10th ANKARA INTERNATIONAL AEROSPACE CONFERENCE **AIAC-2019-152**
18-20 September 2019 - METU, Ankara TURKEY

# REGENERATIVE COOLING OF A LIQUID ROCKET ENGINE WALLS USING CFD-CHT MODELLING

Tacettin Utku Suer[1] and Burak Cenik[2]
TOBB University of Economics and Technology
Ankara, TURKEY

Sitki Uslu[3]
TOBB University of Economics and Technology
Ankara, TURKEY

## ABSTRACT

*Computational fluid dynamics (CFD) simulations of regenerative cooling channels of liquid rocket engine, verified with experimental and simulation studies investigated by Wadel and Meyer in NASA Lewis Research Center Rocket Engine Test Facility, are performed in this paper. Heat transfer from hot gas in the chamber to the wall is modeled by Bartz's equation. Conjugate heat transfer (CHT) is used for prediction of temperature distrinution in the walls.*

## INTRODUCTION

Regenerative cooling is a method for cooling liquid fuel rocket engine walls. The coolant fluid to cool the walls can often be fuel and sometimes oxidizer. The coolant liquid enters to the cooling channels from the near the nozzle exit and cools the hot walls while flowing to the towards the face of injectors (engine dome). The coolant liquid passing through the ducts allows the wall to cool down along the engine, preventing the rocket from reaching the temperature at which the engine walls can be damaged. Generally, the most critical location is the throat region where the highest heat flux occurs. The increase in the temperature of the liquid to the injector results in an increase in the internal energy of the liquid sent to the engine, so that the rocket engine has a positive effect on the speed of the nozzle. Despite these positive effects, the regenerative cooling process reduces the pressure of the liquid exiting the injectors. For these reasons, a good design of regenerative cooling should be to lower pressure drop and hot-gas-side wall temperature [Boysan, 2008; Ulas, 2011].

In this study, experimental previous and numerical results of High Aspect Ratio Cooling Thrust Combustion Chamber is going to be compared with present CFD predictions for validation purposes. After the validation, the another regenerative cooling case is going to be studied.

[1] M.Sc. Student in Department of Mechanical Engineering, Email: tusuer@etu.edu.tr
[2] M.Sc. Student in Department of Mechanical Engineering, Email: bcenik@etu.edu.tr
[3] Asst. Prof. in Department of Mechanical Engineering, Email: suslu@etu.edu.tr

## METHOD

In this study, there are two domains; liquid and solid. Three-dimensional, steady-state and RANS turbulence model were used for fluid flow region. For solid region, three-dimensional, steady-state, heat conduction equation is solved. CHT methodology is employed for solving the temperature in the solid walls. The heat flux between the hot gas and the wall resulting from the combustion was modeled using Bartz equation (eq. 2-3) [Bartz, 1965].

$$q'' = h_g\left(T_{aw} - T_{wg}\right) \tag{1}$$

$$h_g = \left[ C\left(\frac{\mu_g}{D_*}\right)^{0.2}\left(\frac{c_{p,g}}{{Pr_g}^{0.6}}\right)\left(\frac{P_c g}{C^*}\right)^{0.8}\left(\frac{D_*}{r_c}\right)^{0.1}\right]\left(\frac{A_*}{A}\right)^{0.9}\sigma \tag{2}$$

$$\sigma = \left[\frac{1}{2}\left(\frac{T_{wg}}{T_c}\right)\left(1+\frac{k-1}{1}M^2\right)+\frac{1}{2}\right]^{-0.68}\left[1+\frac{k-1}{1}M^2\right]^{-0.12} \tag{3}$$

$$T_{aw} = T_c\frac{\left[1+r*\frac{k-1}{1}M^2\right]}{\left[1+\frac{k-1}{1}M^2\right]} \tag{4}$$

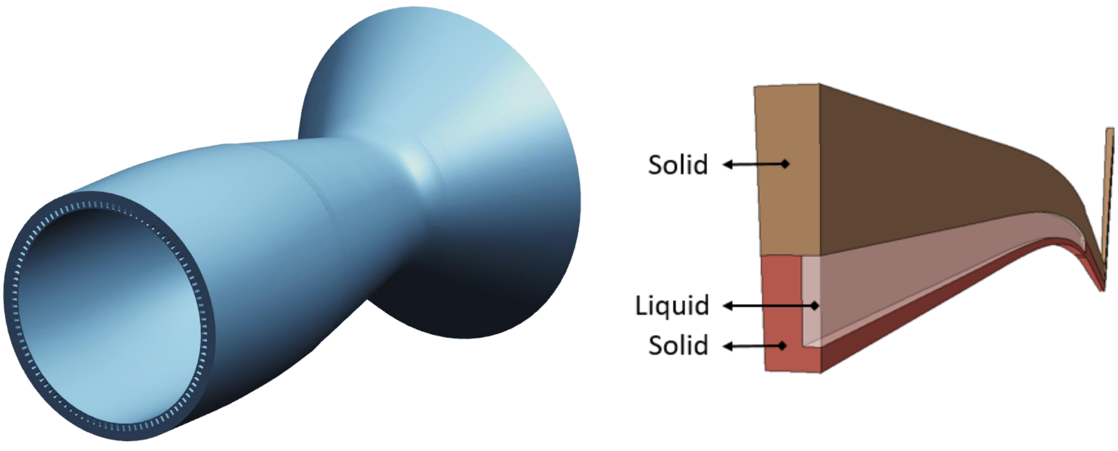
$$r = {Pr_g}^{0.33} \text{ (turbulent flows)} \tag{5}$$

where, $q''$ is heat flux from hot gas to wall, $h_g$ is heat transfer coefficient, $T_{aw}$ is adiabatic wall temperature, $T_{wg}$ is hot-gas-side wall temperature, $D_*$ is throat diameter, $P_c$ is combustor chamber pressure, $r_c$ is radius of throat curvature, $A_*$ is throat area, $A$ is local area, $\sigma$ is correction factor, $T_c$ is combustion chamber temperature, $M$ is local Mach number, $r$ is recovery factor.

Siemens STAR-CCM+ solver is used for CFD-CHT computations.

## VALIDATION CASE

Numerical study was carried out by Wadel, Meyer and Boysan. Present CFD-CHT results are compared with the experimental and available numerical results.

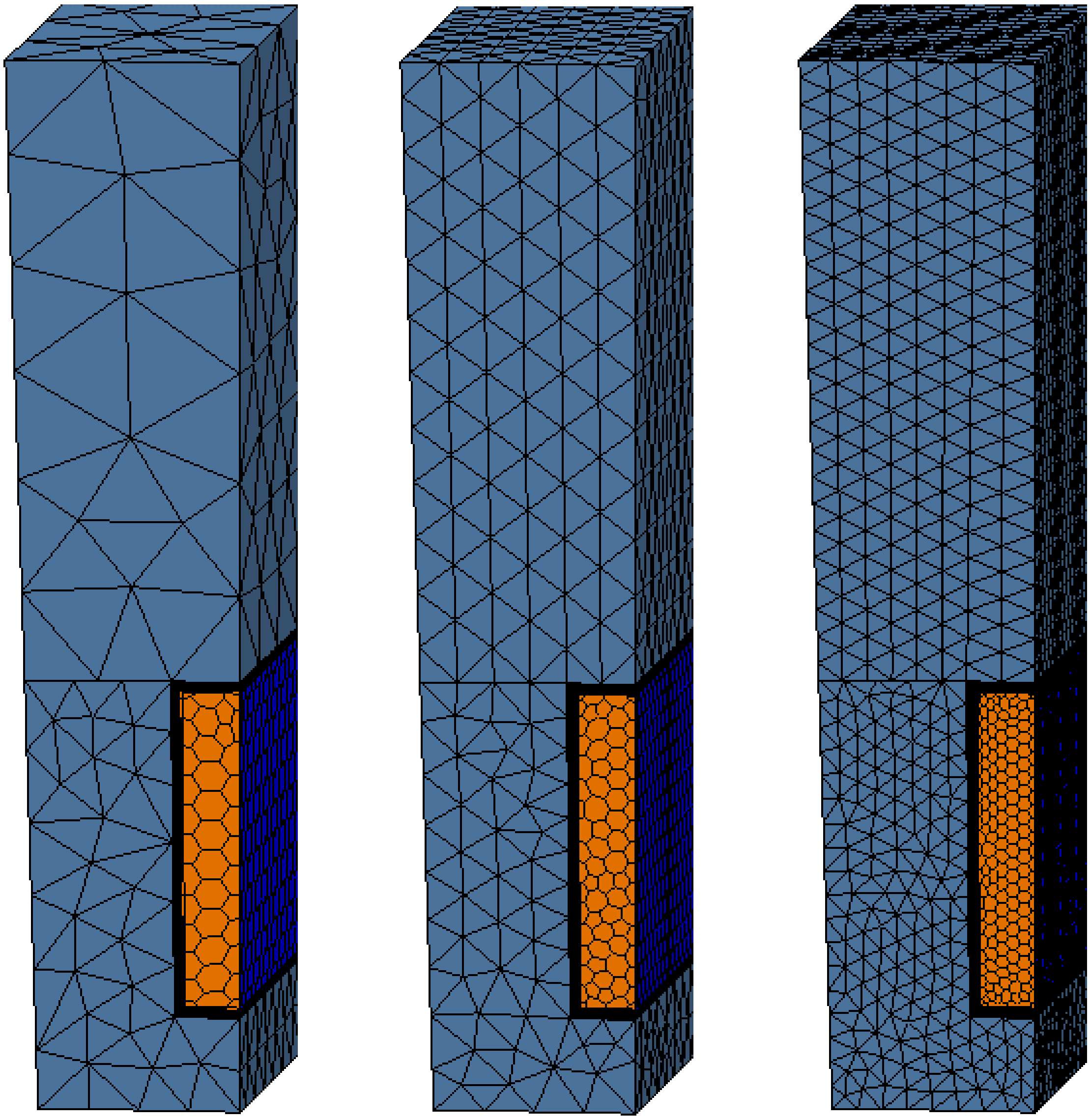



(a) (b)

Figure 1: (a) Geometry of the high aspect ratio cooling thrust combustion chamber, (b) CHT computational domain

The computational domain was taken from the article [Wadel, 1998]. The created domain is seen in the Figure 1a. There are 100 main cooling channels located around circumference of the chamber. The main channels are sub-divide into two channels (bifurcation) for better cooling of the critical region of throat area. Due to the increase in the number of channels, there are high aspect ratio (channel's height/width) in the throat region. Owing to high aspect ratios, the heat flux is increasing and critical region temperatures are reduced, but pressure drop rises.

As seen in the Figure 1b, there are two solid and a liquid regions. Half of the geometry was used for computations by making the use of symmetry boundary conditions.



(a) (b) (c)

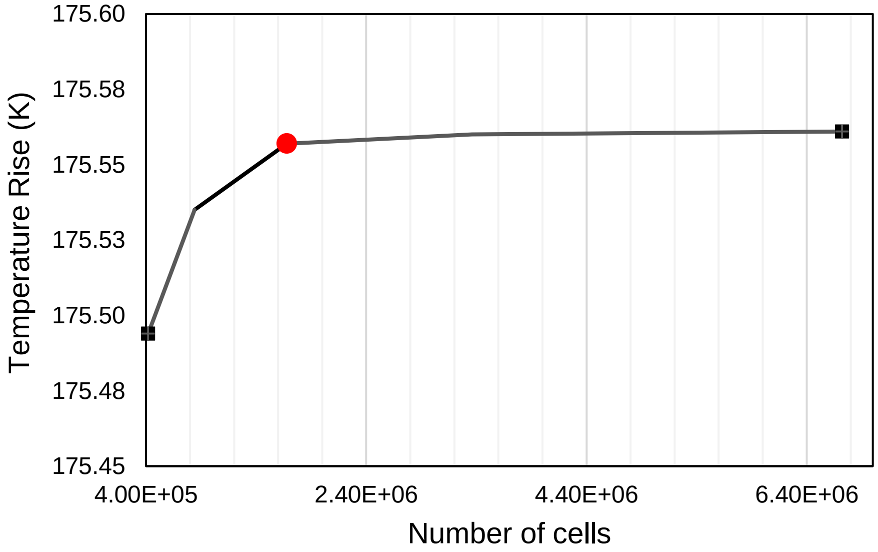
Figure 2: (a) Coarse mesh, (b) Medium mesh, (c) Fine mesh

Coarse, medium and fine mesh scenes are shown in Figure 2. Number of cells for different mesh are in Table 1. The number of cells are approximately increased fourfold for mesh study.

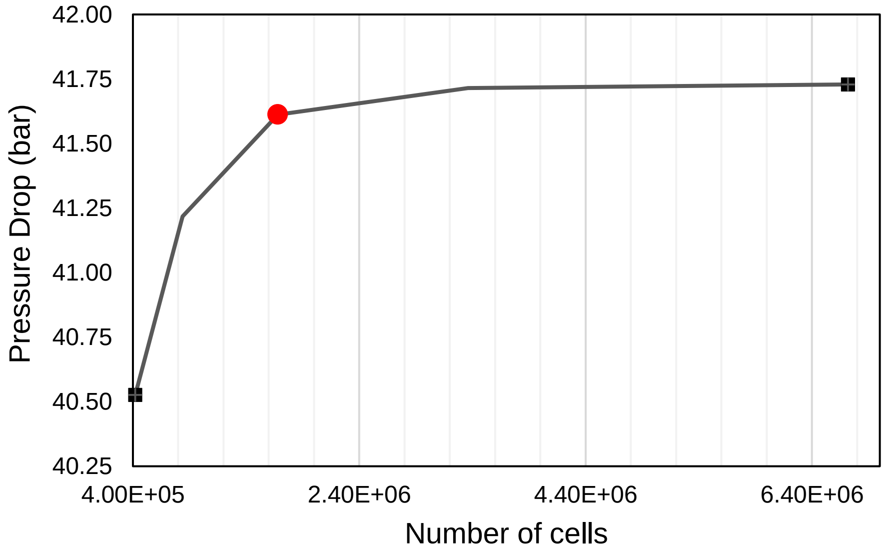
Table 1: Number of cells

| Number of cells | Liquid domain | Solid domain | Total |
|---|---|---|---|
| Coarse mesh | 300,000 | 130,000 | 430,000 |
| Medium mesh | 1,200,000 | 500,000 | 1,700,000 |
| Fine mesh | 4,800,000 | 1,900,000 | 6,700,000 |

The red dot shown in the Figure 3 represents the medium mesh and the values that are temperature rise and pressure drop do not change much after the point. Considering the CPU cost, the medium mesh was selected for further analysis.




(a)




(b)

Figure 3: (a) Temperature rise and (b) pressure drop for different mesh numbers

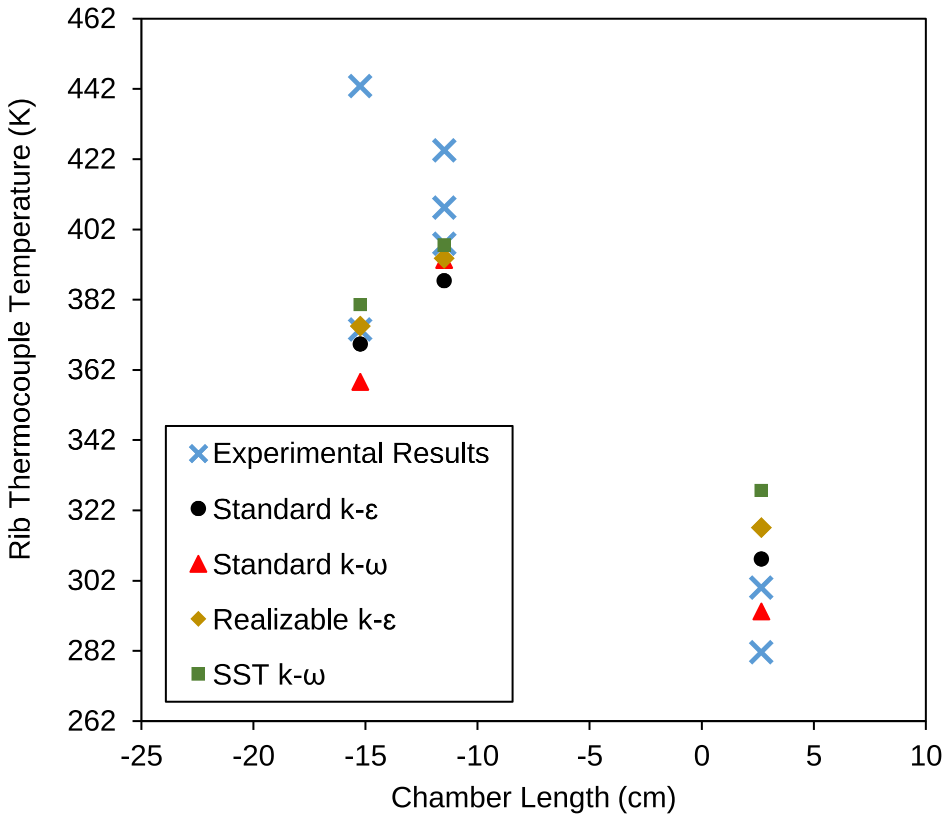



Figure 4: Rib thermocouple temperature for different turbulence models

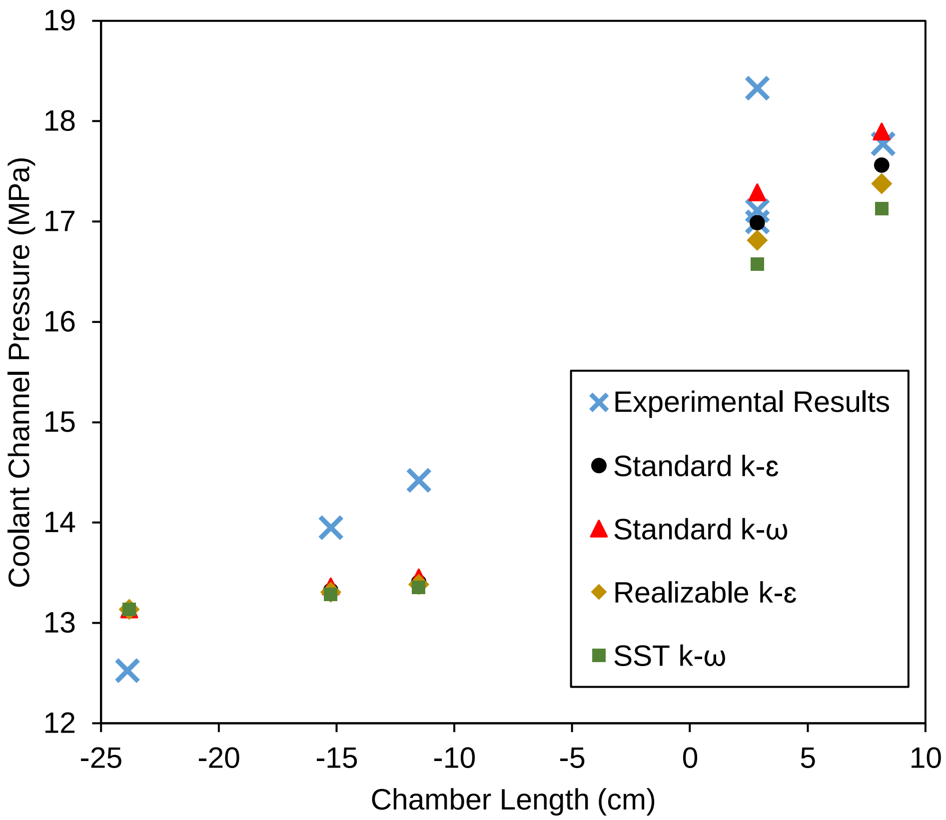



Figure 5: Coolant channel pressure for different turbulence models

As shown in Figure 4 and 5, the turbulence models used in this study generally showed a similar trend. The k-ε family predicts better results the k-ω family in this study. When compared the k-ε family in itself, results of the Realizable k-ε is better than the Standard k-ε at more points.

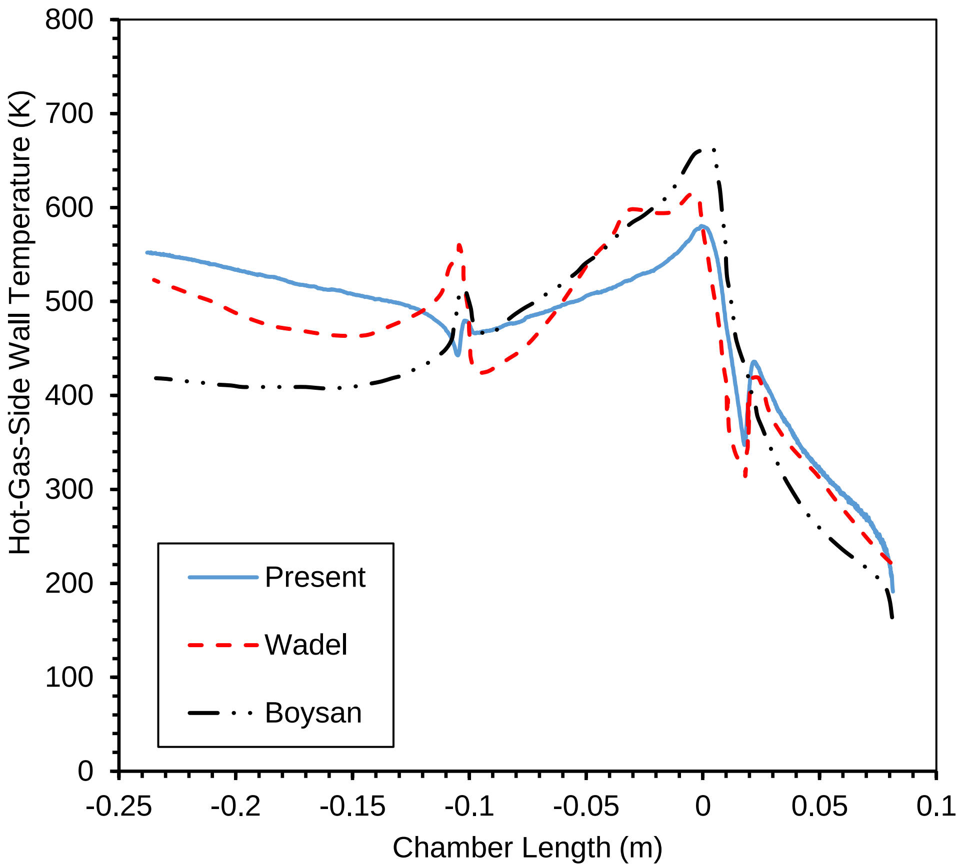



Figure 6: Comparison of hot-gas-side wall temperature for different CFD computations

Comparison of hot-gas-side wall temperature is shown in Figure 6 [Boysan, 2008; Wadel, 1996; Wadel, 1997]. The sudden change at the end of bifurcation zone can be seen in all three studies but the other change at the beginning of bifurcation zone can only be seen in Wadel and this study.

Wall temperature results for points between two cooling channels are displayed in Figure 7. When the deviation in the experimental study is considered, the results in this study are close to the experimental results.

The pressure distribution in the channel is shown in Figure 8. In contrast to Wadel's numerical study, the values of the coolant channel pressure are underpredicted in many regions.

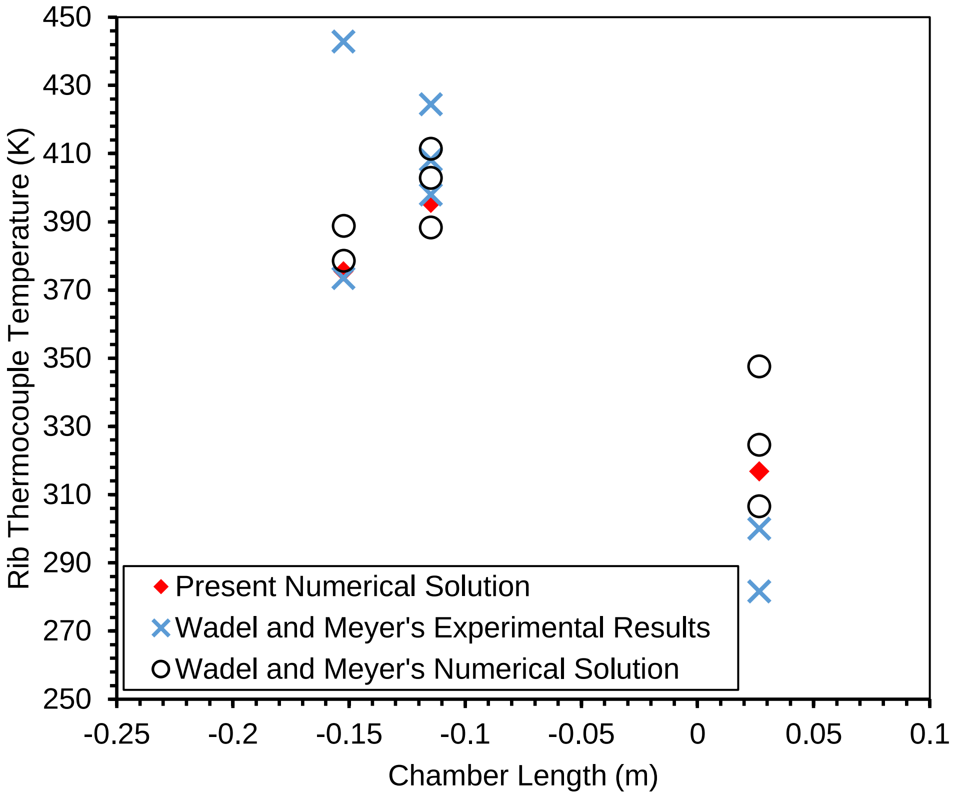



Figure 7: Rib thermocouple temperature

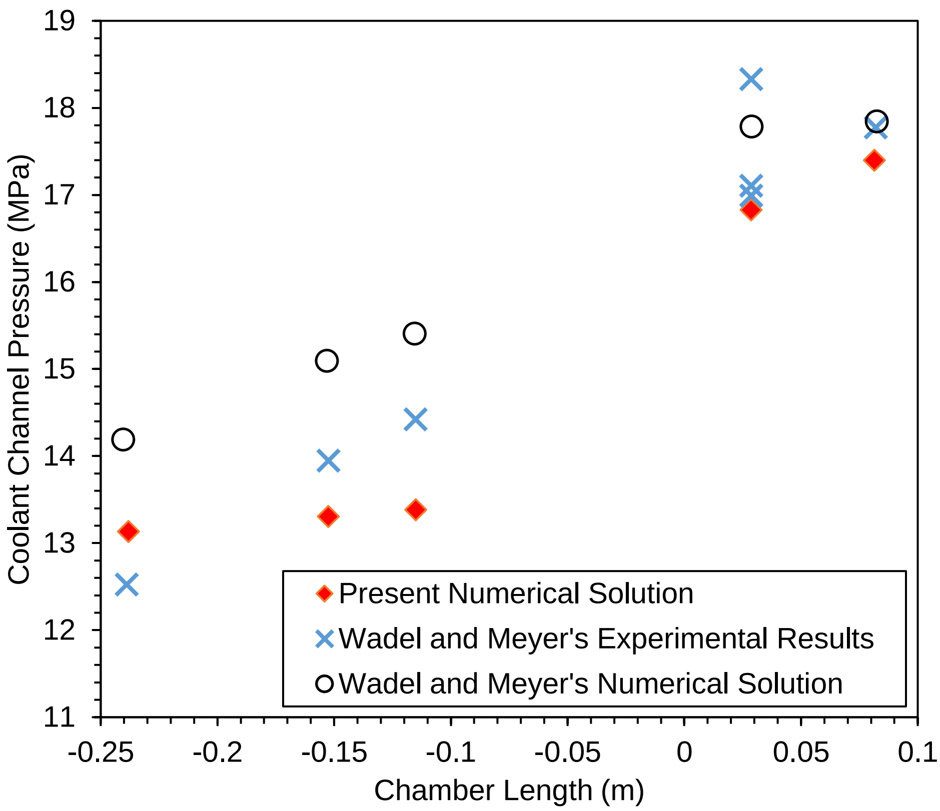



Figure 8: Coolant channel pressure

## RESULTS

After the validation study, 5 new geometries were created using the analysis features used. Reynolds number, total mass flow rate are kept constant. The aspect ratios (h/w ratio) of channels for all geometries are not changed, only the number of channels is changed. Bifurcation channel structure in validation are not used for this part. Other parameters are shown in Table 2.

Table 2: Parameters of new analyzes

| Number of Channels | Mass Flow Rate (kg/s) | Channel Height (mm) |
|---|---|---|
| 50 | 0.0230 | 2.477 |
| 75 | 0.0153 | 1.651 |
| 100 | 0.0115 | 1.238 |
| 125 | 0.0092 | 0.991 |
| 150 | 0.0077 | 0.825 |

As shown in Figure 9, the highest wall temperature is for 50-channel condition and as the number of channels increases, temperatures decrease.

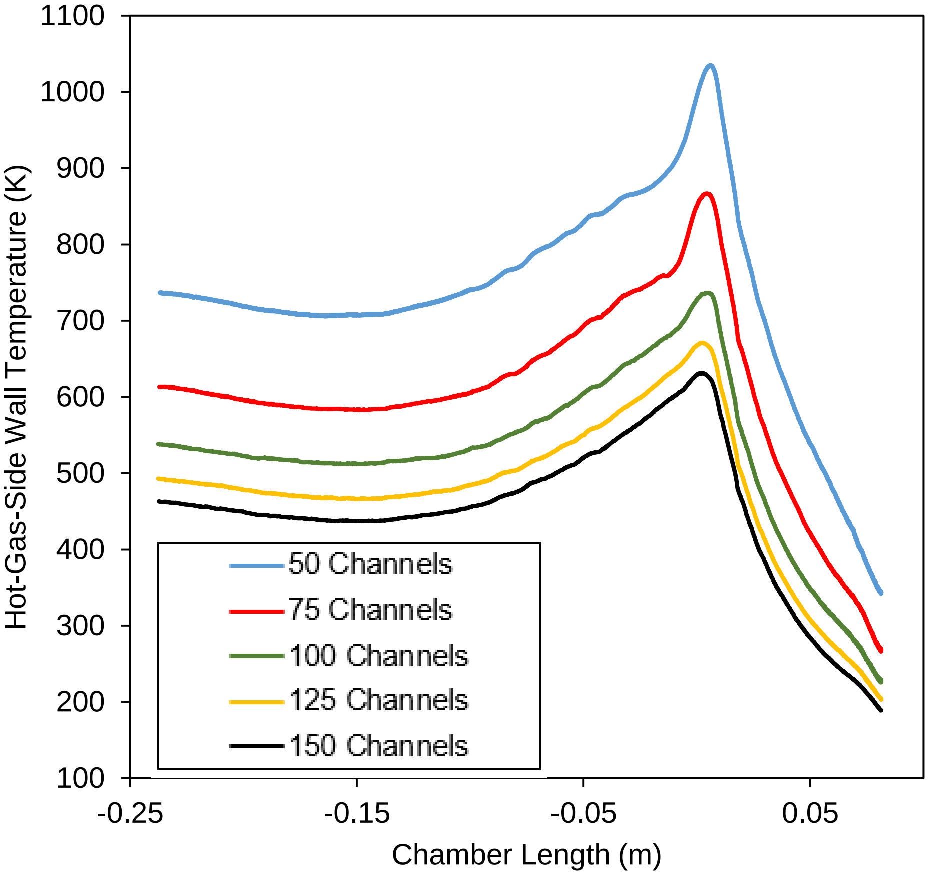



Figure 9: Hot-gas-side wall temperature

For lower solid temperatures the number of channels should be increased however, as can be seen from the figure 10(b), increasing the number of channels also has the disadvantage. Because the pressure drop increases exponentially with increasing number of channels.

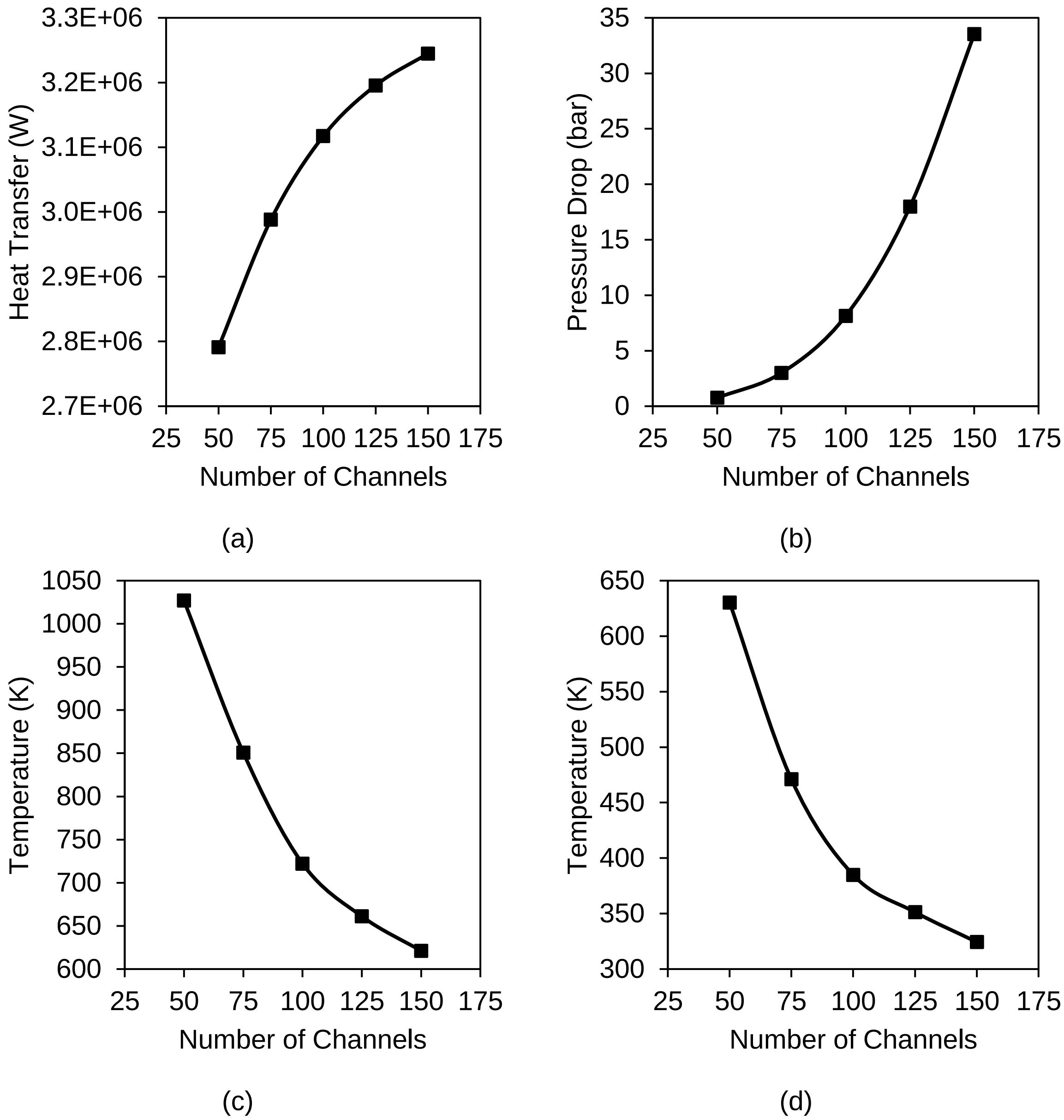


Figure 10: (a) Amount of heat passing through the wall, (b) pressure drop in the channel, (c) maximum temperature in the solid and (d) maximum temperature in the fluid

In figure 10 (a), amount of heat passing through the wall increases as the number of channels increases. As a result, the maximum temperatures in the solid and fluid are reduced by the number of channels.

## DISCUSSION AND CONCLUSION

In this study, the effect of channel number on regenerative cooling effectiveness is studied. As a result, as the number of channels increases, temperature decrease and heat transfer passing through the wall and pressure drop in the channels increase.

In the future, the effect of different channel cross section geometry and different proportions will be investigated on regenerative cooling effectiveness.

# References

Bartz, D.R. (1965) Turbulent Boundary-Layer Heat Transfer from Rapidly Accelerating Flow of Rocket Combustion Gases and of Heated Air, Jet Propulsion Laboratory, California Institute of Technology Pasadena.

Boysan, M.E. (2008) Analysis of Regenerative Cooling in Liquid Propellant Rocket Engines (Master's thesis, Middle East Technical University), December 2008.

Ulas, A. and Boysan, E. (2011) Numerical Analysis of Regenerative Cooling in Liquid Propellant Rocket Engines, Elsevier, November 2011.

Wadel, M.F. and Meyer, M.L. (1996) Validation of High Aspect Ratio Cooling in a 89 kN (20,000 $lb_f$) Thrust Combustion Chamber, AIAA.

Wadel, M.F. (1997) Comparison of High Aspect Ratio Cooling Channel Designs for a Rocket Combustion Chamber, AIAA.

Wadel, M.F. (1998) Comparison of High Aspect Ratio Cooling Channel Designs for a Rocket Combustion Chamber With Development of an Optimized Design, NASA, January 1998.