CFD Check-up Study of a Solid Propellant Rocket Engine Internal Pressure and Performance Under Random Nozzle Diameter Change

ADRIAN COMAN, ADRIAN ARGHIROPOL, MIRCEA BOSCOIANU
Aircraft Department
Military Technical Academy
81-83, G. Cosbuc street, 5 sector, Bucharest
ROMANIA

***Abstract**: Flow inside a solid rocket engine has been a topic of research and 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-dimensional and axis-symmetrical solid propellant rocket engine inside flow parameters, because of nozzle's critical diameter random decrease soon after initialization of the engine. Such study, based on cost effective CFD simulations, will indicate how pressure in the engine's burning chamber can change, and further help to evaluate some of the implications on reliability and correct functionality of a real product.*

***Key-Words**: Rocket engine, Flow parameters, Critical diameter, CFD simulations, Burning chamber*

1. Introduction

Following a practical case study in Bucharest's Military Technical Academy, the authors of the research tried to evaluate in a cost effective and pragmatic manner the worst case scenario functionality of a real rocket engine with nozzle's critical diameter random decrease for unknown reasons.

The solid propellant rocket engine (see Fig. 1), or the Device Under Test (DUT) was fired on the specific bed test platform and faced few reliability and functionality issues after a time from manufacturing. Potential issues like: vibrations, shocks, nozzle's critical diameter random change because of inside broken mechanical particles, bad installation procedures during manufacturing, warehouse temperature and humidity range implications are together and/or separate enough to generate such reliability issues. Because few of the engines blew up in the first 0.2 s after ignition, the above research task will have a very important final matter, in order to help evaluating one of the main possible reasons of the event, i.e. the decrease of the nozzles critical diameter and rapid rise of the inside pressure.

Fig 1. – Solid Propellant Rocket Engine - DUT

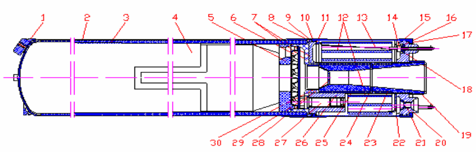

Fig.1

Based on specific bed tests and other various experiments, it was acknowledged by the group of authors, that one of the most important parameter in order to significantly decrease the reliability of the DUT was the inside pressure range variations in the first 0.25 seconds after ignition of the engine, like a function of nozzle's random critical section diameter possible variation.

2. CFD case study and initial data available

To accomplish the task, the two-dimensional and axis-symmetric CFD was employed, using the FLUENT[R] software package. Results of the study will show first the (***F***) thrust and (***p***) pressure maximal values possible and second how the velocity, turbulence and temperature of the rocket engine's internal flow will change, in the worst case scenarios of possible critical diameter section change, up to the level to mechanically damage of the rocket engine. The first case of the below study will be the validation of the results of CFD modeling solution, versus experimental data available.

Case 1 - nominal critical diameter **D_n=12.5 mm** within next time moments:

- **t=0.0 s**
- **t=0.2 s**
- **t=6.0 s**

Above time moments are explicit on the DUT's bed tests pressure (***p***) and thrust (***F***) versus time already known diagrams. If matching results can be obtained with CFD simulations, i.e. internal pressure (***p***) and thrust (***F***) parameters at same moments of time, it means will validate the overall CFD output results also for all other similar modeling cases of the study in the above initial conditions. The authors will consider acceptable for this particular study an average error of maximum 5-7% between CFD output results and experimental tests made, taking in consideration also the known and approved dispersion of the thrust and internal pressure parameters of the DUT in advance.

Based on CFD output results of pressure (***p***) and thrust (***F***) for the **Case 1** versus experimental various tests, it was possible to confirm the Spalat-Allmaras, double precision, two-dimensional and axis-symmetric CFD modeling solution, within 5% error range with the experimental values and generate the necessary step forward to employ the **Case 2** and **Case 3** of the DUT's check-up research study.

As far the **Case 1** was the initial CFD study validation case, the input conditions for **Case 2** and **Case 3** were supposed to define two other different values of the worst case critical section diameter. The most probable values were taken from the statistical malfunction study, performed in advance by the research team. The variation of the inside rocket flow parameters study was further done for the next diameters:

- **Case 2** with critical diameter **D_1=10.0 mm** at **t = 0.0 s**
- **Case 3** with critical diameter **D_2= 6.0 mm** at **t = 0.0 s**

It was proved to be more than adequate to simulate the internal flow parameters after rocket engine ignition (**t = 0.0 s)**, neglecting the igniter pressure influence, based on fact that all the internal pressure values are close to a maximum, and decreasing the critical nozzle's diameter impact will be the most significant in the reliability case study, because of increasing inside pressure with all the mechanical implications may occur. There was also a computing time constrain, as far lowering the diameter was supposed to increase/refine the number of cells, with a significant rise of computing time and overall study costs.

<u>Initial conditions:</u>

- at **t = 0.0 s** there will be no igniter influence taken in consideration and all the solid rocket fuel surfaces are considered to start burning in the same time with the same speed as was designed. The burn in temperature of the fuel was taken from the manufacturer database, and will be considered constant and not higher than 3060 [^{o}K];
- worst case scenario for the ambient temperature: + 50 [^{o}C];
- all gases were simplified as ideal gas, based on previous experience of the authors with other similar cases;

<u>CFD input data, coupled-imposed Spalat-Almaras solver:</u>

- T_{amb} = + 50 [C^o]
- Combustion chamber temperature: 3060 [K^o]
- p_{max} = 160 [atm] – starting pressure for the numerical simulation;
- fuel weight: m = 3.9 [kg]
- fuel density = 1.72 [kg/dm^3]
- fuel burning speed u = 18.0 [mm/s] – (accordingly with DUT's solid propellant manufacturer BOFORS)
- mass flow ≈ 1 [Kg/s]
- combustion chamber blow up pressure: p_{sp} = 230 [barr]
  (taken from DUT specs)
- hydrostatic maximal pressure value of the chamber: p_{inc} = 195 + 5 [barr] (taken from DUT specs)

3.Case 1–CFDoutput results for *p* and *F*

Case 1 – nominal critical diameter **D_n=12.5 mm** at time moment:

- **t=0.0 s** CFD simulation will calculate aprox. 151 [atm] inside static pressure ***p*** and a thrust force ***F*** of aprox. 2752 [N] (see also Fig.2 and Fig.3 in ADENDUM pictures area)

iter continuity x-velocity y-velocity energy nut
cd monitor-debit time/iter
42060 3.7452e-01 1.8220e-02 7.8468e+00 1.3297e-01 1.2684e-03 -4.4082e+03 -1.0e+00 134:04:30

Force vector: (1 0 0) pressure viscous total pressure viscous total
zone name force force force coefficient coefficient coefficient
[N] [N] [N]
------------------------- -------------- -------------- -------------- -------------- -------------- -------------------
wall (perete) -2796.3627 44.249155 -2752.1136 -4565.4902 72.243519 -4493.2466
------------------------- -------------- -------------- -------------- -------------- -------------- -------------------
net -2796.3627 44.249155 -2752.1136 -4565.4902 72.243519 -4493.2466

=====================================
==================

Maximal static pressure in the rocket engine's burning chamber: 151.03 [atm

Mesh model based on:
92388 quadrilateral cells, zone 2, binary.
17 2D mass-flow-inlet faces, zone 3, binary.
707 2D mass-flow-inlet faces, zone 4, binary.
86 2D mass-flow-inlet faces, zone 5, binary.
383 2D mass-flow-inlet faces, zone 6, binary.
119 2D pressure-outlet faces, zone 7, binary.
521 2D axis faces, zone 8, binary.
53 2D mass-flow-inlet faces, zone 9, binary.
50 2D mass-flow-inlet faces, zone 10, binary.
1936 2D wall faces, zone 11, binary.
182840 2D interior faces, zone 13, binary.
94325 nodes, binary.
94325 node flags, binary.

- **t=0.2 s** CFD simulation will calculate aprox. 147 [atm] inside static pressure ***p*** and a thrust force ***F*** of aprox. 2700 [N] (see also Fig.4 and Fig.5 in ADENDUM pictures area)

iter continuity x-velocity y-velocity energy nut cd monitor-debit time/iter
64010 4.3177e-02 6.7621e-05 1.0421e-01 3.8395e-02 2.0862e-05 -8.8531e+03 -1.000e+00 111:56:22

Force vector: (1 0 0) pressure viscous total pressure viscous total
zone name force force force coefficient coefficient coefficient
[N] [N] [N]
------------------------- -------------- -------------- -------------- -------------- -------------- -------------------
wall (perete) -2744.6652 44.825668 -2699.8395 -4481.086 73.184763 -4407.9012
------------------------- -------------- -------------- -------------- -------------- -------------- -------------------
net -2744.6652 44.825668 -2699.8395 -4481.086 73.184763 -4407.9012
===

Maximal static pressure in the rocket engine's burning chamber: 147.31 [atm]
===

Mesh model based on:
110465 quadrilateral cells, zone 2, binary.
52D mass-flow-inlet faces, zone 3, binary.
708 2D mass-flow-inlet faces, zone 4, binary.
86 2D mass-flow-inlet faces, zone 5, binary.
370 2D mass-flow-inlet faces, zone 6, binary.
126 2D pressure-outlet faces, zone 7, binary.
546 2D axis faces, zone 8, binary.
53 2D mass-flow-inlet faces, zone 9, binary.
50 2D mass-flow-inlet faces, zone 10, binary.
1936 2D wall faces, zone 11, binary.
218990 2D interior faces, zone 13, binary.
112406 nodes, binary.
112406 node flags, binary.

- **t=6.0 s** CFD simulations will calculate aprox. 34 [atm] inside static pressure ***p*** and a thrust force ***F*** of aprox. 1581 [N] (see also Fig.6 and Fig.7 in ADENDUM pictures area)

iter continuity x-velocity y-velocity energy nut cd monitor-debit time/iter
64010 4.3177e-02 6.7621e-05 1.0421e-01 3.8395e-02 2.0862e-05 -8.8531e+03 -1.000e+00 111:56:22

Force vector: (1 0 0) pressure viscous total pressure viscous total
zone name force force force coefficient coefficient coefficient
[N] [N] [N]
------------------------- -------------- -------------- -------------- -------------- -------------- -------------------
wall (perete) -1611.5589 30.242738 -1581.3161 -2631.1165 49.375898 -2581.7406
------------------------- -------------- -------------- -------------- -------------- -------------- -------------------
net -1611.5589 30.242738 -1581.3161 -2631.1165 49.375898 -2581.7406
===

Maximal static pressure in the rocket engine's burning chamber: 33.87 [atm]
==

Mesh model was based on:
94719 quadrilateral cells, zone 2, binary.
17 2D mass-flow-inlet faces, zone 3, binary.
707 2D mass-flow-inlet faces, zone 4, binary.
86 2D mass-flow-inlet faces, zone 5, binary.
383 2D mass-flow-inlet faces, zone 6, binary.
126 2D pressure-outlet faces, zone 7, binary.
520 2D axis faces, zone 8, binary.
53 2D mass-flow-inlet faces, zone 9, binary.
50 2D mass-flow-inlet faces, zone 10, binary.
1934 2D wall faces, zone 11, binary.
187500 2D interior faces, zone 13, binary.
96658 nodes, binary.
96658 nodes flags, binary.

Important observation: **t = 6.0 s** study was done at a different mass flow (0.325 kg/s) value, in order to fit with the mass flow range at the end of the engine's normal working time. There was also an increase of the diameter of the critical area of the nozzle, up to 12.65 mm after 6.0 s from ignition, just to be very accurate with the real situation based on experimental tests.

4. Case 2 – CFD output results for *p* and *F*

Case 2 – the modified critical diameter at **D_1=10.0 mm** and **t=0.0 s.** The static pressure inside the burning chamber was considered to have a rise pattern and increased probability to occur during the first moments after ignition in above diameter conditions. The first measurable effect on the bed test was the inside static pressure rapid rise within a negative effect on overall DUT reliability.

The CFD's mass flow value convergence was terminated by the authors as much as 0.98863 [kg/s] – or 1.2% error comparing with the starting mass flow input value, for specific computing time constrains (see Fig.8). In the above critical diameter conditions, the static pressure ***p*** in the burning chamber rise up to aprox. 184 [atm] and thrust force as much as aprox. 3868 [N], as follows (see also Fig.9 and Fig.10 in ADENDUM pictures area) :

```
  iter continuity   x-velocity   y-velocity     energy          nut
cd  monitor-debit  time/iter
141850 1.2563e-03 8.2116e-07 9.7020e-04 1.2528e-03 3.3694e-09 -
6.3151e+03 -9.8863e-01    68:19:57
Force vector: (1 0 0)     pressure          viscous             total
pressure       viscous            total
zone name                          force               force             force
coefficient   coefficient   coefficient
                                   [N]              [N]              [N]
------------------------ -------------- -------------- -------------- -------------- -
------------- -------------------
wall (perete)          -3917.3732  49.387777   -3867.9854 -6395.7114
80.633106       -6315.0783
------------------------ -------------- -------------- -------------- -------------- -
------------- -------------------
net                       -3917.3732  49.387777   -3867.9854 -6395.7114
80.633106       -6315.0783
====================================
==================
```

Maximal static pressure in the rocket engine's burning chamber: 183.8 [atm]

=====================================
===================

Mesh model was based on:

```
93751 quadrilateral cells, zone  2, binary.
    17 2D mass-flow-inlet faces, zone  3, binary.
   707 2D mass-flow-inlet faces, zone  4, binary.
    86 2D mass-flow-inlet faces, zone  5, binary.
   383 2D mass-flow-inlet faces, zone  6, binary.
   126 2D pressure-outlet faces, zone  7, binary.
   502 2D axis faces, zone  8, binary.
    53 2D mass-flow-inlet faces, zone  9, binary.
    50 2D mass-flow-inlet faces, zone 10, binary.
  2046 2D wall faces, zone 11, binary.
 185517 2D interior faces, zone 13, binary.
  95737 nodes, binary.
  95737 node flags, binary.
```

5. Case 3 – CFD output results for *p* and *F*

Case 3 – the modified critical diameter at **D_1=6.0 mm** and **t=0.0 s** . The first estimated effect can be the rise of the probability to have a blow up of the whole DUT, considering the static pressure results far away of the maximum admissible values of static pressure. Same as **Case 2**, the CFD's mass flow value convergence was terminated by the authors up to 0.95124 [kg/s] – with an 5.1% accepted error rate from the 1.0 kg/s imposed mass flow, because of the serious computer time constrains issues employed (see Fig.11).

In the above critical diameter conditions, the static pressure ***p*** in the burning chamber will rise up to aprox. 433 [atm] and thrust force ***F*** will decrease because of critical section lower diameter, as much as aprox. 2480 [N], as follows (see also Fig.12, Fig.13 in ADENDUM pictures area):

```
  iter continuity   x-velocity   y-velocity     energy          nut
cd  monitor-debit  time/iter
448050 2.5090e-03 2.2949e-06  5.6361e-03 4.3690e-03 5.5371e-09 -
4.0486e+03 -9.5124e-01    1020:40:42
Force vector: (1 0 0)     pressure          viscous             total
pressure       viscous            total
zone name                          force               force             force
coefficient   coefficient   coefficient
                                   [N]              [N]              [N]
------------------------ -------------- -------------- -------------- -------------- -
------------- -------------------
wall (perete)          -2519.2613   39.446605  -2479.8147 -4113.0797
64.40262      -4048.6771
------------------------ -------------- -------------- -------------- -------------- -
------------- -------------------
net                       -2519.2613   39.446605  -2479.8147 -4113.0797
64.40262      -4048.6771
====================================
==================
```

Maximal static pressure in the rocket engine's burning chamber: 433.15 [atm]

=====================================
===================

Mesh model was based on:

```
93751 quadrilateral cells, zone  2, binary.
    17 2D mass-flow-inlet faces, zone  3, binary.
   707 2D mass-flow-inlet faces, zone  4, binary.
    86 2D mass-flow-inlet faces, zone  5, binary.
   383 2D mass-flow-inlet faces, zone  6, binary.
   126 2D pressure-outlet faces, zone  7, binary.
   502 2D axis faces, zone  8, binary.
    53 2D mass-flow-inlet faces, zone  9, binary.
    50 2D mass-flow-inlet faces, zone 10, binary.
  2046 2D wall faces, zone 11, binary.
 185517 2D interior faces, zone 13, binary.
  95737 nodes, binary.
  95737 node flags, binary.
```

6. Conclusions

Based on **Case 1**, **Case 2** and **Case 3** CFD simulation results, the authors will consider the next conclusions:

1. – **Case 1** performs the validation of the CFD results (***p*** and ***F***) within a 5% relative error versus the experimental available data using ideal gas parameters. At that stage, there was no reason to spend more time to evaluate the real gas parameters in order to increase accuracy;
2. – **Case 2** was considered like the minimal nozzle's diameter before facing severe damage of the entire rocket engine, with possible negative impact of the reliability

of the whole product, because of increased mechanical and thermal stress combination may occur after ignition of the engine with further unknown functionality impact;

3. – like a consequence of the **Case 2** study, there has been made a special call to a further analysis for all the previous bed test results with ***p*** and ***F*** close to the **Case 2** CFD output values, in order to quantify the probability of facing an unknown decrease of the nozzle's diameter for unknown reasons during the 6 – 8 s of normal operation of the DUT;
4. – the possible decrease of the critical section diameter in the engine's nozzle area down to 10.0 mm will increase the probability to register a mission disaster;
5. – it was proved that any decrease of the critical section diameter of the nozzle lower than **Case 2** will damage the engine with possible impact against humans and/or technology in the nearby area, so **Case 3** will be considered within a 95% probability for a total damage of the DUT and mission;

References:

[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.

ADENDUM – CFD static pressure simulation results inside the DUT

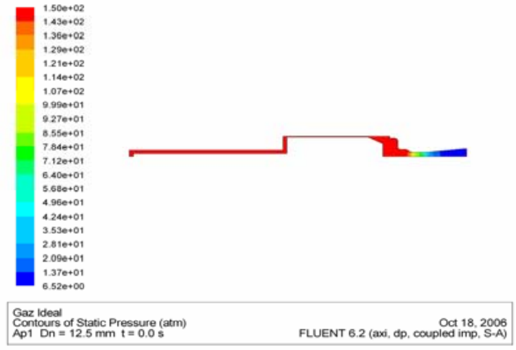

Fig. 2

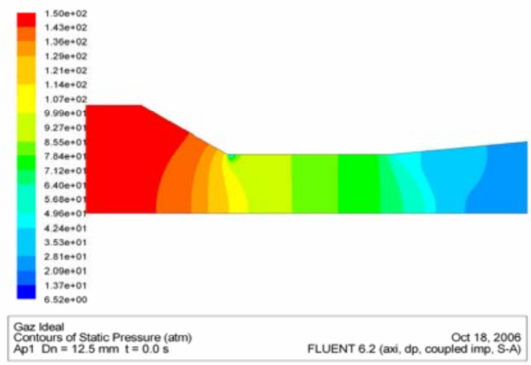

Fig. 3

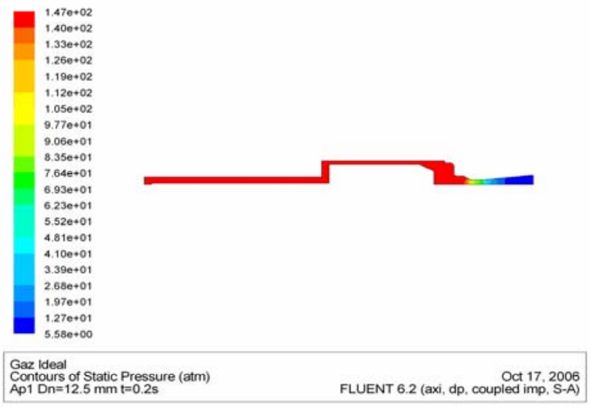

Fig. 4

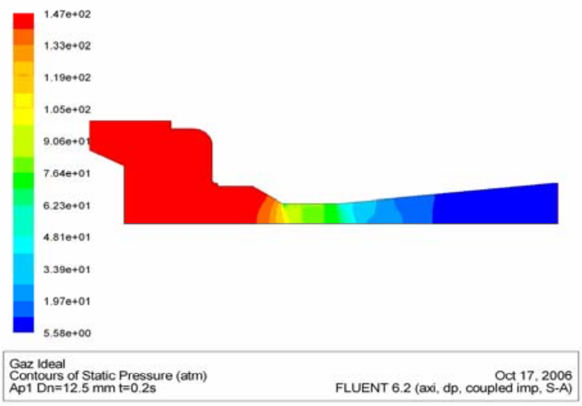

Fig. 5

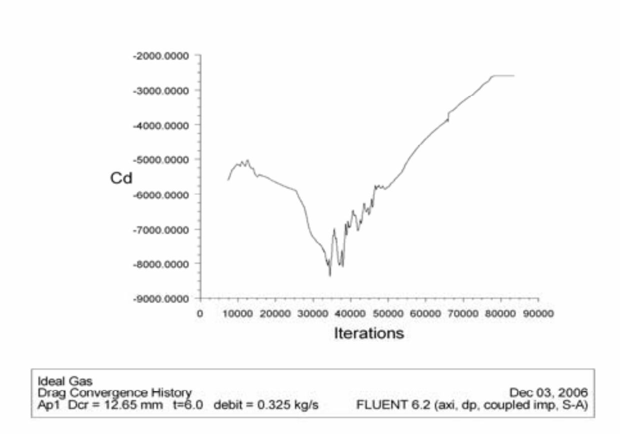

Fig. 6

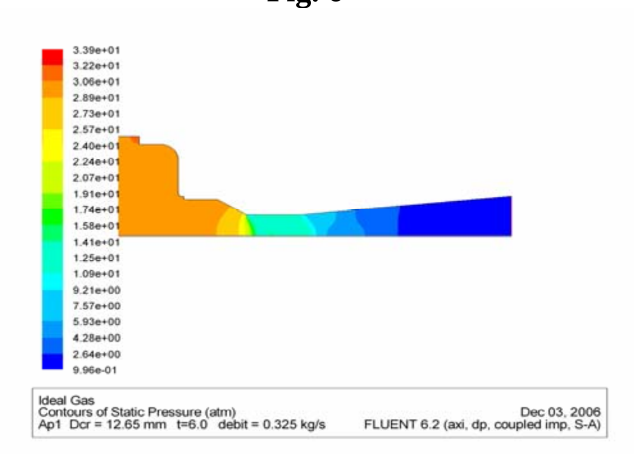

Fig. 7

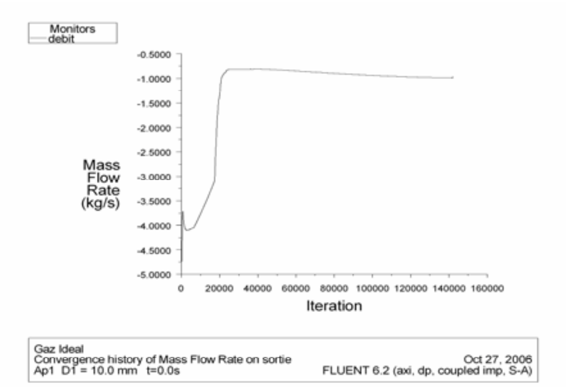

Fig. 8

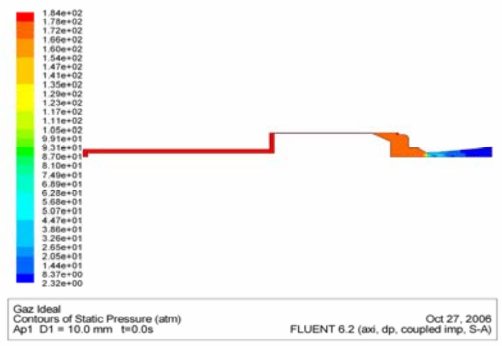

Fig. 9

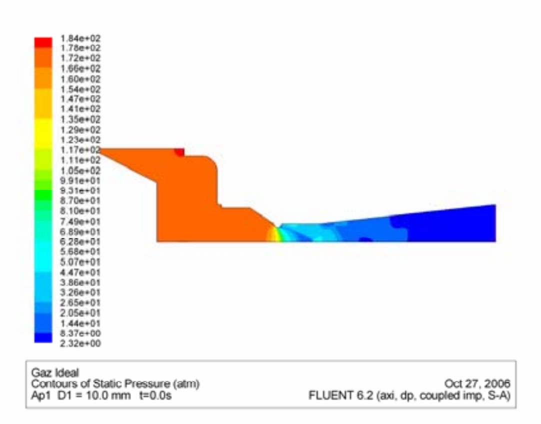

Fig. 10

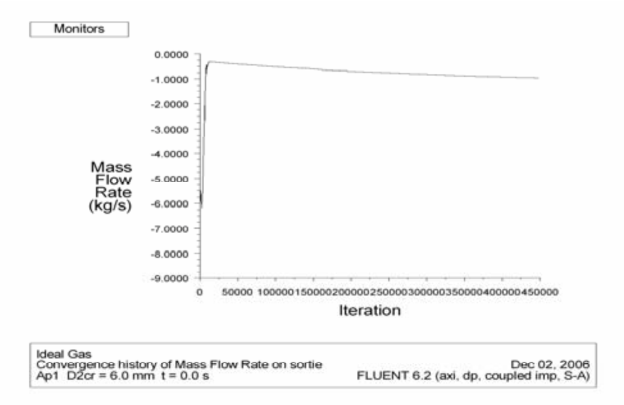

Fig. 11

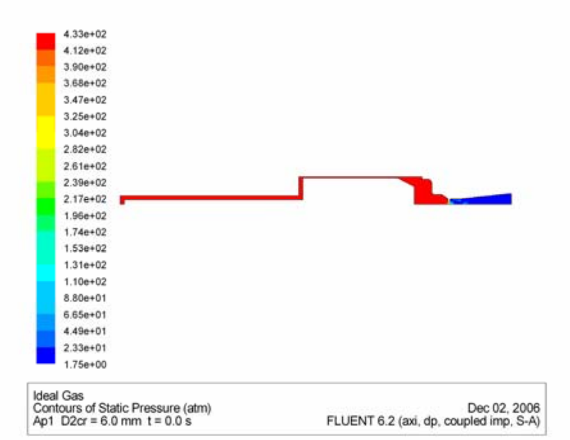

Fig. 12

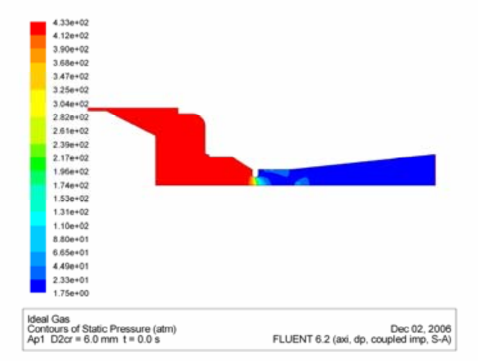

Fig. 13