Comparison of empirical correlations for the estimation of conjugate heat transfer in a thrust chamber

M.J. Iqbal[1], N.A. Sheikh[2], H.M. Ali[1], S. Khushnood[1], M. Arif[1]

[1.] Department of mechanical engineering, University of engineering and technology, Taxila, Pakistan.
[2.] Department of mechanical engineering, Muhammah Ali Jinnah University, Islamabad, Pakistan.
ndahmed@gmail.com

Abstract: Due to high temperatures and pressures in the thrust chamber, regenerative cooling along with film cooling is one of the basic requirements for safe operation. This is ensured by controlling the rate of heat transfer from the walls of chamber through the coolant flow rate. For optimum performance of thrust chamber using efficient combustion, specific heat transfer rates through the nozzle section are required to guarantee the structure integrity of the chamber. Analytical procedures for the thermal design of thrust chambers are fairly limited and designers have to rely on empirical relationships and/or computational methods to calculate the heat transfer rates. Experimental correlations are usually used to predict heat transfer through the internal wall however the exact mechanism of heat transfer is not fully understood. Here a comparison of analytical and empirical approaches has been made for a simplified geometry consisting of two concentric shells. The simplified geometry allows direct application of analytical approach and provides a test ground for the empirical approaches. Results for heat flux and hot side wall temperatures are also compared with a coupled numerical simulation using commercial software Fluent. While estimations for cooling fluid outlet temperature and temperature of outer wall of coolant shell are also compared with the experimental data. The comparison indicates that the analytical method for the heat transfer calculation matches the numerical simulations and experimental data reasonably well.
[Iqbal M.J, Sheikh N.A., Ali H.M., Khushnood S., Arif M. **Comparison of empirical correlations for the estimation of conjugate heat transfer in a thrust chamber**. *Life Sci J* 2012;9(4):708-716] (ISSN:1097-8135). http://www.lifesciencesite.com. 111

Keywords: Heat transfer; Thrust chamber; Bartz correlation; Dual shell.

1. Introduction

The modern liquid rocket engine thrust chambers are exposed to high pressure and high temperature environments. The flow in the thrust chamber is turbulent and supersonic. Reducing wall temperature at hot-gas side wall by 50-100^{o}C could result in doubling the chamber life cycle, which is very important to the industry of spaceflight due to the expensive cost of manufacturing. Therefore, ways to enhance the cooling rate of thrust chamber is very important to the rocket engine while keeping the performance of chamber to its optimal level. The heat flux varies along the thrust chamber wall according to geometry and design parameters of thrust chamber. In a typical design maximum heat flux is found near the zone where the area is reduced to its minimum; while the lowest values are usually observed near the nozzle exit of the thrust chambers.

Regenerative cooling is the most widely used method of cooling a thrust chamber and is accomplished by flowing high-velocity coolant over the sides of the chamber hot gas wall to convectively cool the hot gas liner. The coolant is normally the fuel used for the chamber, as it takes the heat by cooling the liner and getting discharged into chamber through injector and utilized as a propellant. The heat flow through the chamber wall is usually very high ranging up to 20 MW/m^2. The amount of heat that can flow into the coolant is controlled by many factors including the temperature difference between the chamber and the coolant, the heat transfer coefficient, the thermal conductivity of the chamber wall, the velocity in the coolant channels and geometrical design of the flow channels with the flow velocity of the hot gas in the chamber. However the flow rate of the coolant is usually limited to design constrains of the number of channels, channel velocity and chamber fuel requirements, as in most of the cases, the coolants are the propellant itself. In complex designs, the flow path is usually intricate and results in highly turbulent flows in the channel geometry. While the gas flow in the nozzle is also highly turbulent and supersonic. The optimal performance and structural integrity are highly difficult to obtain at times and one may need to rely on extensive experimentation to obtain required performance.

Due to the intricate geometric configurations and temperature dependant physical properties of fluids involved generally empirical formulations have been adopted for heat transfer calculations. For practical thrust chamber design Rocket Thermal Evaluation (RTE) code [1] and Two Dimensional Kinetics (TDK) nozzle performance code [2] are commonly used. In addition to empirical heat transfer models a number of computational and experimental testing are also employed to optimize the coolant channel design so the chamber wall temperatures can be limited without influencing other critical parameters [3, 4]. Several investigations of the regenerative cooling and optimization have been performed using numerical codes (for instance see Carlos et al. [5], Niu et al. [6], Han [7], Li and Liu [8], and Toyama [9]). However mostly the inner chamber gas flow and associated heat transfer is modelled using empirical formulations with very few exceptions including coupled regenerative cooling for simplified configuration (for instance see Li and Liu [8]).

Further enhancements of the empirical one-dimensional model for flows in a rocket engine with regenerative cooling are also introduced. For example, Carlos et al [5], highlighted the importance of temperature varying fluid properties in the estimation of maximum inner wall temperature of the gas side. Naraghi et al [9] approximated the heat conduction in the engine wall by modeling the channel walls as fins. Sailesh et al [10] used commercial software to estimate the temperature and pressure distribution of the gas side for heat transfer calculations.

The enhanced relationship is further used for calculating aspect ratios of the cooling channel at various mass flow rates and pressure drops by Boysan et al [11]. However the exact nature of the heat transfer is still unclear especially in the presence of film cooling often introduced in large scale systems. Merkle et al [6] and Naraghi et al [12] studied the three-dimensional flow in the regenerative cooling passages along with film cooling resulting in lower maximum wall temperatures than usual. For conjugate heat transfer extensive experimentation and numerical simulations have been used to develop empirical correlations such as Namkoung et al [7]. Despite extensive experimental and numerical simulations, the heat transfer physics in complex geometric configuration is not well understood. The flow physics is often turbulent and requires extensive computational resource to resolve before estimations for heat transfer could be made. On the other hand one-dimensional empirical correlations cannot take some important effects (such as geometric and flow physics effect on heat transfer) into account, thus ad hoc design methods are used before experimental testing can be performed.

Understanding the heat transfer phenomena using first principles is important for safe design and optimal performance. The conventional designs often involve intricate paths for cooling channels to maximize the contact area between two sides but this limits the scope for understanding the physics. Here we propose a simplified geometry for coolant transport in a thrust chamber to study the heat transfer. The geometry involved two concentric shells (see Fig. 1) for gas side and coolant side respectively. This is the most simplified configuration with lowest area of restriction for the coolant (less than 4% for assembling using spot welds). One dimensional heat transfer model is used for radial heat transfer along the length of the chamber. Numerous empirical correlations for the estimation of heat transfer coefficient have been proposed due to complex flow physics, for instance Bartz Equation [13].

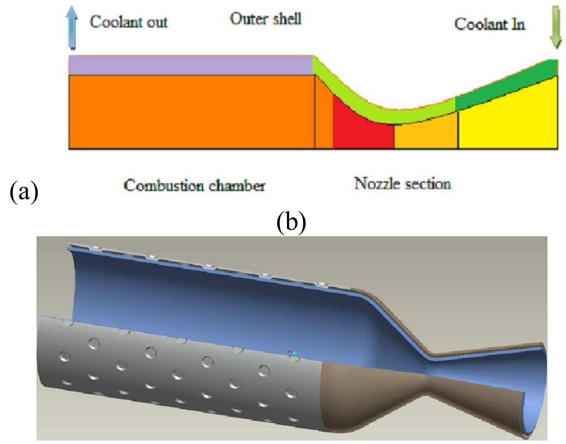

Figure 1: a. Layout of the Dual shell thrust chamber b. Cad model of dual shell chamber.

In the simplified geometry we test theoretical model for the estimation of convective heat transfer coefficient along with different empirical correlations. Results are compared with the numerical simulations using axi-symmetric geometry in commercial software. Temperature of the coolant at the exit from the chamber is also measured experimentally to validate the models. This allows a direct comparison between the experimental setup and analytical model in a realizable configuration.

2. Analytical and Empirical Models

Heat transfer in a regeneratively cooled chamber can be described as the heat flow between two moving fluids, through a multilayer partition as given below and shown in Figure 2.

$$\dot{q}_{tot} = \dot{q}_g = \dot{q}_s = \dot{q}_c \tag{1}$$

2.1 Gas Side Heat Transfer

The heat transfer between the combusted gases and thrust chamber wall is through convection and radiation.

$$\dot{q}_g = \dot{q}_{g,conv} \tag{2}$$

In thrust chamber, before the combusted gases can transfer heat to the wall, the heat energy must pass through a layer of stagnant gas along the wall, boundary layer. This basic correlation for this complicated convective heat transfer can be expressed by the following equation:

$$\dot{q}_{g,conv} = h_g (T_{aw} - T_{wg}) \quad (3)$$

The adiabatic wall temperature of combustion gas at a given location in the thrust chamber may be obtained from the following expression:

$$T_{aw} = T_c \left[\frac{1 + r\left(\frac{\gamma - 1}{2}\right) M^2}{1 + \left(\frac{\gamma - 1}{2}\right) M^2} \right] \quad (4)$$

where recovery factor 'r' can be estimated for turbulent flows as:

$$r = (P_r)^{0.33} \quad (5)$$

Conventional formulation for the calculation of gas side heat transfer coefficient is usually given in terms of several dimensionless parameters

$$\frac{h_g D}{k} = 0.026 \left(\frac{D v \sigma}{\mu} \right)^{0.8} \left(\frac{\mu C_p}{k} \right)^{0.4} \quad (6)$$

where $Nu = \frac{h_g D}{k}$, $\mathrm{Re} = \frac{D v \sigma}{\mu}$, $P_r = \frac{\mu C_p}{k}$

In simplified form h_g can be calculated as follows

$$h_g = 0.026 \frac{(\sigma v)^{0.8}}{D^{0.2}} P_r^{0.4} \left(\frac{k}{\mu^{0.8}} \right) \quad (7)$$

Determination of gas side heat transfer coefficient presents a very complex problem. Comparisons of analytical results with experimental heat transfer data have often shown disagreements. The differences are largely attributed to the initial assumptions for analytical calculations. Based on experience with turbulent boundary layer, some relatively simple correlations for the calculation of gas side heat transfer have been developed. Bartz Correlation [13] is a well known equation used for estimation of rocket nozzle convective heat transfer coefficients based on thermal properties of combusted gases and isentropic gas equations. Heat transfer coefficient can be estimated in terms of gas side wall temperature by using Bartz Correlation.

$$h_g = \frac{0.026}{d_t^{0.2}} \left(\frac{\mu_g^{0.2} C_{p,g}}{\mathrm{Pr}_g^{0.6}} \right) \left(\frac{P_c}{C^*} \right)^{0.8} \left(\frac{A_t}{A} \right)^{0.9} \sigma \quad (8)$$

$$\sigma = \left[0.5 \frac{T_{wg}}{T_c} \left(1 + \frac{\gamma - 1}{2} M^2 \right) + 0.5 \right]^{0.68} \left(1 + \frac{\gamma - 1}{2} M^2 \right)^{-0.12} \quad (9)$$

Apart from the Bartz correlation, based on the experimental studies of Ciniaref and Dobrovoliski [14] the relationship for convective heat transfer is also given as,

$$h_g = \frac{k_g}{d} 0.0162 P_r^{0.82} \mathrm{Re}_g^{0.82} \left(\frac{T_{aw}}{T_{wg}} \right)^{0.35} \quad (10)$$

For initial benchmarking results for Bartz correlation is presented, however a comparison is also provided later.

2.2 Coolant Side Heat Transfer

The heat transfer between the coolant and thrust chamber wall is by forced convection.

$$\dot{q}_{l,conv} = h_l (T_{wl} - T_l) \quad (11)$$

The correlations used for coolant side heat transfer are principally based on the conventional Dittus-Boelter equation for turbulent, thermally fully developed flow for fluids with constant property values. The following correlation is generally used for regenerative cooling analysis as given in equation (12) [15].

$$Nu = \frac{h_l D_h}{k_l} = 0.027 \mathrm{Re}_l^{0.8} \mathrm{Pr}_l^{0.33} \left(\frac{\mu_l}{\mu_{l,cw}} \right)^{-0.14} \quad (12)$$

3. Solution technique

The thermal flux received by the wall of the thrust chamber is the flux due to convection. Using equation (1),

$$\dot{q}_t = \dot{q}_c = h_g \left(T_{aw} - T_{wg} \right) \quad (13)$$

In steady state, this flux $\dot{q}_t$ passes through the first metallic wall of thickness t_{win} so that if k_{win} is the thermal conductivity of the wall material, we get:

$$\dot{q}_t = \frac{k_{win}}{t_{win}} \left(T_{wg} - T_{wl} \right) \quad (14)$$

In addition for coolant liquid, we know that:

$$\dot{q}_t = h_l \left(T_{wl} - T_l \right) \quad (15)$$

where h_l is determined using the Nusselt number (Eq. 12). The three relations given above determine the temperatures T_{wg} and T_{wl} , i.e., wall temperatures along the gas and liquid sides respectively. For calculations, the length of thrust chamber is divided into small segments. Each segment is assumed to have a constant heat flux. Heat balance is calculated for each segment with T_{in} at the inlet with T_{out} is the exit temperature of the segment. Heat taken by coolant in first segment is

$$(T_{out})_i = (T_{in})_i + \frac{qA}{C_{pcool}\dot{m}} \quad (16)$$

For all the elements,

$$T_{out} = T_{in} + \sum_i \frac{qA}{C_{pcool}\dot{m}} \quad (17)$$

where C_{pcool} is the mean specific heat of coolant and $\dot{m}$ is mass flow rate of coolant. The solution is obtained using an iterative procedure. The calculation starts with a guess value of T_{wg} and is updated using the average of the newly calculated values and its previous value, mathematically:

$$(T_{wg})_2 = \frac{T_{wg} + (T_{wg})_1}{2} \quad (18)$$

Initially, using the assumed value of T_{wg} at each station T_{wg}, h_g and $\dot{q}_t$ are calculated using equations (7) and (15) respectively. For different empirical correlations respective equations/models are used instead of equation (7). The calculated heat flux is used for the calculation of temperature distributions, i.e., T_{wg}, T_{lw}, T_{wl} and T_l. An update value of T_{wg} is obtained by taking the average of previous and newly calculated value of T_{wg}. This new value of T_{wg} is used for next loop and procedure is continued until required convergence is achieved. Any values of T_{wg} can be assumed, however, clever guess of T_{wg} can reduce the number of iterations. The solution is obtained using MATLAB. The hot gas in the thrust chamber is assumed to be composed of 9 constituent species. Table 1 shows the molar fractions of the species used for the calculations.

The properties of the constituent (such as heat capacity, viscosity etc) are averaged before using in the model and/or experimental correlation. The values of the temperature dependant properties are initially calculated using the guess values of the temperature. The values are continuously updated through the iterative scheme to obtain the steady state temperature at the wall of thrust chamber

Table 1: Molar Fraction of species

Species	Molar fraction
O_2	0.00137
N_2	0.282
H_2	0.1067
CO	0.163
CO_2	0.069
H_2O	0.344
NO	0.00259
OH	0.016
H	0.129

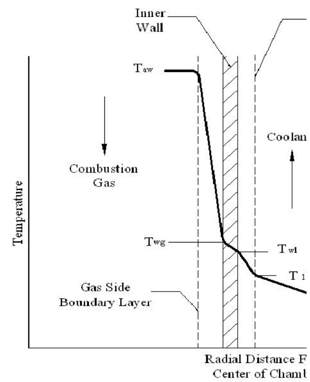

Figure 2. A schematics of the Temperature gradient in a Thrust chamber

3.1 Initial validation

For initial validation of the code, a comparison is made for the prediction of heat flux on the gas side of the chamber. The details for the thrust chamber used for the comparison can be found in [13]. The designed chamber pressure is 3.818 kPa with propellants LH2/LO2 and mass flow rate of propellant 18.599 kg s-1. The chamber uses LH2 for regenerative cooling. Figure 3 shows the comparison for the heat flux calculation using Bartz correlation and published data [13]. At the nozzle section (also shown in the Fig. 3), peak value of heat flux is found 25.8 MW/m2 compared to 28 MW/m2 found in [13].

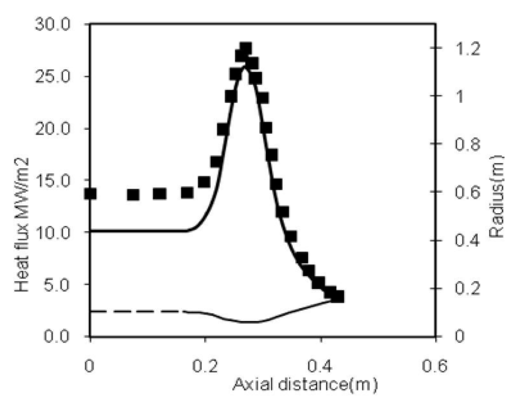

Figure 3. Comparison of heat flux using Equation (8) and results published in Ref [13]

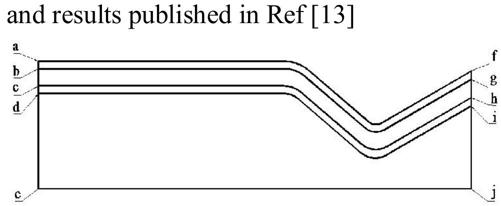

Figure 4. Schematics of the thrust chamber with labels indicating different boundaries. Details of the boundaries are mentioned in Table 2.

Table 2. Details of the boundaries indicated in Fig. 4

Section	Designation	Condition
abfg plane	Outer wall	Solid SS
bcgh plane	Coolant zone	Water
cdhi plane	Inner wall	Solid SS
deij plane	Hot gas zone	Ideal gas
de boundary	Pressure inlet	Pc=50 bar, Tc= 3000K
ij boundary	Pressure outlet	Pe= atm
bc boundary	Coolant pressure outlet	Pregout= 51 bar
gh boundary	Coolant velocity inlet	vin= 3.5m/s
ej boundary	Axisymmetric	axis
af boundary	Convective heat flux	hatm= 10 W/m2k

It can be observed from Fig. 3 that the peak heat flux is reasonably well predicted. The variation in the heat flux along the nozzle section also matches with the data. However higher values of the heat flux are observed in the combustion chamber compare to the predictions of the analytical code. It is worth mentioning here that the combustion chamber was designed with corrugation pattern on the coolant side different from the nozzle section (see ref [13] for details). The result presented here uses the corrugation pattern of the coolant side for the nozzle section only.

4. Thrust Chamber Geometry

The geometry of the thrust chamber used for this work is a simple dual shell. The outside diameter of the combustion chamber is 85mm with the length of the 195mm. The nozzle section starts immediately after the combustion chamber. The minimum diameter of the nozzle section is 18.2 mm located at a distance of 54.3mm from the start of the nozzle section. The total length of the nozzle section is 126.2mm with exit diameter of 54.4mm. The geometry is purposely selected and built for this research work to study the heat transfer in a simplified configuration. The exact profile of the geometry shall be displayed in all the results in following sections. The thrust chamber has a total thrust of 205 N. The inner and outer shells are manufactured using stainless steel joined together using spot welds. The experiments are conducted at an experimental facility equipped with appropriate data acquisition and instrumentations. The chamber is designed to operate with UDMH and N_2O_4 propellants. For the purpose of regenerative cooling, water is used with fixed flow rate of 1.0 kg/s. Details of the experiment, instruments and results are provided in forthcoming section.

5. Numerical Simulations

The numerical simulations are carried out using commercial software. The software solves conservation equations of mass, momentum and energy for compressible flow for the gas side, incompressible flow for the coolant side coupled with the conjugate heat transfer calculations. The simplified geometry can be approximated as two dimensional axi-symmetric. The conservation equations for the problem are given below. The continuity balance is given by:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x}(\rho v_x)+\frac{\partial}{\partial r}(\rho v_r)+\frac{\rho c_r}{r}=S_m \quad (19)$$

The axial and radial momentum equations are given below:

$$\left[\begin{array}{l}\frac{\partial}{\partial t}(\rho v_x)+\frac{1}{r}\frac{\partial}{\partial x}(r\rho v_x v_x)+\frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r v_x)=-\frac{\partial p}{\partial x}+\\ \frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left(2\frac{\partial v_x}{\partial x}-\frac{2}{3}(\nabla\cdot\vec{v})\right)\right]+\\ \frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(\frac{\partial v_x}{\partial r}+\frac{\partial v_r}{\partial x}\right)\right]+F_x\end{array}\right] \quad (20)$$

$$\left[\begin{array}{l}\frac{\partial}{\partial t}(\rho v_r)+\frac{1}{r}\frac{\partial}{\partial x}(r\rho v_x v_r)+\frac{1}{r}\frac{\partial}{\partial r}(r\rho v_r v_r)=-\frac{\partial p}{\partial r}+\\ \frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\left(\frac{\partial v_r}{\partial x}+\frac{\partial v_x}{\partial r}\right)\right]+\\ \frac{1}{r}\frac{\partial}{\partial r}\left[r\mu\left(2\frac{\partial v_r}{\partial r}-\frac{2}{3}(\nabla\cdot\vec{v})\right)\right]\\ -2\mu\frac{v_r}{r^2}+\frac{2}{3}\mu\frac{v_r}{r}\left(\nabla\cdot\vec{v}\right)+\rho\frac{v_z^2}{r}+F_x\end{array}\right] \quad (21)$$

The energy equation is given as follow

$$\begin{array}{l}\frac{\partial y}{\partial x}(\rho E)+\nabla\cdot(\vec{\upsilon}(\rho E+p))=\\ \nabla\cdot(k_{eff}\nabla T-\sum_i h_j\vec{J}_j+(\overline{\overline{\tau}}_{eff}\cdot\vec{\upsilon}))+S_h\end{array} \quad (22)$$

The energy transport equation used by FLUENT in solid regions has the following form:

$$\frac{\partial}{\partial t}(\rho h)+\nabla\cdot(\vec{\upsilon}\rho h)=\nabla\cdot(k\nabla T)+S_h \quad (21)$$

For the turbulent flow with heat transfer, the Renormalization Group Theory (RNG) $k-\varepsilon$ model is used. The RNG $k-\varepsilon$ model has an additional term in its ε equation that significantly improves the accuracy for rapidly strained flows suitable in the current case.

Table 3: Grid independence study

	Hot gas	Inner wall	coolant	Outer wall	Exit Ma.	Outlet Temp (K)
1	25×100	5×100	5×100	5×100	2.92	328.7
2	50×300	10×300	10×300	10×300	3.1	328.9
3	50×600	15×600	15×600	15×600	3.1	329.0

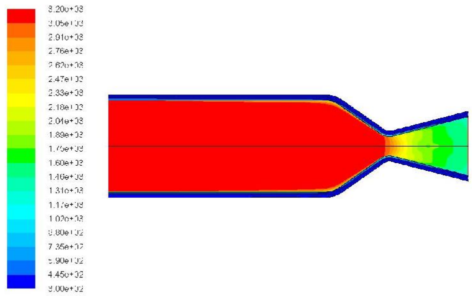

Figure 5. Contour plot showing variation of the steady state Temperature (K) in the thrust chamber along with coolant channel.

5.1 Boundary Conditions and Zones

The boundaries of the thrust chamber with different zones are mentioned in Figure 4. The chamber has two flow channels for gas and propellant flows respectively. While the solid wall separates the gas and liquid side and allows the heat transfer through conduction.

Commercial software Fluent is used for coupled simulation of gas and regenerative cooling fluid. For the thrust chamber analytical engine parameters are used as input from CEA software [17]. CEA calculates the resultant temperature and pressure which can directly be used as inlet boundary conditions. DTRM model is used for radiation heat flux calculated. Turbulence model is used with peak value of y^+(non-dimensional vertical height) less than 120. Initially three different grid sizes were used to carry out Grid independence study. The results for nozzle exit Mach number and outlet temperature of coolant are shown in table 3 for three grids.

It was noted that the variation in the outlet temperature and Mach no. is negligible for the case 2 and case 3. Therefore case 3 mesh is selected for further calculations and comparison.

The steady state variation in the temperature of the thrust chamber is shown in Figure 5. It can be noticed that the hot gases from the combustion chamber loses temperature as it flows through the nozzle section of the thrust chamber. The maximum heat transfer occurs in the region where the chamber diameter is reduced to its minimum. It is also seen that the hot gas loses temperature as the nozzle section expands after the throat. The coolant is introduced from the exit side. The heat transfer occurs along the nozzle section and resultant temperature on the coolant side increases as it comes out from the combustion side of the thrust chamber.

Figure 6 and 7 shows the variation of heat flux and the temperatures of the wall at the gas side, coolant side and the outermost side of the thrust chamber.

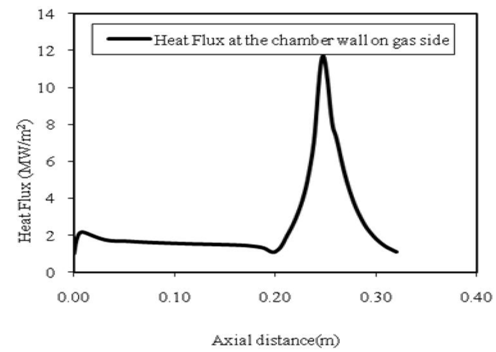

Figure 6. Variation of total heat flux along the length of the thrust chamber.

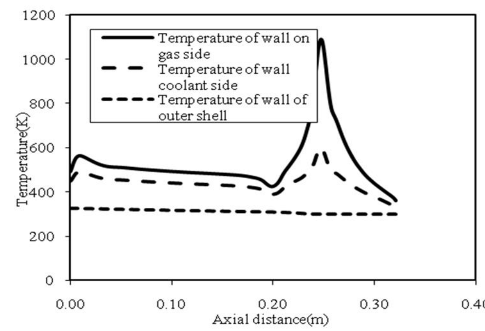

Figure 7. Variation of temperature along the length of the thrust chamber for a) the wall on the gas side, b) the wall on the coolant side c) outer most wall.

It can be observed from Figure 6 that the maximum heat transfer occurs near the throat region of the nozzle. In addition, highest temperature is also observed in the same region as seen in Figure 7. One may notice that the peak temperature is ~1200 K. For sustained working the peak temperature of the thrust chamber must be lowered to a safe limit. This can be controlled using the mass flow rate of the coolant. However in practice the mass flow rate of the coolant is controlled by the combustion process inside the chamber. Thus better estimation of the thrust chamber heat flux and wall temperatures is essential for the longer life of the chamber as well as for its better performance.

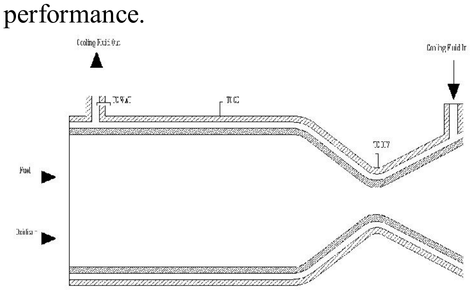

Figure 8. Schematic of experiment conducted along with the locations of the instruments installed.

6. Experimentation

For validation of the proposed technique, a specially designed thrust chamber was fired. For generating high temperature gases inside the chamber, UDMH and N2O4, are used as propellants. Water is used for the regenerative cooling. The thrust chamber is made up of high conductivity stainless steel inner and outer shell. Three thermocouples were installed on the chamber; first one on the outer body of thrust chamber almost at the center of the combustion zone, second one on outer side of the divergent part near the throat portion and the third one was directly measuring the temperature of the outlet water of the chamber in the outlet pipe of cooling fluid as shown in the figure 8. Schematic diagram of experimental setup is shown in figure 8. After the hot test of the thrust chamber, temperature data obtained by the three thermocouples and pressure plots are shown in figure 9 and 10 respectively.

The thrust chamber develops a steady pressure of ~ 50bar right after the ignition is made. The test lasts for a little more than 25 sec. The whole duration of the test the thrust chamber experiences constant pressure indicating the controlled environment present for the data acquisition. Using the three thermocouples data, one can observe that the temperature variations become steady in first 5 sec of the test. The temperature measured in the experiment indicates that the coolant acquires water during the first few seconds of the test and remains fairly constant afterwards. This indicates the initial transients of the chamber settle down in a short period of time. Thus the heat transfer rates can be calculated from steady state conditions.

Due to higher temperatures at the inner wall of the chamber and chances of perturbation of flow in the coolant channel, the data of inner wall temperature could not be monitored in this experimental setup.

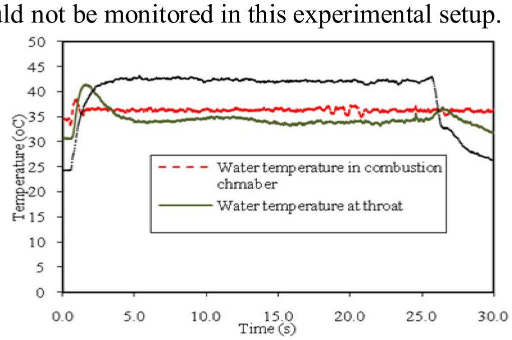

Figure 9. Thermocouple measured data for the temperature variations at the three locations labelled in Figure 8.

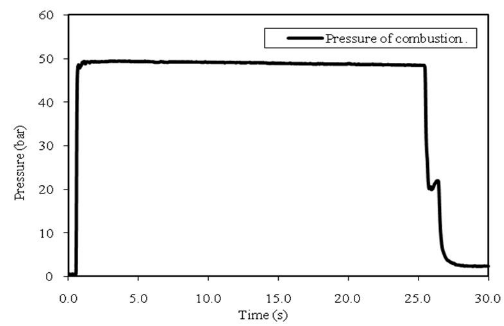

Figure 10. Real time Pressure record data of the thrust chamber.

However the temperature acquired by the coolant is used to calculate the total heat gained during the process. Also the analytical code as well as the numerical simulations produces temperature predictions for the coolant side. In next section we compare the results with experimental data.

7. Results and discussion

The results for the variation of heat flux and wall temperature on the gas side of the chamber are compared with the numerical simulations. Figure 11 and 12 shows the comparison between the predictions using analytical code with numerical simulations for heat flux and gas side wall temperature respectively. As anticipated the variation of heat flux along the chamber length shows a peak around the throat region. Correspondingly highest temperature is also observed in the same region.

The prime focus is the region where the gas accelerates to generate thrust for the chamber. The nozzle section is the most critical part of the chamber. High heat flux and resultant high temperatures may cause damage to the nozzle section. Thus any design calculations shall account the heat flux experienced by the chamber

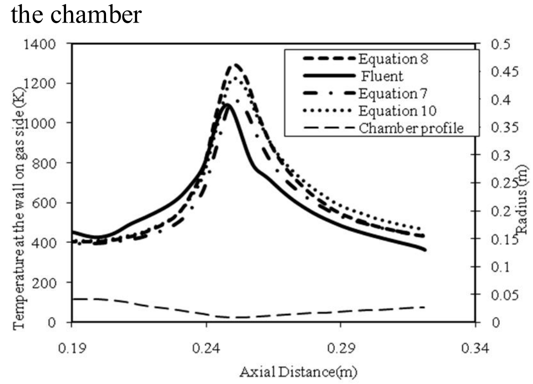

Figure 11. Comparison of temperature on the gas side of the thrust chamber.

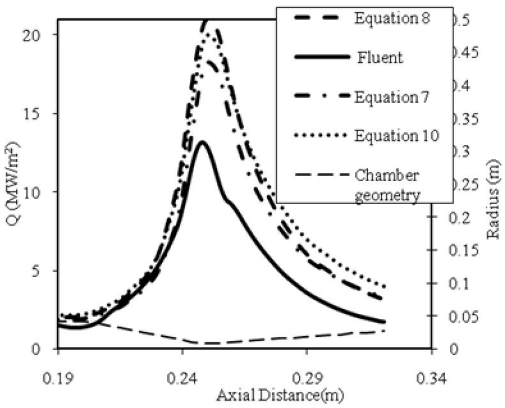

Figure 12. Comparison of heat flux on the wall of the thrust chamber.

. The temperatures in the nozzle section are high and slight deviation from the designed temperature may lead to structural damage. The comparison carried out here shows that different correlations predict highest temperature ranging from ~1300-1150 K. One can observe that the peak temperature predicted using equation 7 matches well with the numerical simulations. While the predictions of commonly used correlation of Bartz (equation 8) shows ~20% higher temperature in the throat region.

Apart from the peak temperature predictions, the variation in the temperature along the nozzle section and location of the peak temperature point varies in all the cases. The empirical correlation of Bartz (equation 8) predicts the peak temperature earlier compared to other formulations. Similarly Bartz predicts a slightly quick decrease in the temperature compared to numerical simulations, while the predictions using equation 7 matches well with the numerical simulations. One may also observe that using the simplified geometry the theoretical models are relatively best suited for the predictions of heat flux (see Figure 12). On the other hand the predictions using Bartz (equation 8) correlation are conservative predicting very high heat flux compared to numerical simulations. The predictions may well suit the designer's guideline for safe operation of the chamber; however the predictions may not be entirely correct or representative of the system.

Table 4. Comparison of wall temperature on the outer shell and coolant outlet temperature

	Eq. 7	Experimental	Fluent
Max wall temperature K)	1102	-	1080
Coolant exit temperature K	315.5	315	330
Outer wall temp of CC K	306	309	321
Outer wall temp of Divergent K	300	307	300

Comparison of wall temperature on the outer shell is presented in Table 4. Analytical predictions (using Equation 7) of the outer shell temperatures and coolant temperature show a reasonable match with the experimental findings. One may notice that the temperature of the coolant at the exit is 315 K in comparison with 315.5 K predicted by the code. This suggests that the analysis carried out here is valid and the predictions of the analytical method are reliable. The analytical predictions using equation 7 may be used for the design guidelines. However with the change in coolant flow configuration and net effective heat transfer area, the predictions of the analytical model using equation 7 may no longer remains reliable. A systematic study may reveal necessary modifications required for the correct form of the heat transfer. One the other hand, empirical predictions provide a conservative guideline for safe operations. But the entire physics may change with different geometrical features and flow physics.

8. Conclusion

Optimal design of thrust chambers requires correct estimation of heat flux experiences by the material of the chamber during operations. A number of empirical relationships have been proposed in the literature and can be used as a design guideline. However complete understanding of the heat transfer phenomena is a must for optimal and safe operations of the system. Here a comparison is made between the predictions of theoretical formulation and empirical relationships along with results using numerical simulations and experimentation. The geometrical configuration is kept simple to ensure validity of assumption and simplification of flow geometry. The dual shell configuration allows a direct comparison between the results of numerical simulations and analytical formulation. It is evident from the results that the heat transfer is dominated by the convective heat exchange between the hot gas and coolant. The form of the heat transfer coefficient is correctly estimated using analytical formulation. On the other hand other formulations show slight deviations. Although the deviations seem slight, but in the context of the geometrical configuration used for the study, the deviations are as high as 20%. One may notice that the empirical correlations predict higher temperatures, resulting in conservative design guideline for the chamber design. However this research work attempts to model the non-linear heat transfer problem using first principles. A parametric study using additional geometrical configurations may indicate the short comings in the analytical formulation. Rigorous study using numerical simulations and experimentation may yield correct relationship for heat transfer coefficient including possible geometric parameters.

Corresponding Author:
Dr. Nadeem Ahmed Sheikh
ME Department
Muhammad Ali Jinnah University
Islamabad, Pakistan
E-mail: ndahmed@gmail.com

References

1. Naraghi, M.H., "RTE - A Computer Code for Three-Dimensional Rocket Thermal Evaluation", User Manual, Tara Technologies, LLC, Yorktown Heights, NY 2002.
2. Dunn, S.S., Coats, D.E., and French, J.C., "TDK'02™ Two-Dimensional Kinetics (TDK)Nozzle Performance Computer Program", User's Manual, prepared by Software &Engineering Associates, Inc., Dec 2002
3. Sieder, E., N., Tate, G., E., "Heat Transfer and Pressure Drop of Liquids in Tubes", Industrial and Engineering Chemistry, Vol. 28, No. 12, pp. 1492-1453 , Dec. 1936.
4. Takase, K., "Numerical prediction of augmented turbulent heat transfer in an annular fuel channel with repeated two-dimensional square ribs", Nuclear Engineering and Design, Vol. 165 Nos 1/2, pp. 225-37. 1996
5. Carlos Henrique Marchi,Fernando Laroca, Anto´nio Fa´bio Carvalho da Silva and Jose´ Nivaldo Hinckel " Numerical solutions of flows in rocket Engines with regenerative cooling" Numerical Heat Transfer, Part A, 45: 699–717, 2004
6. C. L. Merkle, D. Li and V. Sankaran "Analysis of regenerative cooling in rocket combustors", report, Purdue University, West Lafayette IN.
7. Hyuck-Joon Namkoung, Poong-Gyoo Han, Hwa-Jung Kim, Kyoung-Hun Lee, Young-Soo Kim and Chongam Kim "A Study on Heat Transfer Characteristics of Small Liquid Rocket Engine with Calorimeter" 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit 9 - 12 July 2006, Sacramento, California 2006
8. Li, J.W. and Liu, Y., "Method of computing temperature field in regeneration-cooled thrust chamber", Journal of Aerospace Power, Vol. 19 No. 4, pp. 550-6, 2004.
9. Mohammad H. Naraghi and Matthieu Foulon "A simple approach for thermal analysis of regenerative cooling of Rocket engines", Proceedings of 2008 ASME International Mechanical Engineering Congress and Exposition October 31-November 6, 2008, Boston, Massachusetts, USA.
10. Sailesh.R , G Ayyappan and B Anil " Design and analysis of a liquid rocket engine nozzle with LOX/H2 Propellant" 10th National Conference on Technological Trends (NCTT09) 6-7 Nov 2009.
11. M. E. Boysan, A. Ulas, K. A. Tokerl and B. Seckin "Comparison of Different Aspect Ratio Cooling Channel Designs for a Liquid Propellant Rocket Engine" 2007 IEEE.
12. M.H. Naraghi and S. Dunn and D. Coats "Dual regenerative cooling circuits for liquid rocket engines" AIAA-2004-3852 Joint Propulsion Conference, Fort Lauderdale 2004.
13. Bartz, D., R., "A Simple Equation for Rapid Estimation of Rocket Nozzle Convective Heat Transfer Coefficients", Technical Notes, California Institute of Technology, DA-04-495, 1957.
14. Ciniaref, G., D., Dorovoliski , M., B., "Theory of Liquid-Propellant Rockets", Moscow, 1957.
15. Bartz, D., R., "Survey of Relationship Between Theory and Experiment for Convective Heat Transfer in Rocket Combustion Gases," in *Advances in Rocket Propulsion*, S. S. Penner (Ed.), AGARD, Technivision Services, Manchester, UK, 1968.
16. Zhang Yu Lin, Liu Kun and Cheg Mou Senzhu" Liquid propellant engine kinetic theory and application" Contemporary youth scientific library, 2005.
17. Sanford Gordon, Bonnie J. McBride, "Computer Program for Calculation of Complex Chemical Equilibrium Compositions and Applications II. User's Manual & Program Description", NASA RP- 1311 -P2, 1996.

9/28/2012