Cycling Loading Effect on a Solid Propellant Engine Performances

Adrian ARGHIROPOL*
Dr. Mircea BOSCOIANU **

Flow inside a solid rocket engine has been studied both experimentally and numerically in the Bucharest's Military Technical Academy. The article will discuss previous experimental and computational research on the two dimensions and axis symmetric CFD modeling of the flow inside a solid propellant engine with a specific axial distribution of the propellant's material temperature. The study will indicate how the rocket engine's overall performances like thrust, and internal flow parameters like velocity, pressure and temperature will change because of the variable axial temperature distribution in the solid propellant assumed to be a vascoelastic material under cycling loading.

The solid propellant used as the fuel for the rocket engines is assumed as viscoelastic material. All the viscoelastic materials are dissipative in nature. They will dissipate always large amounts of mechanical energy in the form of heat when subject to high frequency loading. Considering the high-speed of the rocket, or their own carrier – i.e. supersonic aircraft - the solid fuel was proved to be a subject for cycle loading due to high speed vibrations. Heating due to vibration near a resonance frequency may lead to melting, or material failure, or change in the rocket engine performances. *Tormey and Britton*[1] conducted various vibration tests on various solid fuel families and revealed that the heating of the solid propellant due to vibrations increased the material temperature significantly. The scope of this article will be to issue a short overview of the inside flow of a solid rocket motor considering the axial variation of the solid propellant material temperature versus the constant temperature case assumption during the R&D practice.

The axial distribution of the temperature inside the solid fuel is a PhD thesis subject of one of the authors and results based on assumptions will be described briefly in order to reveal the huge advantage of using the CFD simulations versus various costly rocket test bed programs during R&D period.

To simulate the solid rocket propellant, a viscoelastic rod insulated on its lateral surface, as can be seen in Fig.1, is considered like the mechanical equivalent model of the rocket engine. One end stays free, while the other end is attached to a vibrator. The vibrator will have a prescribed stress given by $\sigma = \sigma_0 \cos \omega t$ with σ_0 the stress amplitude, ω the frequency and t the time. The initial temperature T_0 of the vibrator is assumed constant. The convective boundary condition is assumed at $x =$

0 , while ***H*** is the surface conductance and ***K*** is the thermal conductivity of the material.

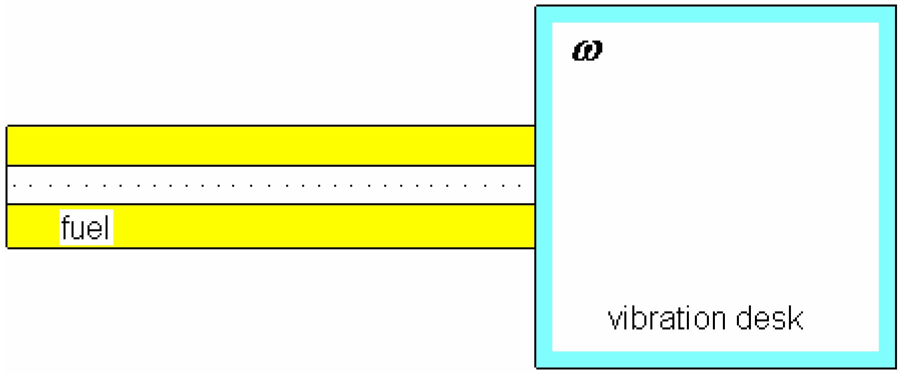


Figure 1. – mechanical equivlent model

$x = 0$ $\quad\quad$ $x = l$

$$\frac{\delta T}{\delta x} = H\frac{(T - T_0)}{K} \qquad T = T_o$$

$$\sigma = 0 \qquad \sigma = \sigma_o \cos\ \omega t$$

The objective of this initial study was to find the temperature distribution along the viscoelastic rod and that was done using the governing equations: the energy balance equation, the equation of motion, the stress-strain relationship for induced vibrations from 1.0 kHz up to 100 kHz from ***x=0*** to ***x = l.***

The governing equations are next:

$$\frac{d^2\sigma_1}{dx^2} + \omega^2\rho(J_1\sigma_1 + J_2\sigma_2) = 0 \qquad (1)$$

$$\frac{d^2\sigma_2}{dx^2} + \omega^2\rho(J_1\sigma_2 - J_2\sigma_1) = 0 \qquad (2)$$

$$\rho C(\frac{\partial T}{\partial t}) = \frac{\partial}{\partial x}(K\frac{\partial T}{\partial x}) + \frac{\omega}{2}J_2(\sigma_1^2 + \sigma_2^2) \qquad (3)$$

Such study was done also in the past by Mehdi Pourrazady and Harish Krishnamurty[2] in their wide research about boundary conditions and implications of the variable thermal conductivity over the temperature of the solid rocket propellants. They demonstrate how the slightest change in the stress amplitude could cause large variations in the dissipation of the heat, and like a consequence a significant change of the temperature along the rod. From the generalized curve of the temperature dependence of thermal conductivity of the polymers[3]

the following equation for the thermal conductivity of the solid rocket propellant will be in a linear form, as follows:

$$K = C_1 - C_2 T \qquad (4)$$

and that will change the form of the energy equation, as next:

$$C_1\frac{\partial^2 T}{\partial x^2} - C_2\frac{\partial}{\partial x}(T\frac{\partial T}{\partial x}) + \frac{\omega}{2}J_1(\sigma_1^2 + \sigma_2^2) \ (5)$$

where ρ is the mass density, ***C*** the specific heat, ***J1*** the storage modules, ***J2*** the loss modules, and $\boldsymbol{\sigma 1}$ and $\boldsymbol{\sigma 2}$ the real and imaginary parts of stress amplitude $\boldsymbol{\sigma}$, respectively.

For our particular solid rocket engine project, the results of the study revelead on a wide frequency variation from 1.0kHz up to 100kHz and from $\boldsymbol{x=0}$ to $\boldsymbol{x = l}$ the next possible distribution of worst case scenario axial temperatures developed from an initial ambiental temperature of 38 grd.C:

Table 1. – solid rocket propellant's temperature variation

Section	x/l	T [grd.F]	T [grd.C]	T [grd.K]
1	0.00	100	38	311
2	0.05	150	66	339
3	0.10	250	121	394
4	0.15	350	177	450
5	0.20	400	204	477
6	0.25	500	260	533
7	0.30	350	177	450
8	0.35	250	121	394
9	0.40	200	93	366
10	0.45	150	66	339
11	0.50	100	38	311
Section	x/l	T [grd.F]	T [grd.C]	T [grd.K]
12	0.55	150	66	339
13	0.60	250	121	394
14	0.65	350	177	450
15	0.70	400	204	477
16	0.75	500	260	533
17	0.80	350	177	450
18	0.85	250	121	394
19	0.90	200	93	366
20	0.95	150	66	339
21	1.00	100	38	311

The propellant lenght was divided in 21 initial cells and finally raffined in 40 cells. The assumption that the temperature will be constant by radius and variable by axial length will be used in the next CFD simulations using FLUENT[R] software package.

First study was done with the assumption that the fuel temperature T = 300 ^oK = constant and like a consequence the fuel's burning temperature assumed constant at 2,891 ^oK .

- ideal gas with c_p = 1006 and molecular weight of 28
- section - minimal or radius = 72 (mm)
- 2d & dp axial simetric
- p = 110 [atm] – work pressure
- $T_{\text{fuel burning}}$ = 2891 ^oK
- T_{ambient} = 300 ^oK = constant

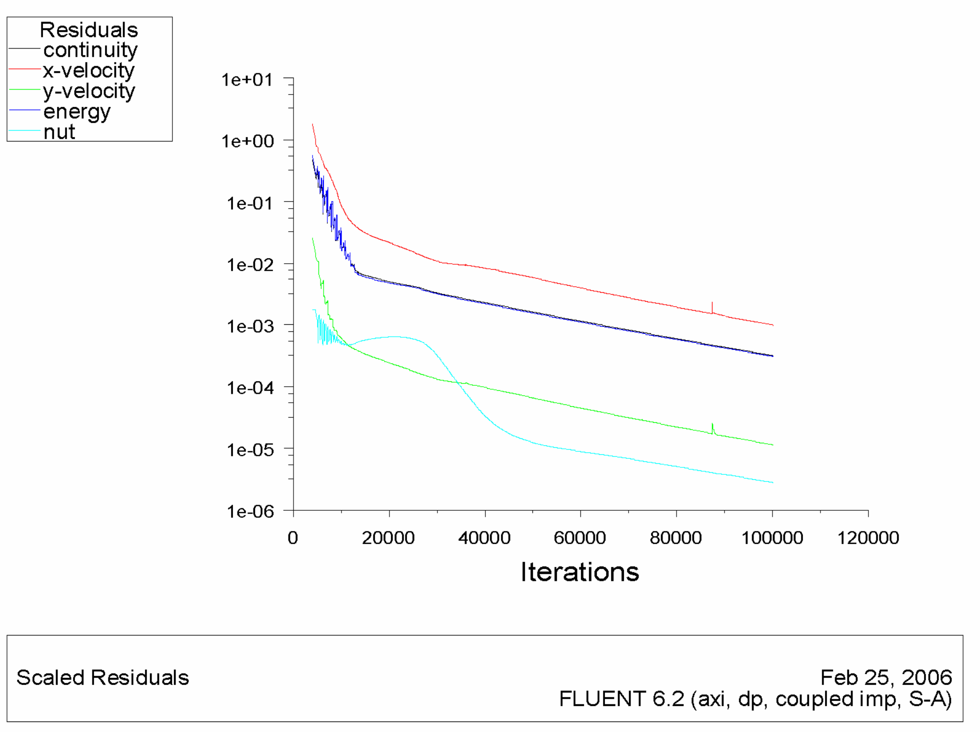
- T_{outlet} = 1460 ^oK
- m = 214kg – propellant mass
- Traction impulse = 51000 [daN x s] – analitical determination
- fuel burning time = 5 [s]
- F_{av} = 10,000 [daN] - average Force of the rocket engine
- p max = 140 [barr] – maximal pressure
- p med = 110 [barr] – average pressure

Results of first CFD simulation under above conditions:
Version: axi, dp, coupled imp, S-A (axi, double precision, coupled implicit, Spalart-Allmaras)
reversed flow in 376 faces on pressure-inlet 7.
turbulent viscosity limited to viscosity ratio of 1.000000e+005 in 2 cells
!179362 solution is converged
179362 1.5177e-05 9.9997e-04 7.9858e-07 1.4388e-05 7.4263e-07
3:35:18 82260

<u>Second study</u> was done with the assumption that the fuel temperature T = var was listed in the Table1. and rafined in 40 equal lenght cells. The starting conditions as next:

- ideal gas with c_p = 1006 and molecular weight of 28
- section - minimal or radius = 72 (mm)
- 2d & dp axial simetric
- p = 110 [atm] – work pressure
- $T_{fuel\ burning}$ = 2891 ^oK
- $T_{ambient}$ = 300 ^oK
- T_{outlet} = 1460 ^oK
- m = 214kg – propellant mass
- Traction impulse = 51000 [daN x s] – analitical determination
- fuel burning time = 5 [s]
- F_{av} = 10,000 [daN] - average Force of the rocket engine
- p max = 140 [barr] – maximal pressure
- p med = 110 [barr] – average pressure

Results of second CFD simulation under above conditions:
scalled Residuals and Drag Convergence curves were almost the same shape in both cases, with only the convergence time differences (30% in number of convergence steps):



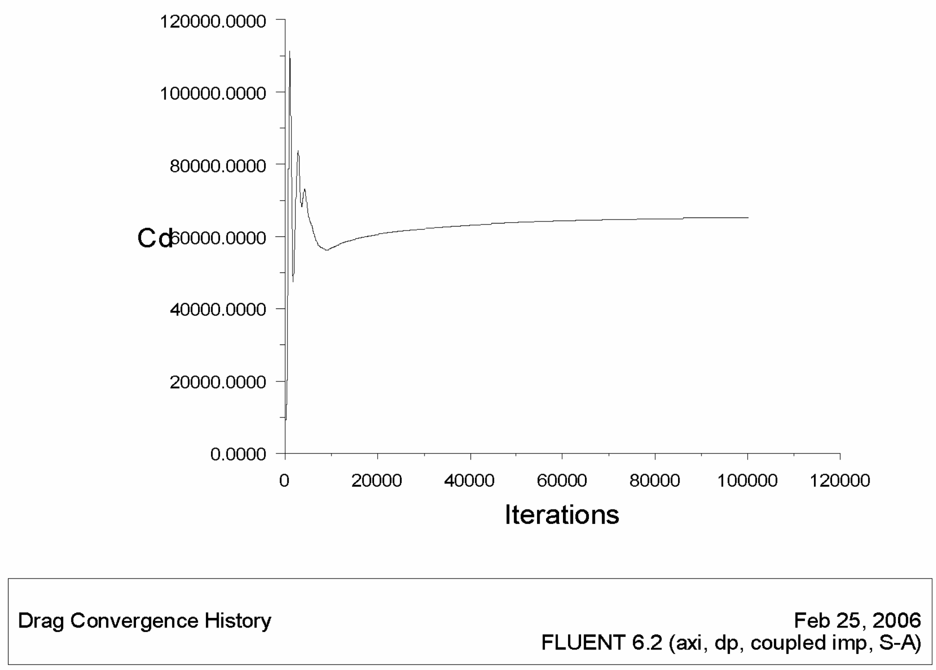


Figure 3. – Scalled Residuals and Drag Convergence History

Picture 6. and Picture 7. will show the presure distribution. The distribution it was almost the same because of the initial conditions. The velocity distribution will have the same shape in both cases as can be seen in Pictures 8. and Picture 9.

The temperature distribution of the flow inside the engine will face an linear increase with the fuel's burning temperature because of fuel heating due to cyclic loading. Thus, fuel's heating due to vibration near a resonance frequency may lead to melting, or material failure, or change in the rocket engine performances. Table 1. distribution of the fuel's temperature are a good example of further thermal stress in this engine particular example. However, the thermal stress will affect also parts of the rocket engine, such as the convergent parts the nozzle. One can expect an increase of thrust due to burning temperature increase in the second simulation. However, both simulations will show a very bad turbulence and there is a real need to redesign the nozzle. See Figure 10. and Figure 11.

Conclusions:

1 – CFD simulation will further cut the high costs of the real tests and will further help R&D department to generate a proper engineering and production design

2 – the engine under simulation is subject of explosion or severe malfunction in case of material failure and authors will strongly recommend a special study on this issue

3 – the fuel melting is not real possible as far in the real world, during a flight, the worst case scenario cycle loading will generate an equivalent amount of heat, but the ambient temperature will be close to – 60 deg. C and will reduce the melting possibility because of fuel temperature lower value with $100^{o}K$.

4 – the nozzle should be redesigned to reduce as much is possible the turbulence in the convergent area.

Bibliography:

1 – Tormey, J.F. and Britton, S.C., "Effect of Cyclic Loading on Solid Propellant Grain Structure", AIAA Journal, Vol.1, 1963, pp.1763-1770

2 – Pourrazady, M. and Harish Krishnamurty, "Thermal Response of A Dynamically Loaded Viscoelastic Rod with Variable Proprieties", University of Toledo, Toledo – Ohio, USA

3 - FLUENT^{R} help files and related documentation. FLUENT^{R} is a trade mark of ANSYS Company.

*** Ph.D student at Bucharest's Technical Military Academy, phone: 0040723653987, e-mail: arghiropol@clicknet.ro**

**** Professor at Bucharest's Technical Military Academy;**

ADENDUM: CFD simulation results

The distribution of the temperature inside the engine is shown in the next pictures:

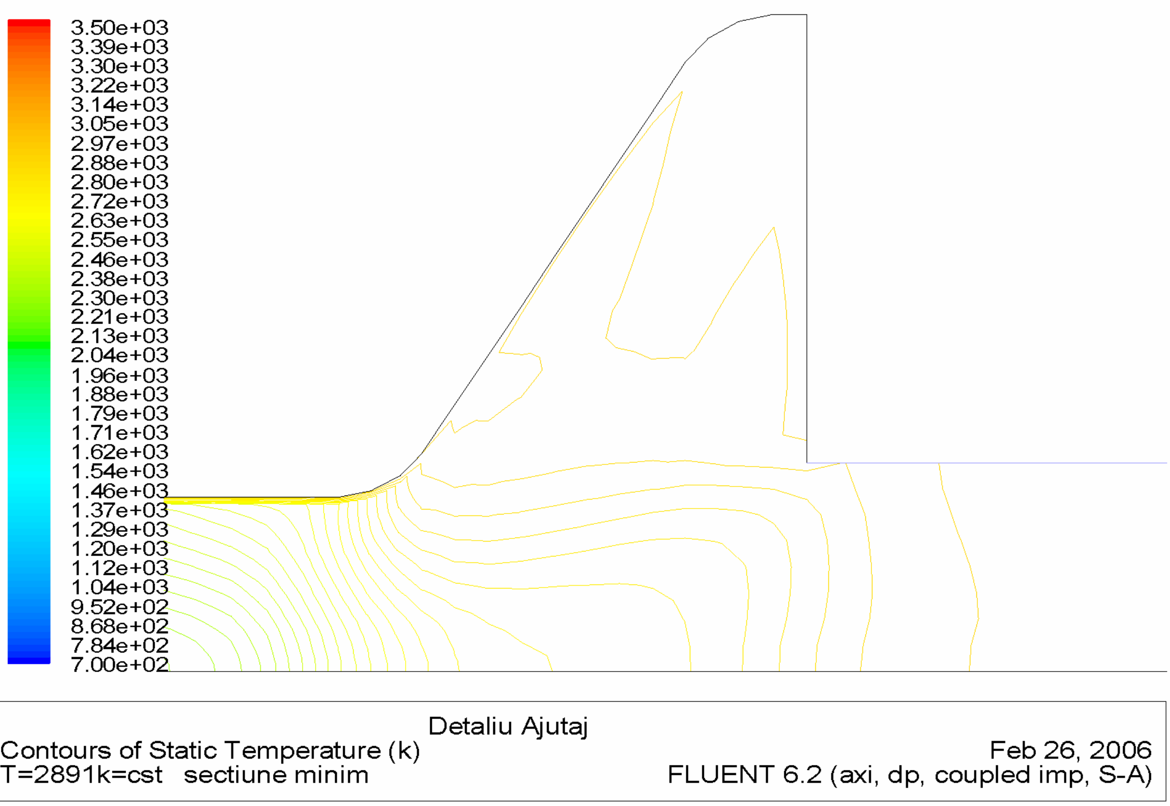


Figure 4. – peak temperature 2,930 ^{o}K

And consequently the simulation for T = variable will bring next results:

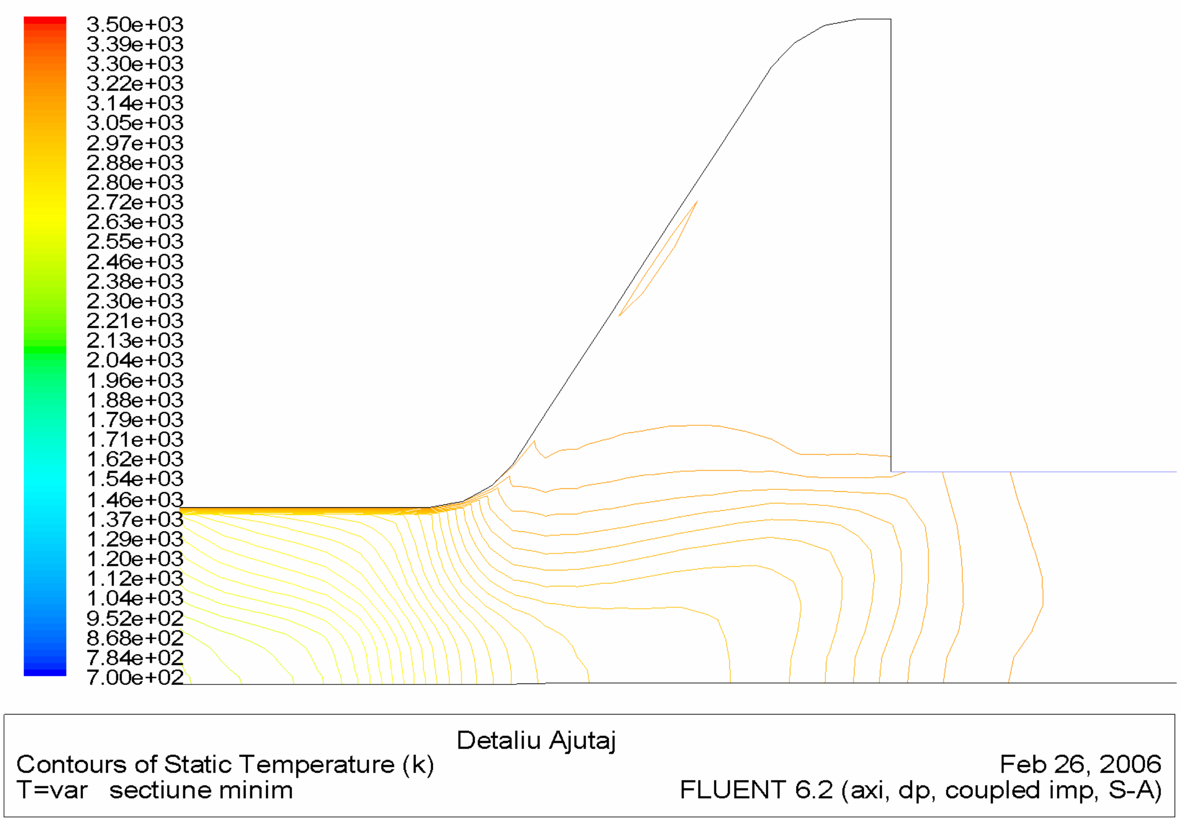


Figure 5. – peak temperature 3,430 ^{o}K

The pressure distribution in the nozzle area is almost the same in both cases, as it was supposed to be found, as can be seen in the Figure 6. and Figure 7.

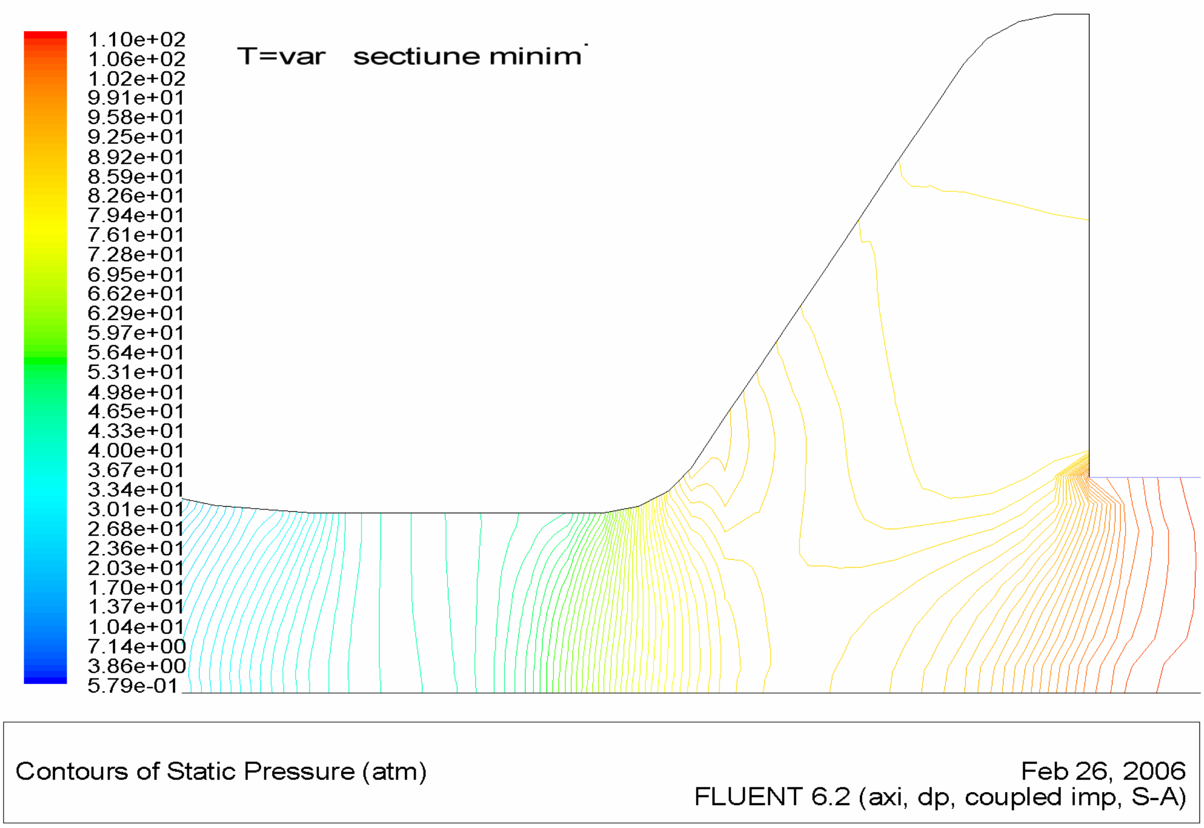


Figure 6.

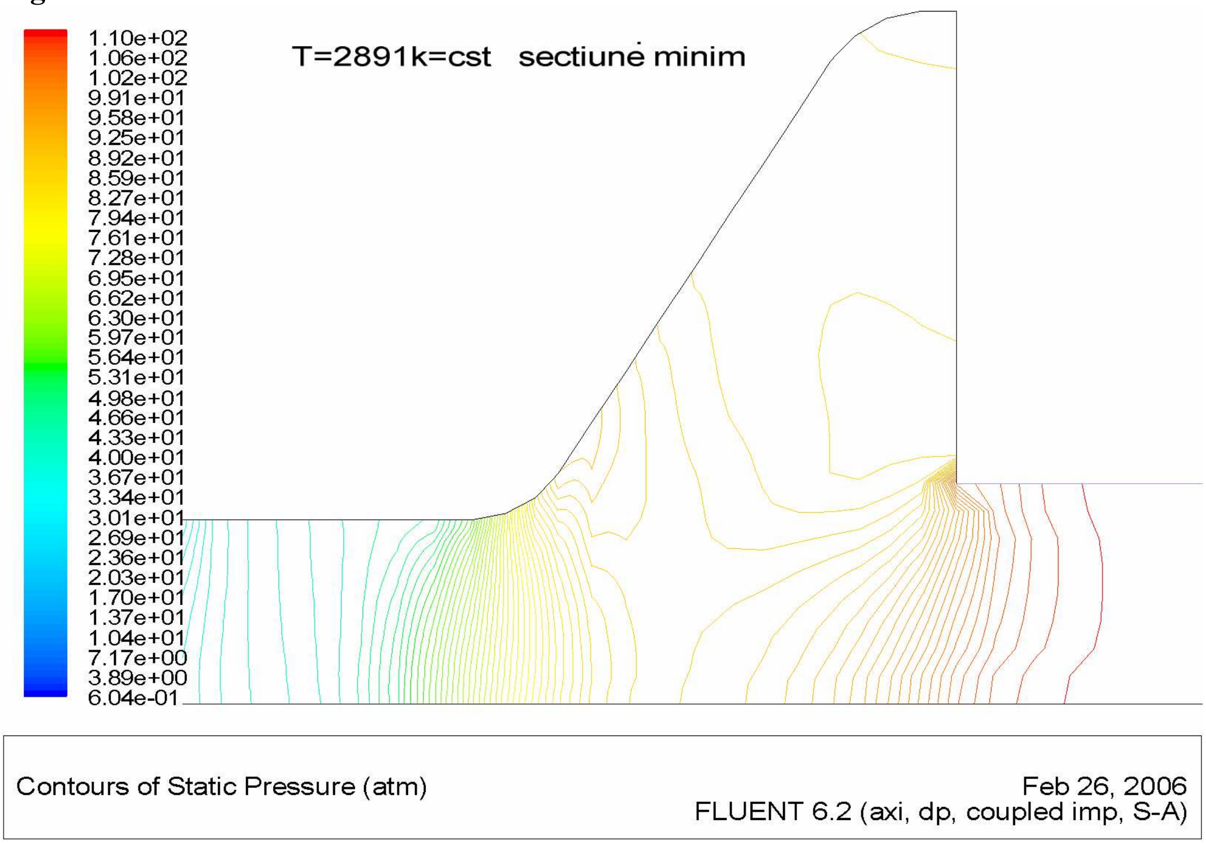


Figure 7.

The velocity distribution in the nozzle area is almost the same in both cases, with very little differences – 6.6%, no matter 18% rise in burning temperature, as can be seen in the Figure 8. and Figure 9.

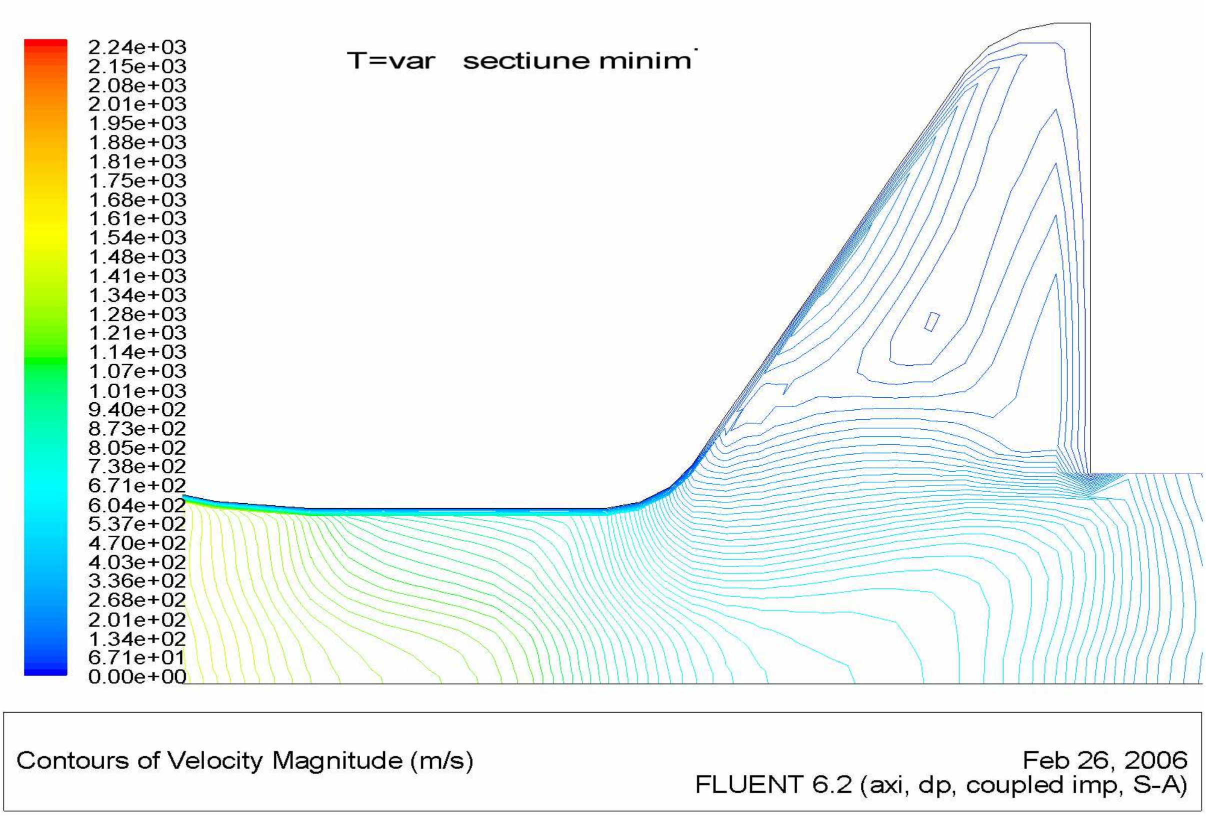


Figure 8.

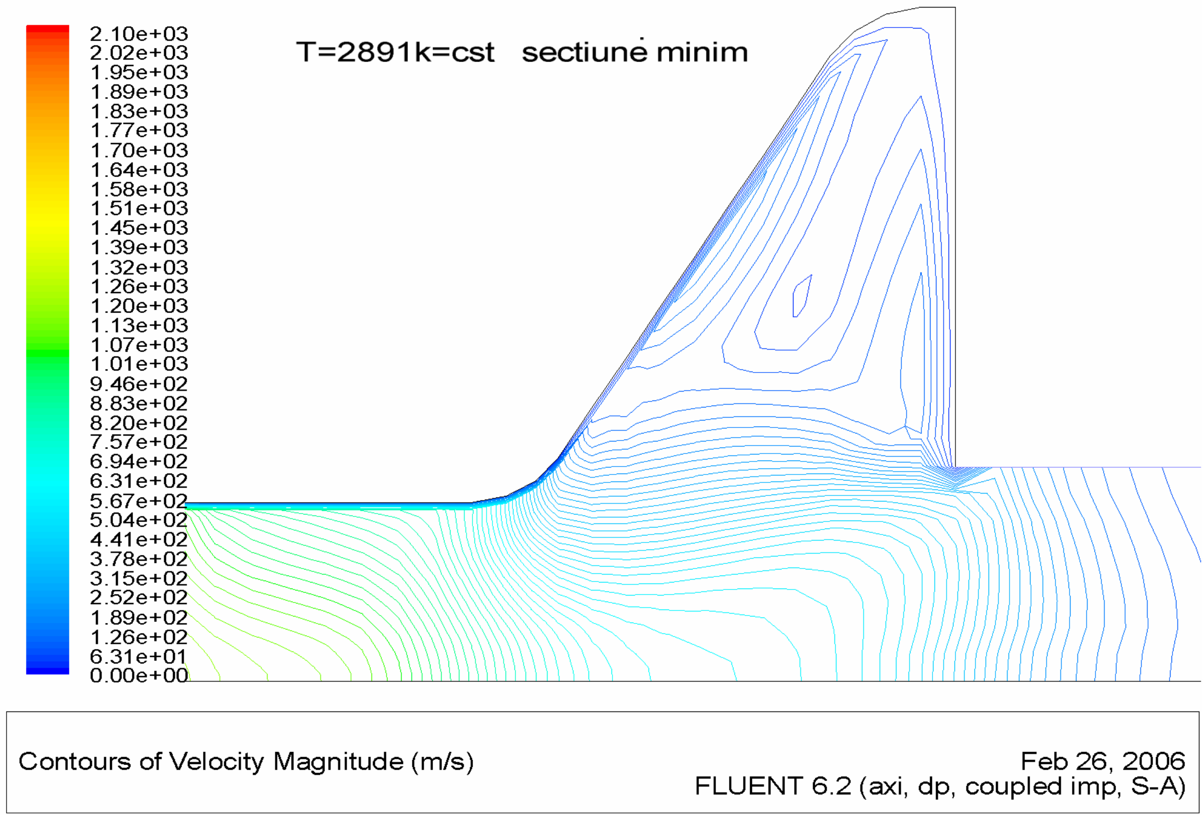


Figure 9.

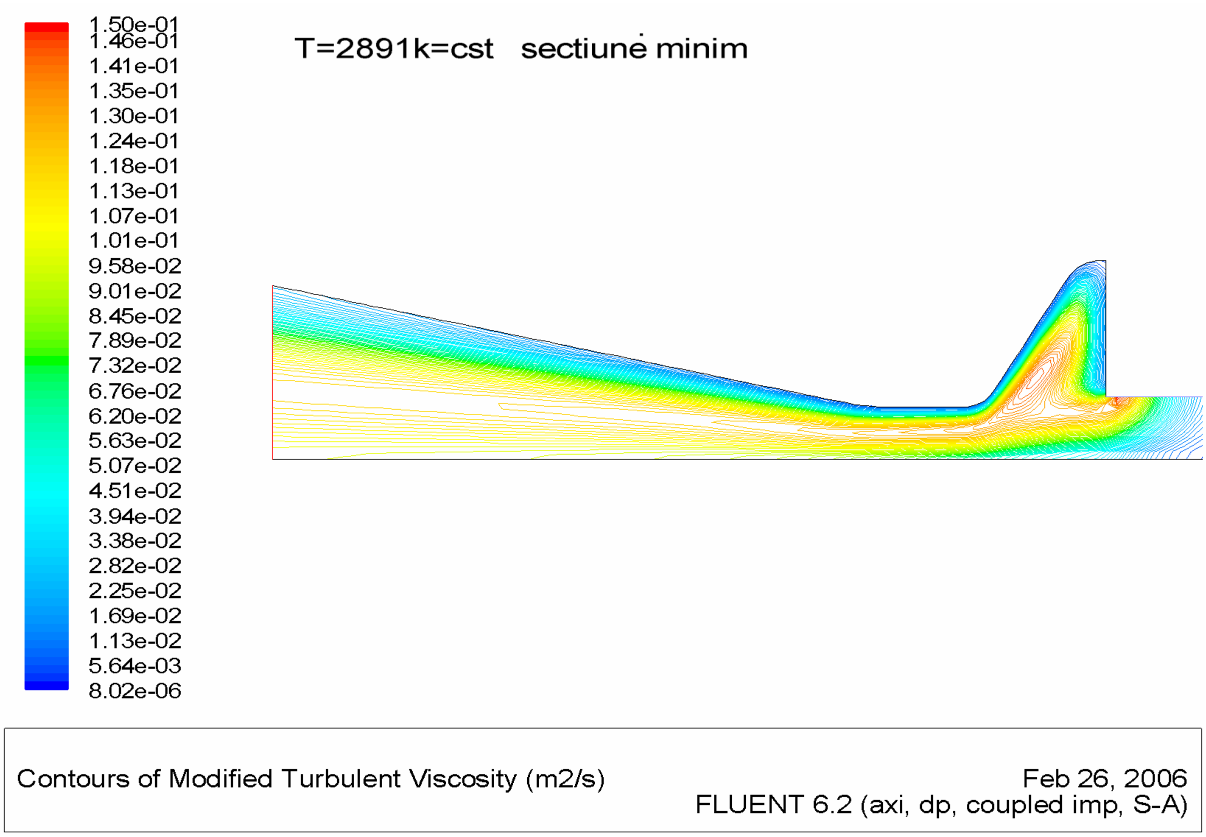


Figure 10.

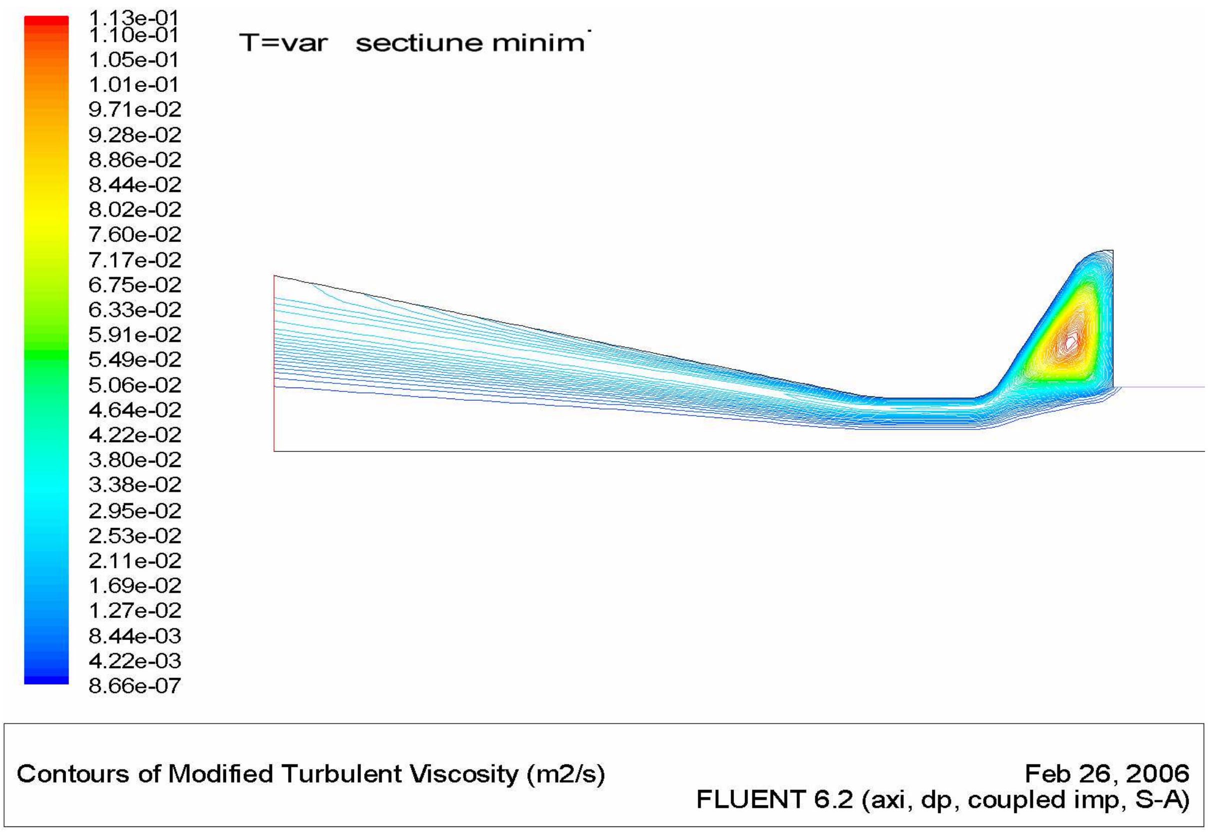


Figure 11.