Heat Exchange in Liquid Fuel Rocket Engine

Jakub Malczenko

June 21, 2018

1. Introduction

In this project, the problem of heat exchange in combustion chamber's wall will be considered. In order to predict the temperature distribution, a special script in MATLAB software has been written. The final version of a program let us modify various input parameters, for example: combustion chamber's and insulation dimensions, parameters of cooling system and mass flow of oxidizer and fuel. The following project will present calculations made for engine designed in MELprop Student's Propulsion Association.

2. Mathematical model

Two different types of boundary conditions has been used. The calculations of temperature distribution has been made using model below:

$$T_{i,k+1} = \frac{1}{M}[T_{i-1,k} + (M-2)T_{i,k} + T_{i+1,k}] \tag{1}$$

where:

$$M = \frac{1}{Fo} \quad Fo - Fourier \quad number$$

For the first node the third type of boundary condition has been used. Knowing the temperature of gases inside the combustion chamber and convective coefficient it is possible to use the following equation to compute heat exchange:

$$T_{s,k+1} = \frac{1}{M}[2CT_{p,k} + [M - 2(C+1)]T_{s,k} + 2T_{1,k}] \tag{2}$$

where:

$$C = \frac{\alpha \Delta x}{\lambda} \quad \lambda - conductivity \quad \alpha - convective \quad coefficient$$

Convective coefficient has been calculated using equation below:

$$h_m = 0.024\frac{C_p G^{0.8}}{D_i^{0.2}}[1 + (\frac{D_i}{L})^{0.7}] \tag{3}$$

C_p - specific heat of combustion mixture G - mass velocity

L - chamber length D_i - chamber inside diameter

The second type of boundary condition has been used on a border between insulation and combustion chamber wall. To ensure the highest possible safety factor, the ideal connection model has been chosen (fourth type boundary condition):

$$T_s = \frac{1}{p+1}(T'_i + pT''_{i+1}) \tag{4}$$

where:

$$p = \frac{\lambda'' \Delta x'}{\lambda' \Delta x''}$$

Beside this equations, special functions interpolating values of specific heat and conductivity have been implemented. These two parameters vary according to temperature, so it is necessary to change these values during calculations.

3. Input data

Chamber wall thickness	3mm
Insulation thickness	1mm
Combustion chamber inner diameter	63mm
Combustion chamber length	240mm
Material	ALuminium 2024
N_2O mass flow	0.01345 kg/s
Kerosene mass flow	0.00236 kg/s
Initial temperature	300K
Temperature in combustion chamber	3000K

Additional calculations for engine with water colling system have been made. This installation had parameters presented below:

Water temperature	300K
Water flow speed	0.1 m/s
Equivalent radius of cooling channel	15mm

4. Results

On every graph there are ten lines showing temperature with 3 seconds interval. The highest line shows temperature distribution after 30 seconds of work.

Figure 1: Without insulation and colling system

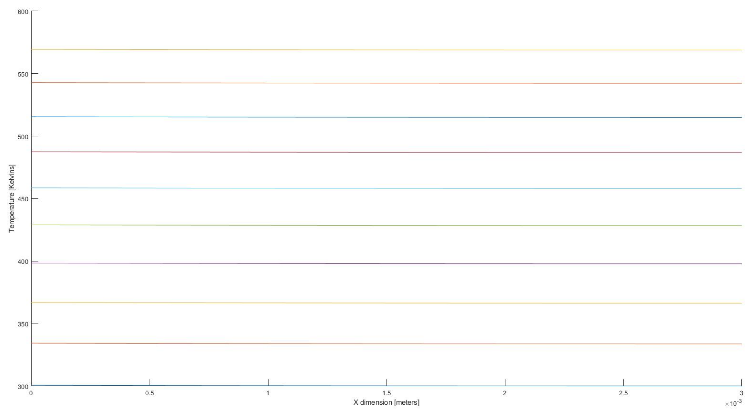


Figure 2: With insulation but without colling system

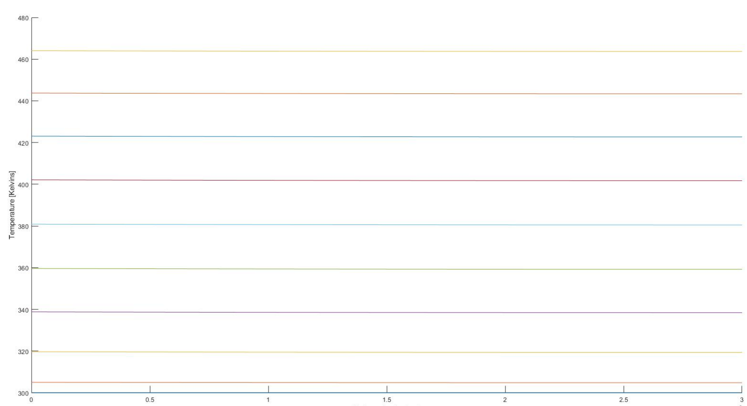


Figure 3: Without insulation but without colling system

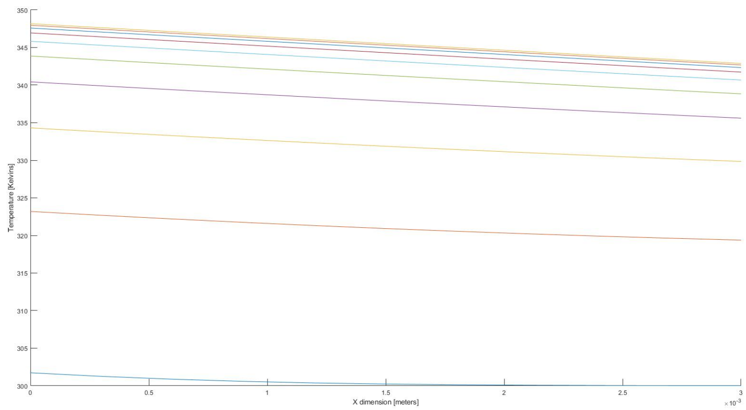
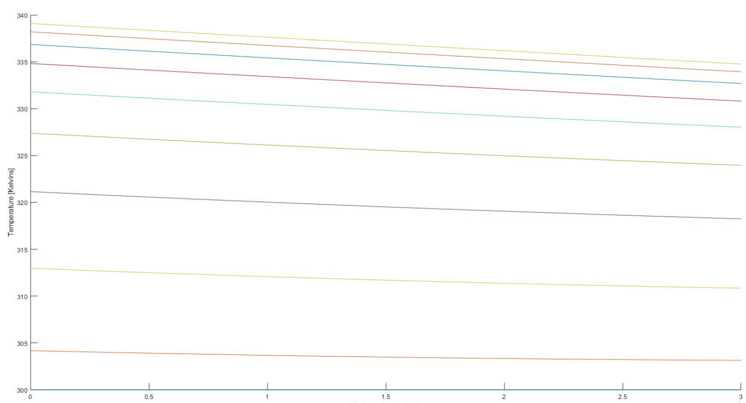


Figure 4: With insulation and colling system



Beside all this graphs, program generates also a csv file, which contains data about temperature in every node and every time step.

5. Summary

All results seem to show to low final temperature. To low convective coefficient is probably the reason of this fact. Unfortunately there are not simply methods, which let us to calculate real value of this factor. The solution of this problem is to take the value of convective coefficient directly from literature and make all the calculations again. Besides problem connected with convection, there are no other unrealistic results. The temperature distribution looks correct. Also differences between results with and without insulation are physically acceptable.

References

[1] F.P.Incropera/D.deWitt, "Fundamentals of Heat Transfer"

[2] www.matweb.com

[3] K. Pietrzak, "Computing of the combustion parameters for a liquid propellant rocket engine"

[4] S. Wiśniewski, T.S. Wiśniewski, "Wymiana Ciepła"