blade manufacturing [6-8]. These composites can bear a very high temperature and thus the output power can easily be increased given by the new technology. Further in conventional Gas turbines for efficient working

it is required to raise the temperature from an optimized range of 450 – 1100 ^{0} C [9].

2 GEOMETRICAL MODELING:

The basic can combustor model is taken from the example done in ANSYS-CFX which is a 3D model of WHITTLE W2/700-21 JET ENGINE (courtesy Cranfield University, UK) as shown in figure 1. The dimensions are taken from the basic can-combustor physical model and the modeling is done using Solid Edge ST6 [10].

Figure 1 Physical Combustor



3 MESH CONVERGENCE:

Above graph explains the dependence of exhaust temperature contour results upon the density of mesh cells. It is evident that initially from 2000~4500 cells trend is quite unpredictable except that is downward but just after 5000 no of cells trend settles down under the range of 1940~1980 K temperature. Further at 6000 cells very minor effect on exhaust profile is observed so results for already existing combustor chamber CAD model is evaluated on the basis of this very mesh convergence methodology.

4 MATHEMATICAL MODELING AND

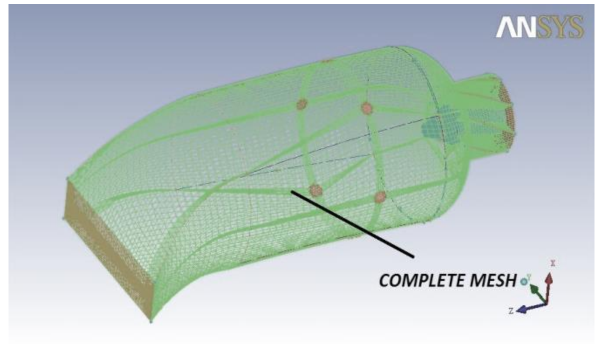


Figure 2 Complete Mesh

BOUNDARY CONDITIONS:

$$C_aH_b + \left[a + \frac{b}{4}\right]\left[O_2 + 3.772N_2\right] \rightarrow aCO_2 + \frac{b}{2}H_2O + xO_2 + 3.772\left[a + \frac{b}{4}\right]N_2$$

Above mentioned equation is used to determine the stoichiometric AFR but actually AFR is greater than the stoichiometry AFR, so for a gas turbine we took a primary good AFR ratio at high power output to model the problem. We took an excess air of about 160% to model the problem. Although, gas turbine can run on very lean AFR even more then 200%. The idea is to get the outcomes on high power working of a gas turbine. So the actual air fuel ratio taken for the working of gas turbine is calculated as given below. For 160% excess air the approximate AFR for methane and air is equals to 27.23 because the stoichiometric AFR ratio is 17.23.

Mathematical Modeling

Constant	Value
Stoichiometric A/F	17.23
Assumed Lambda	1.6
Actual A/F	27.23
Mass Flow Rate of Air	0.0659 kg/s
Mass Flow Rate of Fuel	2.448 kg/s
Equivalence Ratio	0.63
Total Enthalpy of Formation	-802 MJ/kmol
Total Heat of Reaction	-122.75 kJ/kmol
Exhaust Temperature	2128 K
Primary Air Inlet	10 m/s
Fuel Inlet	40 m/s
Secondary Air Inlet	6 m/s

5 RESULTS:

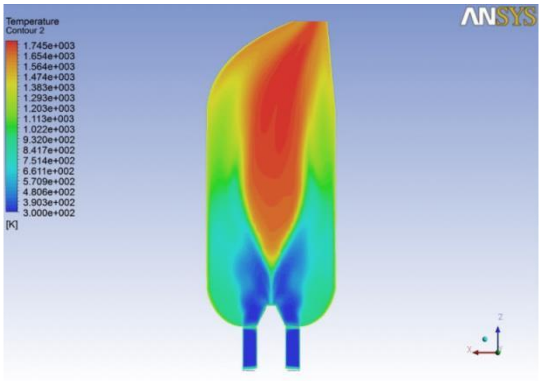


Figure 3 Overall Thermal Distribution

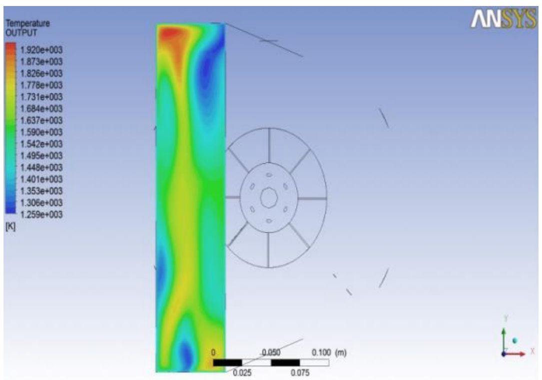


Figure 4 Temperature Contour at Outlet

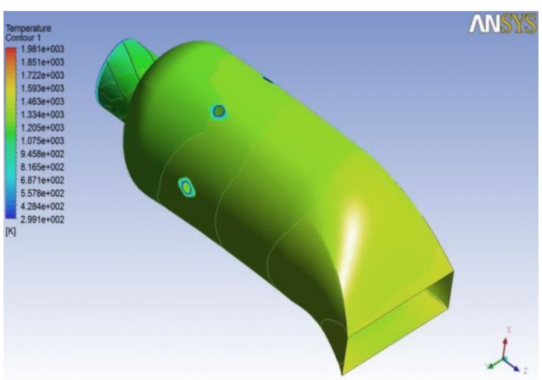


Figure 5 Temperature Distributions at Wall

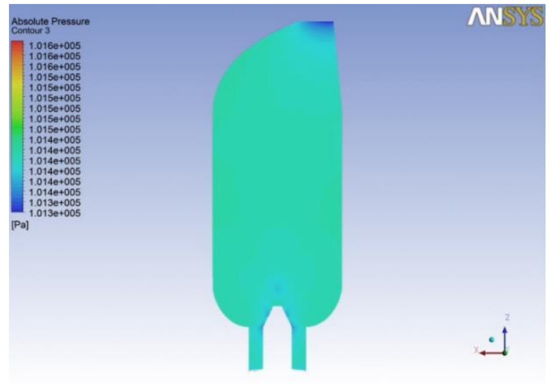


Figure 6 Pressure Distributions

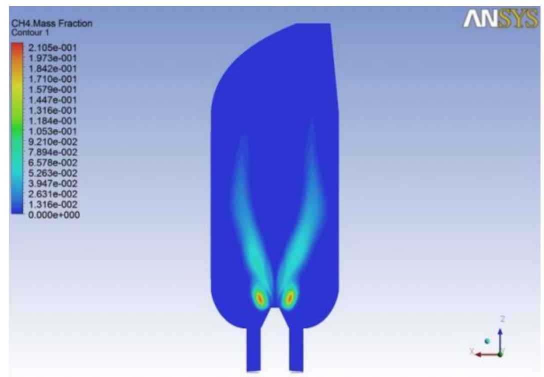


Figure 7 Mass Fraction of CH4

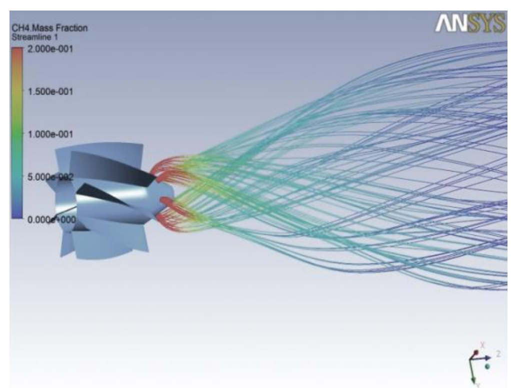


Figure 8 Basic Model Fuel Paths

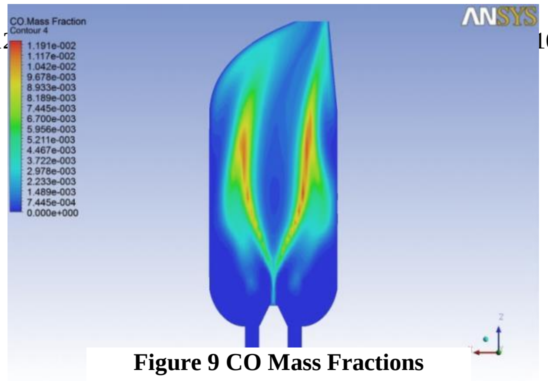


Figure 9 CO Mass Fractions

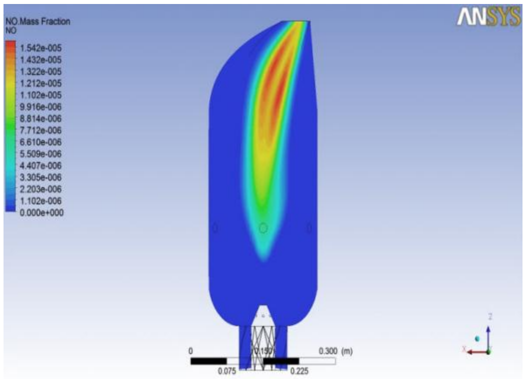


Figure 10 NO Mass Fraction

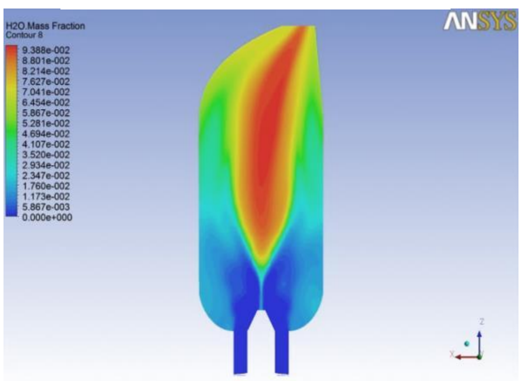


Figure 11 H2O Mass Fraction

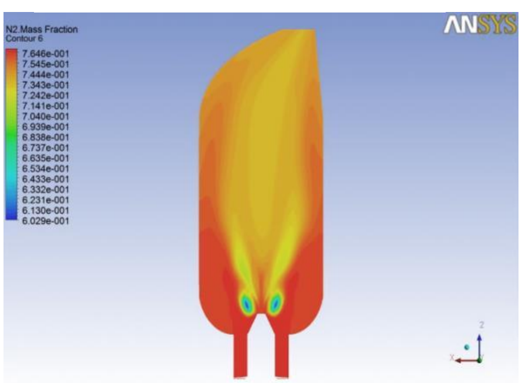


Figure 12 N2 Mass Fraction

Basic Model Results

	Temperature (K)	
Wall Temperature	Maximum Temperature	1700
	Minimum Temperature	307.7
	Average Temperature	1024
	Temperature Outlet (K)	
Outlet Temperature	Maximum Temperature	1944
	Minimum Temperature	1259
	Average Temperature	1599
Mass Fraction	**Mass fraction of CH4**	
	At Secondary Air Inlet	0.0070
	At Outlet	0.00075
	Mass fraction of O2	
	At Secondary Air Inlet	0.123
	At Outlet	0.10
	Mass fraction of CO	0.0027
	Mass fraction of NO	0.0000018
	Mass fraction of N2	0.743
	Mass fraction of H2O	0.070
	Mass fraction of CO2	0.081
	Absolute Pressure At Outlet (Pa)	
Pressure	Maximum Pressure	1.014e+05
	Minimum Pressure	1.012e+05
	Average Pressure	1.013e+05

6. CONCLUSION:

I conclude the following mentioned below results. First the basic model is compared with the theoretical results for validation of basic model. The optimization is successfully done and the results were also validated to the theory. The CFD modeling really helped the optimization and it is cost friendly because it gives the well good results as compared to theory. The numerical methods we applied gave the best possible solution. Present research gives an extension to the optimization of secondary air hole placing, outlet shape, fuel area, secondary air area and the swirler angle which may further be taken as future prospects of present research.

From above table it has been concluded that our CAD model of combustor chamber is in complete agreement with the theoretical results therefore considering this as base point research can further be taken to another level by optimizing each design parameter mainly involving fuel, air hole area and their placing also the designing of outlet. Research can be evaluated on the basis of emission or either temperature contour at wall and outlet.

6.1 VALIDATION OF BASIC MODEL:

Category	Theoretical Calculations	Computational Results	% Error
Mass flow rate of fuel (kg/s)	0.0659	0.0667	1.22 %
Mass flow rate of Air (kg/s)	0.002448	0.00243578	1.78 %
Temperature at exhaust (K)	2123	1944	1.09 %
Temperature Ratio	7.07	6.48	1.09 %

REFERENCES

Cappelletti, A. Numerical redesign of 100kw MGT combustor for 100% H2 fueling. *Energy Procedia 45* , 1412 – 1421. (2014).

[2] Hashimoto, N.Undamental combustion characteristics of Jatropha oil as alternative fuel for gas turbine. *Fuel 126*, 194–201 (2014)..

[3] Haugen, N. E. Hydrogen fuel supply system and re-heat gas turbine combustor. *Energy Procedia 23*, 151 – 160 (2012)..

[4] Khalil, A. E. Low calorific value fuelled distributed combustion with swirl for gas turbine applications. *Applied Energy 98*, 69–78 (2012)..

[5] Khalil, A. E. (2011). Distributed swirl combustion for gas turbine application. *Applied Energy 88*, 4898–4907.

[6] Liedtke, O. Development of a new lean burning combustor with fuel film evaporation for a micro gas turbine. *Experimental Thermal and Fluid Science 27*, 363–369 (2003)..

[1]

[7] O. Liedtke, A. S. Development of a new lean burning combustor with fuel film evaporation for a micro gas turbine.

Experimental Thermal and Fluid Science 27, 363–365 (2003)..

[8] Shinichi Kajita, R. D. Achieving ultra-low emissions in a commercial 1.4 MW gas turbine utilizing catalytic combustion. *5th International Workshop on Catalytic Combustion*, 279-288 (2003)..

[9] Tsujikawa, Y. Numerical simulation of the two-dimensional flow in high pressure catalytic combustor for gas turbine. *Catalysis Today 26*, 267-274 (1995)..

[10] Lefebvre, A. *Gas turbine combustion*, *1983.* Thermal Science and Propulsion Center School of Mechanical Engineering, Purdue University (1983)..