World Academy of Science, Engineering and Technology
International Journal of Aerospace and Mechanical Engineering
Vol:14, No:12, 2020

Conjugate Heat Transfer Analysis of a Combustion Chamber using ANSYS Computational Fluid Dynamics to Estimate the Thermocouple Positioning in a Chamber Wall

Muzna Tariq, Ihtzaz Qamar

Open Science Index, Aerospace and Mechanical Engineering Vol:14, No:12, 2020 publications.waset.org/10011704.pdf

Abstract**—**In most engineering cases, the working temperatures inside a combustion chamber are high enough that they lie beyond the operational range of thermocouples. Furthermore, design and manufacturing limitations restrict the use of internal thermocouples in many applications. Heat transfer inside a combustion chamber is caused due to interaction of the post-combustion hot fluid with the chamber wall. Heat transfer that involves an interaction between the fluid and solid is categorized as Conjugate Heat Transfer (CHT). Therefore, to satisfy the needs of CHT, CHT Analysis is performed by using ANSYS CFD tool to estimate theoretically precise thermocouple positions at the combustion chamber wall where excessive temperatures (beyond thermocouple range) can be avoided. In accordance with these Computational Fluid Dynamics (CFD) results, a combustion chamber is designed, and a prototype is manufactured with multiple thermocouple ports positioned at the specified distances so that the temperature of hot gases can be measured on the chamber wall where the temperatures do not exceed the thermocouple working range.

Keywords**—**Computational Fluid Dynamics, CFD, conduction, conjugate heat transfer, CHT, convection, fluid flow, thermocouples.

I. INTRODUCTION

DIRECT measurement of temperature or heat flux at a surface exposed to fire is very difficult. The temperature at such a surface is high enough to malfunction most devices used for the temperature measurement. This is a very common case inside the combustion chamber of rockets and air vehicles. Furthermore, temperature at the surface of space vehicles during reentry the atmosphere is so high that it cannot be directly measured by a sensor. In such cases, temperature sensors are placed under a hot surface at some distance and inverse heat transfer is performed to estimate the surface temperature and also to analyze the thermo-physical properties of the material during operation. The distance at which these thermocouples are to be positioned is an important factor to be estimated. In this paper, we are using CHT method for the estimation of the distance for accurate placement of thermocouples.

The environment inside the rocket, especially in the combustion chamber region, is a severe case of chemical reactivity, high turbulence, high temperatures and high velocities; hence, making it difficult for thermocouples to measure temperature profiles directly and restricts the type of thermocouple to be used [1].

Boundary layer formation in a rocket nozzle has a direct effect on rocket performance degradation, hence, temperature profile calculation on hot gas side of the wall, as well as, the coolant side of the wall is necessary for this purpose [2].

CHT refers to the coupling of conduction in solids with convection in fluids. In a rocket's combustion chamber, CHT is occurring as the hot gases after combustion go through convection and heat from these gases is conducted through the wall of combustion chamber. Heat is being transferred through the interface of fluid domain to solid domain, making it a CHT problem.

Modeling of a CHT problem is difficult due to the coupling of two heat transfer modes. Compact thermal model is used to model CHT in circular ducts [3]. The CHT approach provides detailed information on heat transfer through the combustion chamber wall, such as peak wall temperature, and also helps in improving engine design for reduced thermal stresses [4]. In CHT modeling and simulation, time scale disparity in solid and fluid domains is an important feature to be recognized. This disparity also justifies the use of quasi-steady coupling of both domains in CHT. This also makes the CHT simulation more time consuming [5]. Another complication that arises in CHT is of very small time step. While performing analyses in ANSYS CFX and FLUENT, a restrictive time step limit is allowed which makes it difficult and more time consuming for the simulation to converge [6]-[8].

Thermocouples have a defined “temperature range” where they can work without malfunctioning. The scope of this paper is to analyze the above mentioned temperature range for an experimental setup which maybe further used in Inverse Heat Transfer Experimentation. A thick walled chamber is required, so, at a certain wall thickness, thermocouple could be inserted to measure the temperature profile within the combustion chamber. CHT Analysis is performed in ANSYS to estimate that certain wall thickness. For analyses purpose, the maximum temperature limit has been taken form the family of k type thermocouples i.e., 1533.15 K [9].

Muzna Tariq is with Institute of Space Technology, Islamabad, Pakistan (corresponding author, phone: +92-336-5612823; e-mail: muznatariq99@gmail.com).

Ihtzaz Qamar is with Institute of Space Technology, Islamabad, Pakistan (e-mail: ihtzazqamar@gmail.com).

Open Science Index, Aerospace and Mechanical Engineering Vol:14, No:12, 2020 publications.waset.org/10011704.pdf

II. ANALYTICAL SOLUTION

A. Combustion Chamber Parameters

A combustion chamber designed and analyzed for the case under study has the parameters as in Table I.

TABLE I
COMBUSTION CHAMBER PROPERTIES

Propellant pair	Liquid oxygen (LOX)/ RP1
Mixing ratio	2.2
Chamber pressure	15 kPa (kg/ms^2)

B. Combustion Chamber Calculation

The parameters given in Table I are used to calculate the combustion chamber dimensions by using formulas given in [10] and then compared with results from an open source software CPROPEP which determines theoretical performance of rocket propellants engine compositions. The calculated parameters of chamber were in accordance with the CPROPEP results which are displayed in Table II. These dimensions are used to make a 3D model of the chamber, and CHT analysis is performed on this model.

TABLE II
COMBUSTION CHAMBER DIMENSIONS

Chamber Temperature (T_C)	3090.45 K
Combustion Chamber Diameter	0.07 m
Throat Diameter	0.01 m
Exit Diameter	0.0175 m
Combustion Chamber Length	0224 m
Convergent Divergent Nozzle Length	0.0756 m
Convergent Cone Angle	28°
Divergent Angle	14°
Exit Mach No. (M_E)	2.37

The combustion chamber used for the experimentation purpose is made of Stainless Steel (SS) 316. At stoichiometric mixing ratio for the propellant pair, the combustion temperature is quiet high for SS 316, as mentioned in Table II, considering that the melting temperature of SS 316 is 1643.15K [11]. This temperature limits the efficiency of combustion, hence making the use of cooling mechanism a necessity for the rocket engine. When a cooling mechanism is integrated in a rocket engine combustion chamber, it becomes a case of CHT as forced convection and conduction are occurring at the same time; hot gases resulting from combustion are convecting heat to the combustion chamber's wall and conduction is occurring in the wall where the coolant used for the cooling purpose is also convecting on the other side of the wall.

III. CHT ANALYSIS

All CFD simulations and data analysis were carried out in ANSYS CFX software and in-house performance codes. CFX is a commercial software and was available at Propulsion Engineering and Research Lab (PEARL), Institute of Space Technology.

A. Mesh Independence and CHT Solver Specification

The geometry was made in SOLIDWORKS and mesh was generated using ANSYS Mesh with preference to keep the grid resolution reasonable fine, including inflation layers and body sizing to capture boundary layer formation along the wall of chamber. Multiple meshes were also constructed for grid independence study to achieve shorter convergence time and higher solution accuracy. The selected mesh, for CHT study, with 2.5 million elements is shown in Figs. 1 and 2.

Inflation layers were added to capture the boundary layer formation across the wall and body sizing of each domain was done to make smaller sized mesh and similar dense mesh because it is the necessary requirement of CHT analyses. These inflation layers can be seen in Fig. 2. Details of mesh sizes are given in Table III.

TABLE III
MESH DETAILS

Nodes	4264346 ≈ 4.2 Million
Elements	2586633 ≈ 2.5 Million

Fig. 1 Mesh view of combustion chamber

Open Science Index, Aerospace and Mechanical Engineering Vol:14, No:12, 2020 publications.waset.org/10011704.pdf

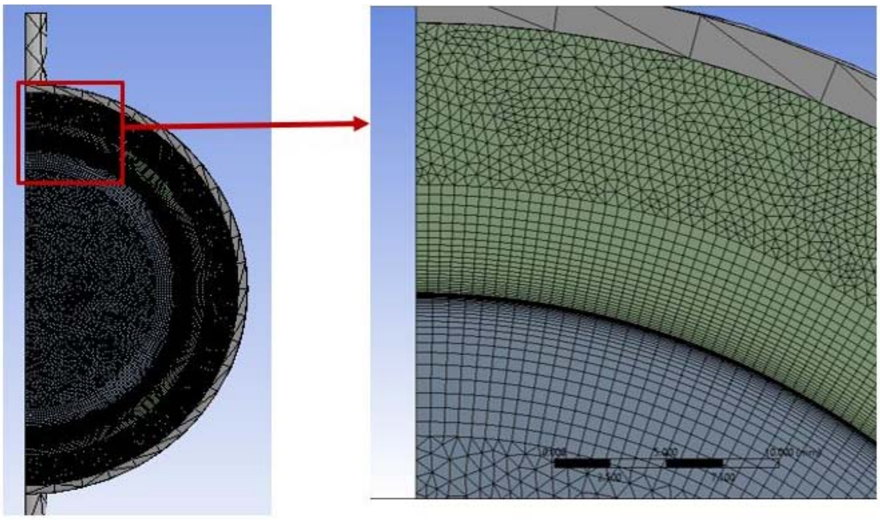

Fig. 2 Mesh view from combustion chamber's entrance

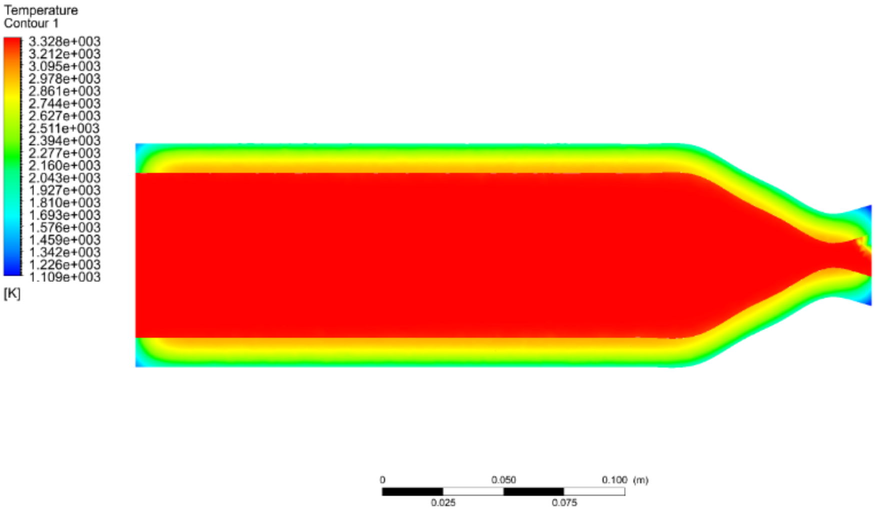

Fig. 3 Temperature contour of combustion chamber without cooling

Table IV gives the detail of body sizing and inflation layers w.r.t the domains.

TABLE IV
INFLATION LAYERS AND ELEMENT SIZE OF MESH

Domain	Inflation Layers	Element Size
Fluid	30	0.00075m
Solid	20	0.0005m

TABLE V
TIME SCALE FACTOR DETAILS FOR CHT

Domain	Time Scale Factor	Domain
Fluid	10	Fluid
Solid	100	Solid

As the problem under consideration is supersonic, total energy model was used and domain interfaces were generated automatically by defining solid and fluid domains. CHT Analysis usually takes a large number of iterations to converge the solution, as flow development in such cases takes high numerical power for computation, hence, parallel run was defined for both (solid and fluid) domains. In CHT, another complexity is with the time scale controls. The Time scale factors defined in CHT simulations are given in Table V.

IV. RESULTS AND DISCUSSION

Multiple models were made with different wall thickness including 0.015 m, 0.012 m, 0.01 m and 0.008 m. Each model was put under CHT analyses with water as a cooling agent with boundary conditions as mass flow rate of 1.75 kg/s at 400 kPa. Temperature variation was observed in each one of the models. It was seen that 0.015 m wall thickness is quite large, as it does not count cooling effect on that side of the wall which is exposed to hot gases and material damage starts to occur on the wall. While 0.008 m and 0.01 m wall thickness allows the cooling effect to lowers the temperature in the wall due to convection though cooling. Hence, the temperature profile obtained in these models was inaccurate as it has been cooled more than the cooling requirement. This was validated by comparing the temperature profile obtained through analytical solution of forward heat transfer. The model with 0.012 m wall thickness was put through CHT and it gave the results with enough cooling without affecting the temperature profile of the wall.

Two cases are analyzed for CHT. In the first case there is no cooling mechanism for the combustion chamber and therefore conduction through the wall is studied at the maximum possible temperatures. This analysis helped in deciphering the idea of efficiency degradation through the melting of wall. Results from this are compared from the one performed with cooling mechanism. Results obtained for the combustion chamber of 0.012 mm wall thickness are discussed below

A. Without Cooling

Hot gases inside a combustion chamber cause the convection to the chamber wall and then heat is transferred in the wall through conduction.

In Fig. 3, it can be seen that temperature range in the chamber wall is high enough to melt SS 316 as this range exceeds beyond the Melting Point of SS 316. Temperature variation across the wall at certain distance from inlet of the combustion chamber is shown in Fig. 4. This can be easily seen that the temperatures in the wall throughout its thickness are out of the range from SS 316 melting point.

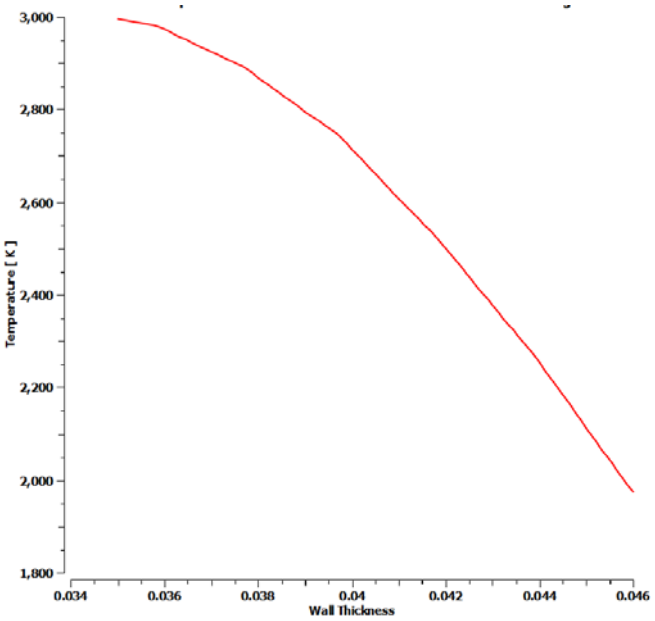

Fig. 4 Temperature variation in combustion chamber wall without cooling

B. With Cooling

Regenerative cooled combustion chamber with a given wall thickness is then analyzed for CHT.

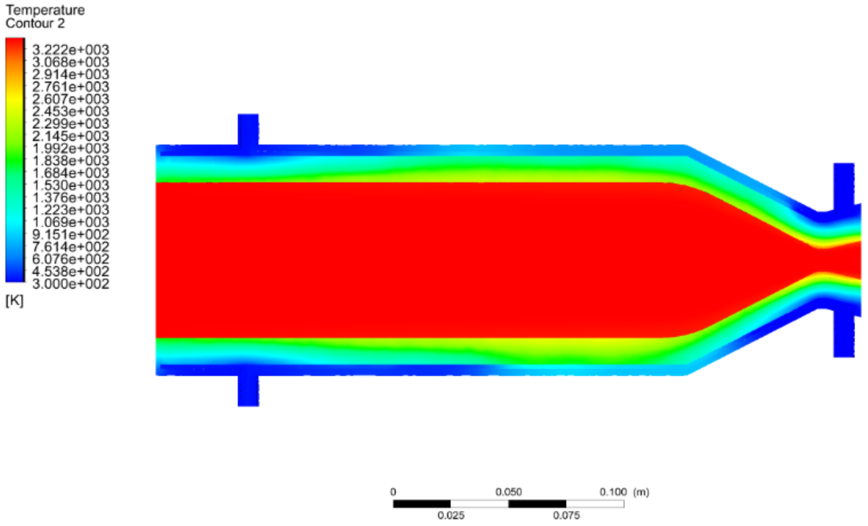

Fig. 5 Temperature contour of combustion chamber after cooling

In Fig. 5, temperature contours inside the cooled combustion chamber are shown. It can be seen that the wall temperature is now lower than the uncooled combustion chamber. Fig. 6 gives the graph of temperature variation in combustion chamber wall. Here, it can be seen that the temperatures now lie in the working range of thermocouple.

V. CONCLUSION

CHT Analyses performed in ANSYS CFX using total energy model and applying domain interfaces gave the temperature variation in combustion chamber wall. The temperatures go beyond SS 316 melting range and thermocouples working range if the combustion chamber is not cooled. But if a cooling mechanism is integrated on the combustion chamber, the temperature in the chamber wall comes under the working range of the K type thermocouples.

The CHT Analyses showed that at 0.004 m of wall thickness from inside the combustion chamber, temperatures start to lie in the working range of thermocouple. Hence thermocouples can be placed at this distance to measure the temperatures at that point for safe use, for inverse heat transfer analyses.

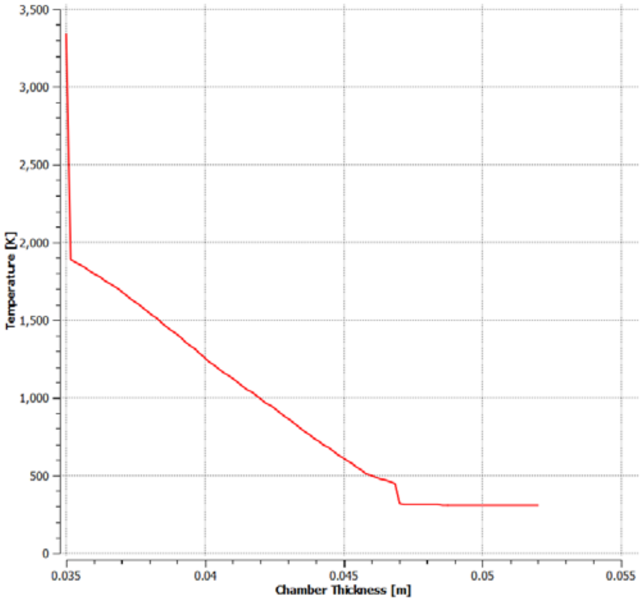

Fig. 6 Temperature variation in combustion chamber after cooling

REFERENCES

[1] R. Warren, "Design of thermocouple probes for measurement of rocket exhaust," Aeronautical and Maritime Research Laboratory, Victoria, Australia, 1994.

[2] K. G. A. K. S. Omori, "Wall temperature distribution calculation for a rocket nozzle contour," National Aeronautics and Space Administration, Washington, United States, 1972.

[3] M. N. Sabry, "Modeling Conjugate Heat Transfer," in 2010 3rd International Conference on Thermal Issues in Emerging Technologies Theory and Applications, Cairo, Egypt, 2010.

[4] C. K. Yuanhong Li Song, "Coupling conjugate heat transfer with in-cylinder combustion modeling for engine simulation," International Journal of Heat and Mass Transfer, vol. 54, no. 11-12, pp. 2467-2478, May, 2011.

[5] P. S. L. H. M. Fadl, "Full conjugate heat transfer modelling for steam turbines in transient operations," International Journal of Thermal Sciences, vol. 124, pp. 240-250, Feburary, 2018.

[6] M. F. L. He, "Multi-scale time integration for transient conjugate heat," International Journal for Numerical Methods in Fluids, vol. 83, no. 12, pp. 887-904, 2016.

[7] T. Perelman, "On conjugated problems of heat transfer," International Journal of Heat and Mass Transfer, vol. 3, no. 4, pp. 293-303, 1961.

[8] M. F. a. L. He, "On Large Eddy Simulation Based Conjugate Heat Transfer Procedure for Transient Natural Convection," Journal of Turbomachinery, vol. 139, no. 11, June, 2017.

[9] "ThermocoupleInfo.com," REOTEMP Instrument Corporation, (Online). Available:https://www.thermocoupleinfo.com/type-k-thermocouple.htm. (Accessed 25th August 2020).

[10] O. B. George P. Sutton, Rocket Propulsion Elements, A Wiley Inter-Science Publication.

[11] "ASM Aerospace Specification Metals Inc.," MatWeb, LLC, (Online). Available: http://asm.matweb.com/search/SpecificMaterial.asp?bassnum=MQ316A (Accessed 4th October 2020).