CFD simulation on J-2X engine exhaust in the center-body diffuser at B-2 facility

Xiao-Yen Wang, Thomas Wey, Robert Buehrle
NASA Glenn Research Center

Thermal & Fluids Analysis Workshop
(TFAWS 2008)
San Jose State University, San Jose, CA
August 18-22, 2008

Contents

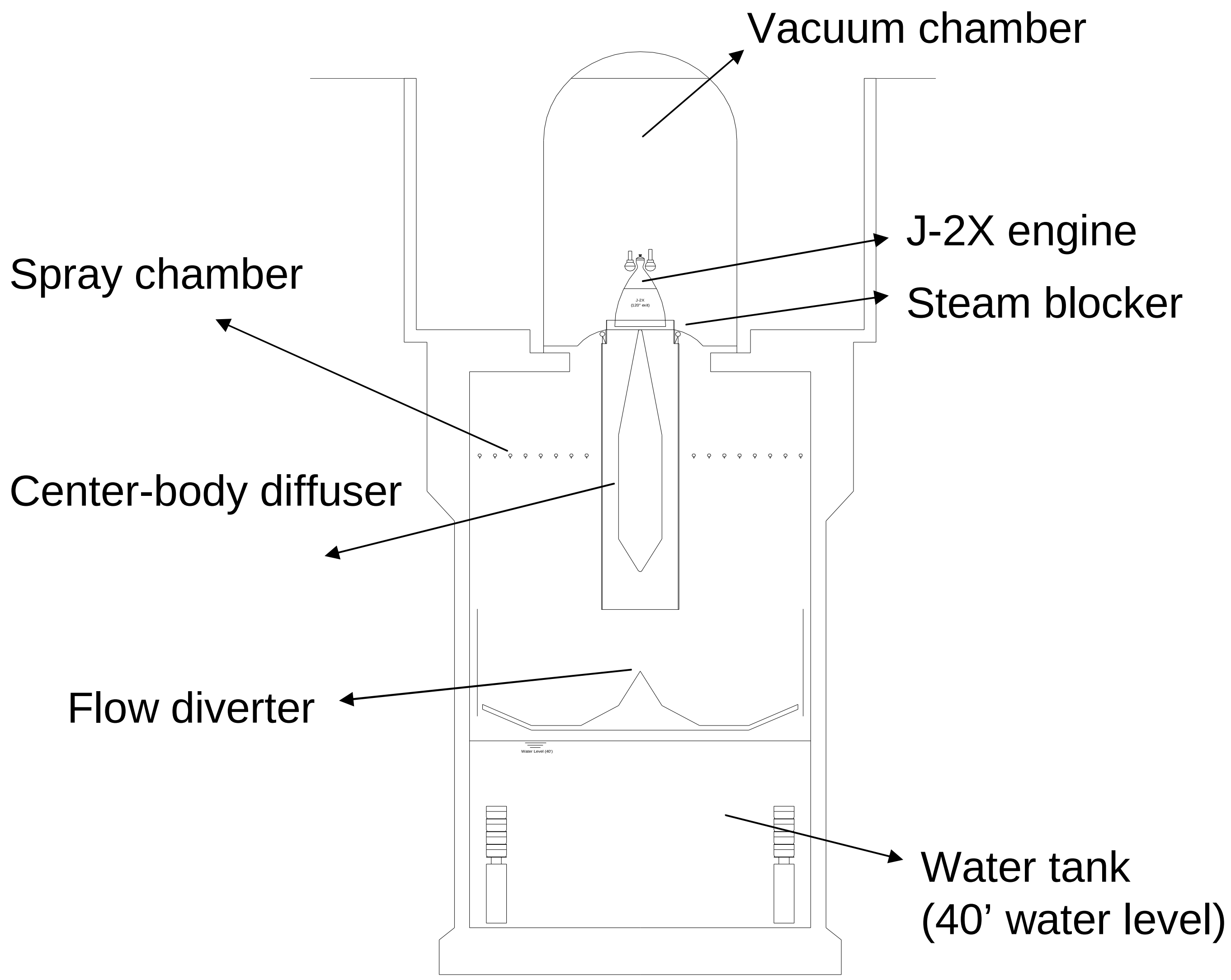


Fig. 1 A sketch of the B-2 facility.

Background

- The B-2 facility in the Plum Brook Station (PBS) was originally designed to test rockets up to 100,000 lbf thrust.
- B-2 facility must be adapted to test the bigger engines, such as J-2X engine (294,000 lbf thrust), which results in more severe thermal environment and larger scale of energy.
- A center-body diffuser at the downstream of the engine is used to slow down the gas before hitting the water. The water is injected inside the spray chamber to cool the hot exhaust. The steam is sprayed through steam blocker to prevent the backflow in the event of J-2X engine shutdown.

J-2X engine performance

Table. Engine designation

	J-2X
fuel	H_2
oxidizer	O_2
chamber o/f ratio	5.5
chamber pressure (psia)	1338
propellant mass flow rate (lbm/s)	650
nozzle throat diameter (in)	11.98
nozzle exit area ratio	92
chamber subtraction area ratio	2.38
thrust (lbf)	294,000

- Use the chemical equilibrium compositions and applications (CEA) code

Table. J-2X performance (CEA results)

	COMB END	THROAT	EXIT
P, BAR	85.62	51.066	0.0592
T, K	3406.75	3210.22	1005.01
RHO, KG/CU M	3.8389	2.4501	9.28E-03
M, (1/n)	12.7	12.806	13.103
Cp, KJ/(KG)(K)	7.52	6.81	2.895
Gammas	1.15	1.15	1.28
SON VEL,M/SEC	1601.3	1549.8	903.7
MACH NUMBER	0.26	1	4.92
VISC,MILLIPOISE	1.0263	0.9823	0.3792
Conductivity, Mill Watts/(CM)(K)	14.71	12.66	1.686
PRANDTL NUMBER	0.5246	0.5283	0.6512
MOLE FRACTIONS			
H2	0.301	0.301	0.307
H2O	0.64	0.655	0.693

CFD simulation on the J-2X exhaust

- Assume ideal gas, use the hot gas properties obtained from CEA
- Use CE/SE 2D/axisymmetric Euler code
- Generate mesh using MSC Patran
- Use non-dimensional variables:

$$p^* = p / \rho_e u_e^2, \quad \rho^* = \rho / \rho_e$$

$$x^* = x / L, \qquad T^* = T / T_e$$

$$u^* = u / u_e, \qquad t^* = t / (L / u_e)$$

where $\rho_e = 9.94e-3\,\mathrm{kg/m^3}$, $u_e = 4446.2\,\mathrm{m/s}$, and $T_e = 1005\,\mathrm{K}$ are the density, velocity, and temperature at the engine nozzle exit, and $L = 1\,\mathrm{m}$ and $\rho_e u_e^2 = 1.965\mathrm{e}+5\,\mathrm{N/m^2} = 28.5\,\mathrm{psia}$.

Assumptions used in CFD simulation

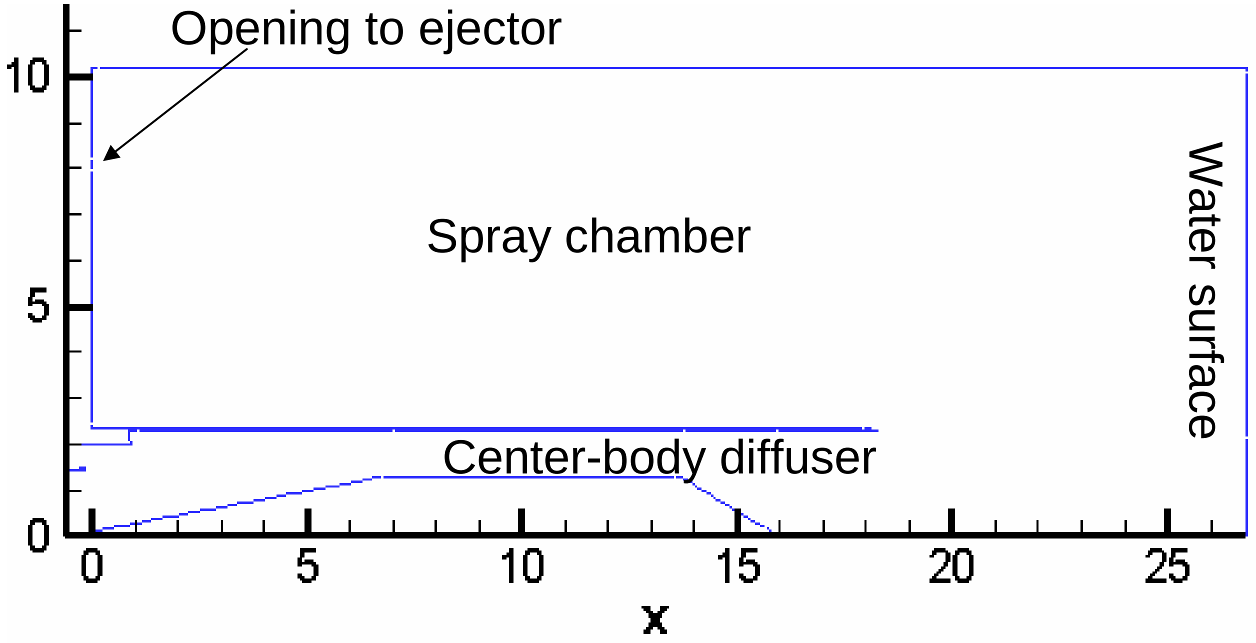


Fig.1 Computational domain.

- Water surface is approximated by using a solid wall.
- At the steam blocker, assuming total pressure p_t = 165 psia, total temperature T_t = 459 K, mass flow rate of steam is 147 lb/s.
- For the core flow, assume the total pressure pt = 1338 psia, total temperature Tt = 3552 K, mass flow rate of steam is 650 lb/s.

- 18119 mesh points, 35332 triangular elements, transient analysis

Initial and Boundary Conditions:

- Initial conditions: (B-2 evacuated conditions)
  $p^* = 0.0056$ ($p = 0.16$ psia)
  $u^* = 0$
  $v^* = 0.0$
  $\rho^* = 0.8047$

- Inlet of engine exhaust conditions:
  $p^* = 0.0368$
  $u^* = 0.9957$
  $v^* = 0.0$
  $\rho^* = 1.1092$

- Inlet of steam conditions:
  $p^* = 0.00723$
  $u^* = 0.7028$
  $v^* = 0.0$
  $\rho^* = 0.7313$

- Opening to ejector conditions:
  $p^*_{back} = 0.014$ ($p = 0.4$ psia)

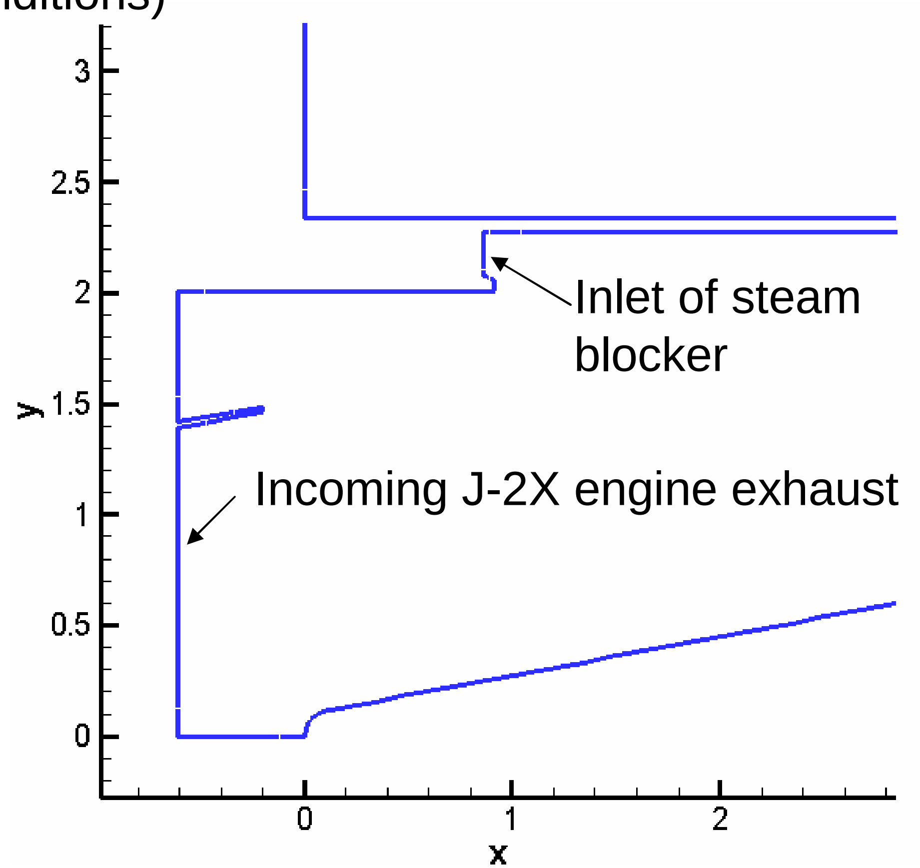
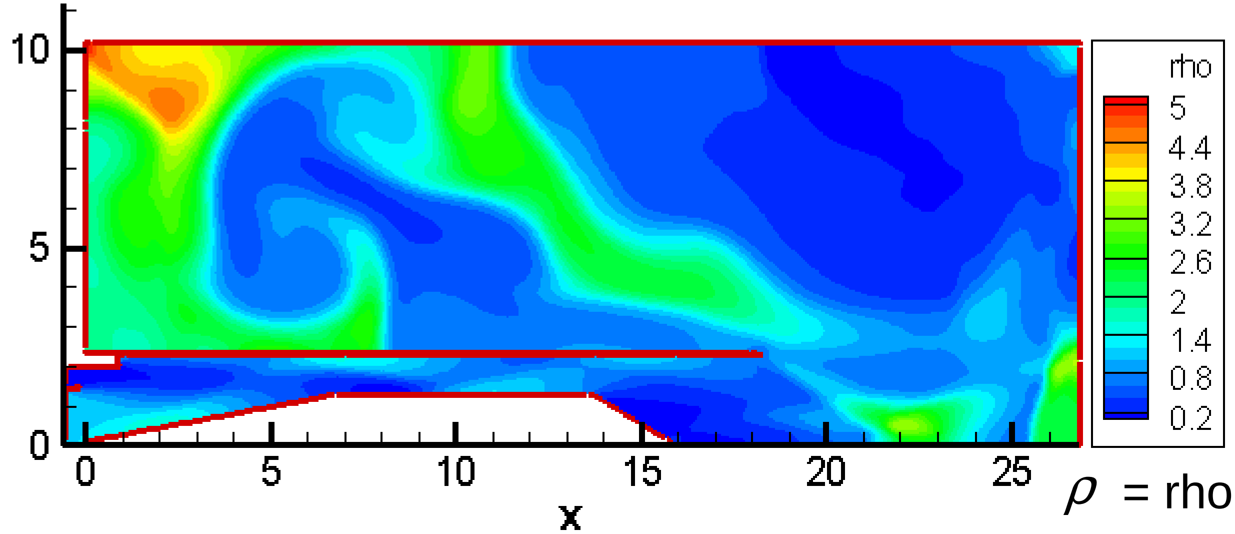


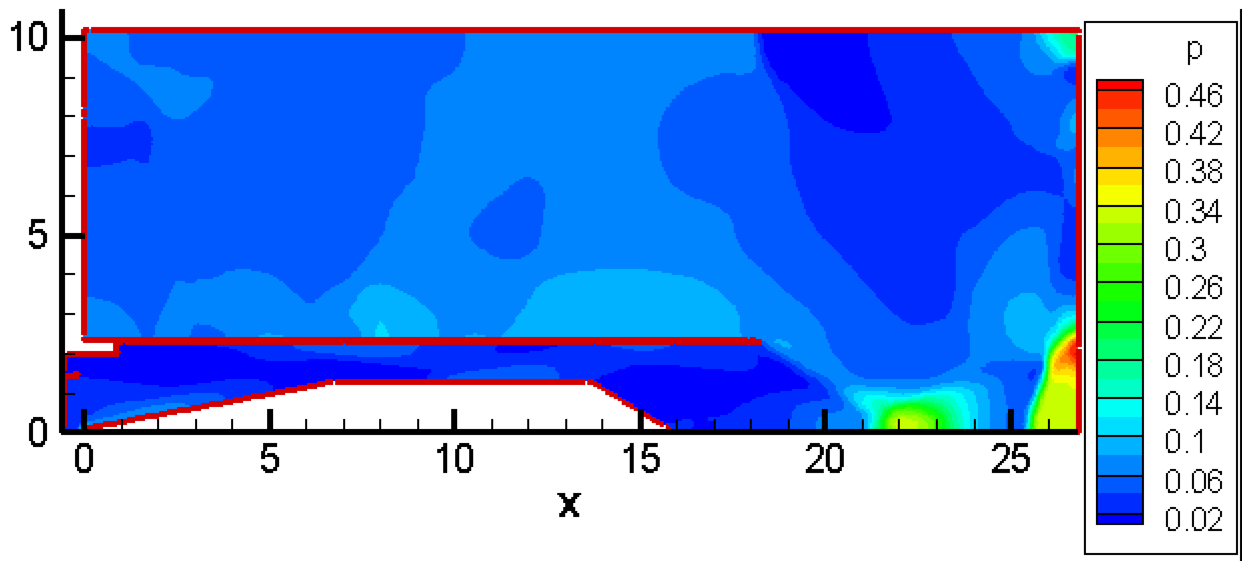
Fig. 2 Close view at the domain inlet.

CFD CESE Results at t = 0.0787 s.

- The flow variables plotted in the contour plots are non-dimensional.

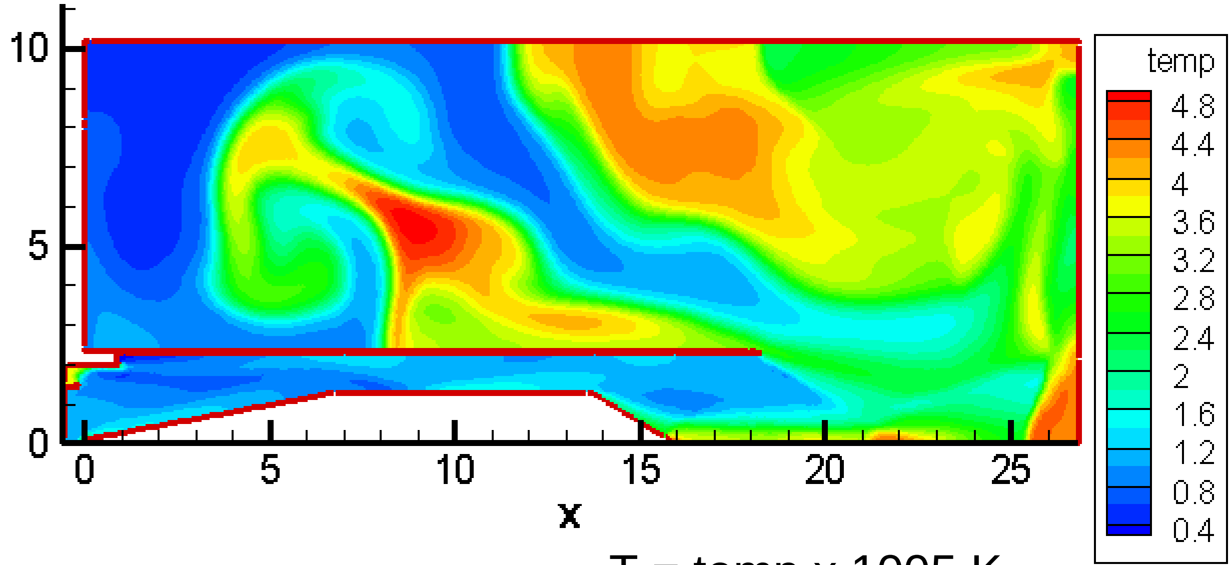


ρ = rho x 9.94e-3 kg/m^3

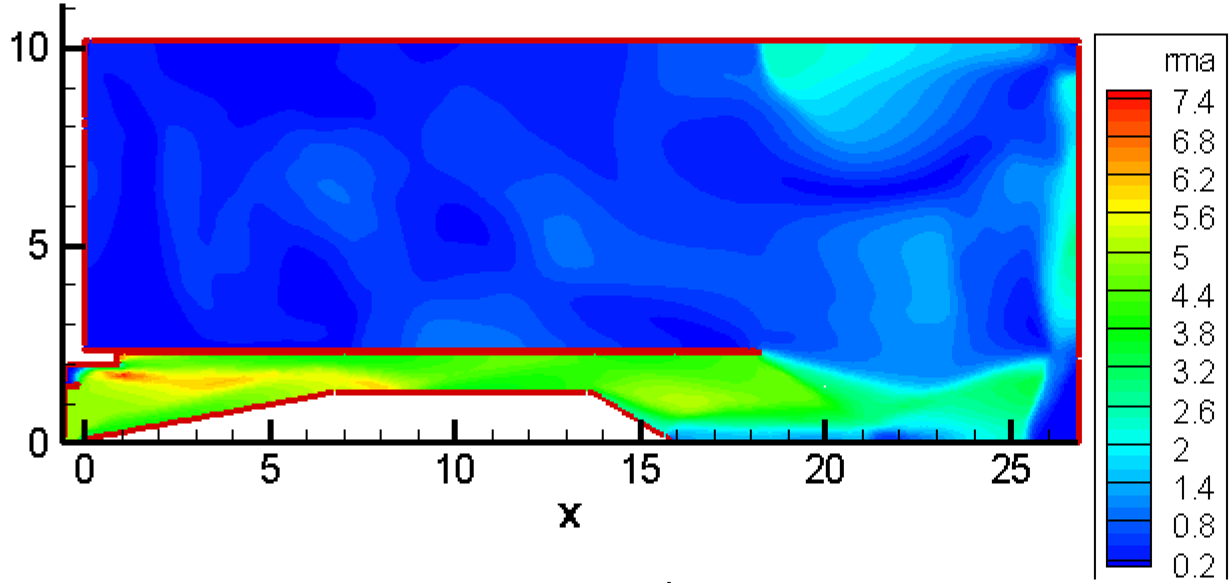


P = p x 28.5 psia

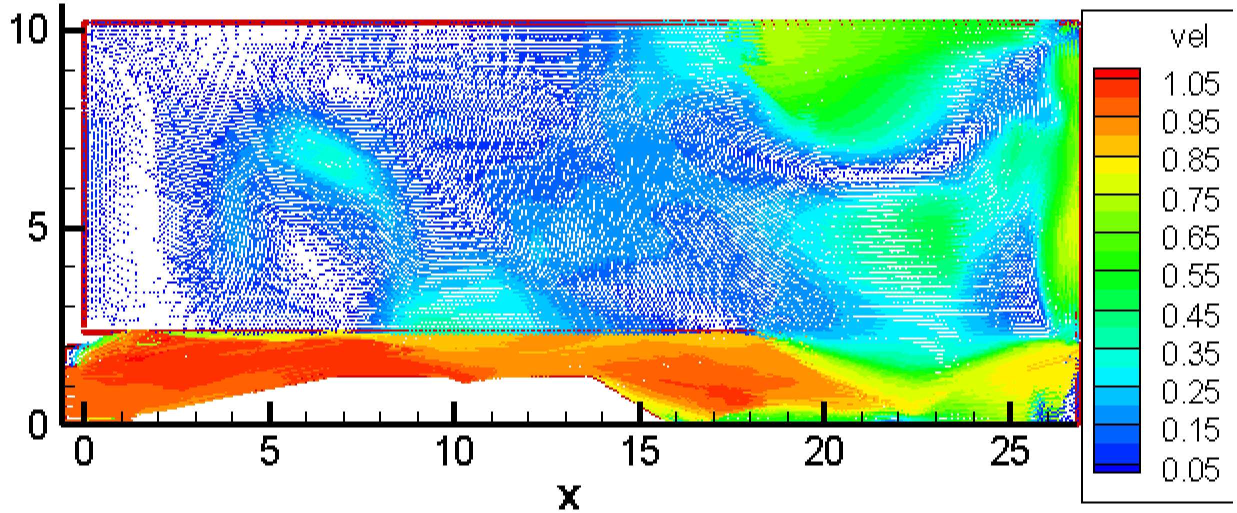
CFD CESE Results at t = 0.0787 s.



T = temp x 1005 K



CFD CESE Results at t = 0.0787 s.



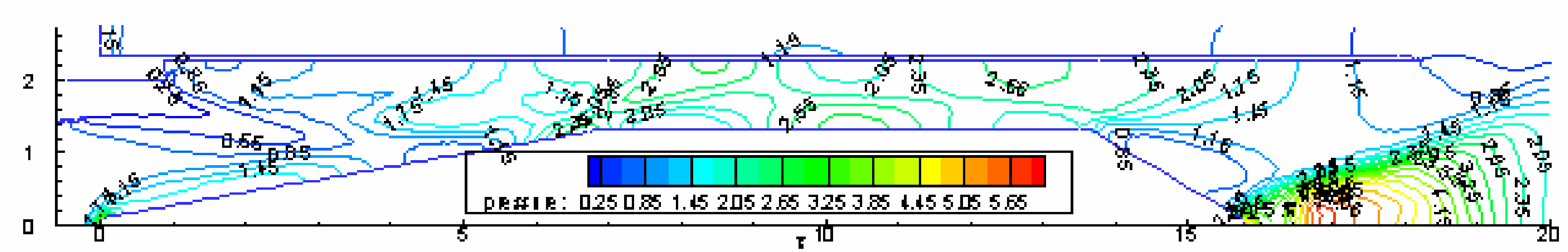
U = vel X 4.4 km/s

Velocity vector plot.

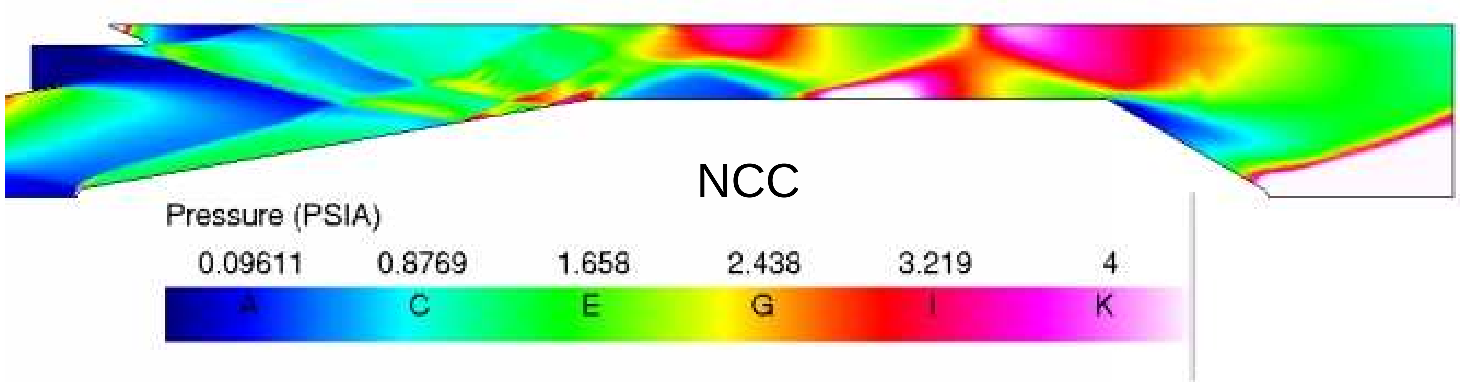
- Within 0.0787s, the flow field inside the diffuser reaches steady-state. The shock waves in the spray chamber still bounce back and forwards.
- Shock waves exist at the exit of the diffuser.
- The animation for the time length of 0.0787 s is attached as follows. All the variables in the animation are non-dimensional variables.

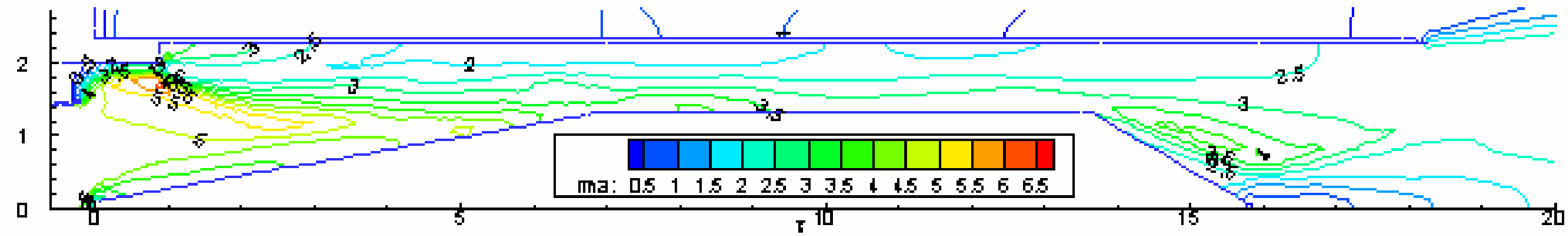
Comparison of CESE and NCC results

- National Combustion Code (NCC) developed in GRC: used in an independent analysis on the same problem.



(a) CESE result of the pressure





(b) CESE result of the Mach number

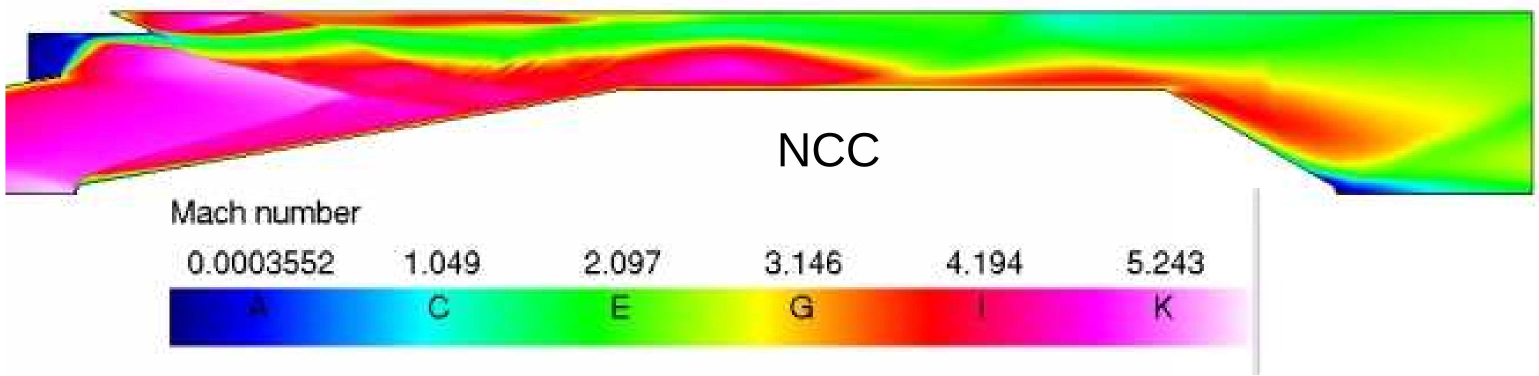
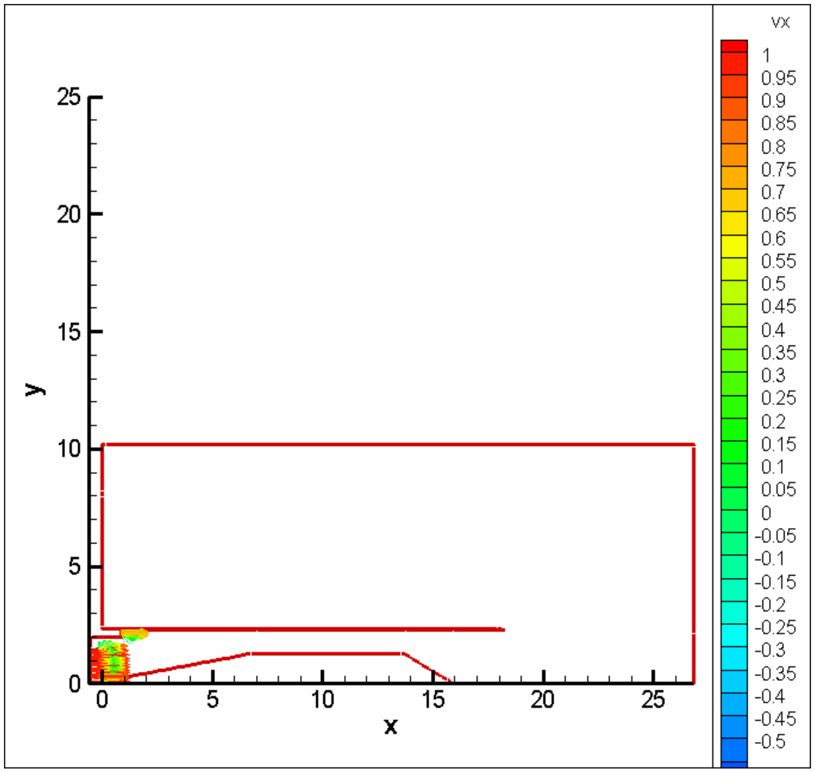


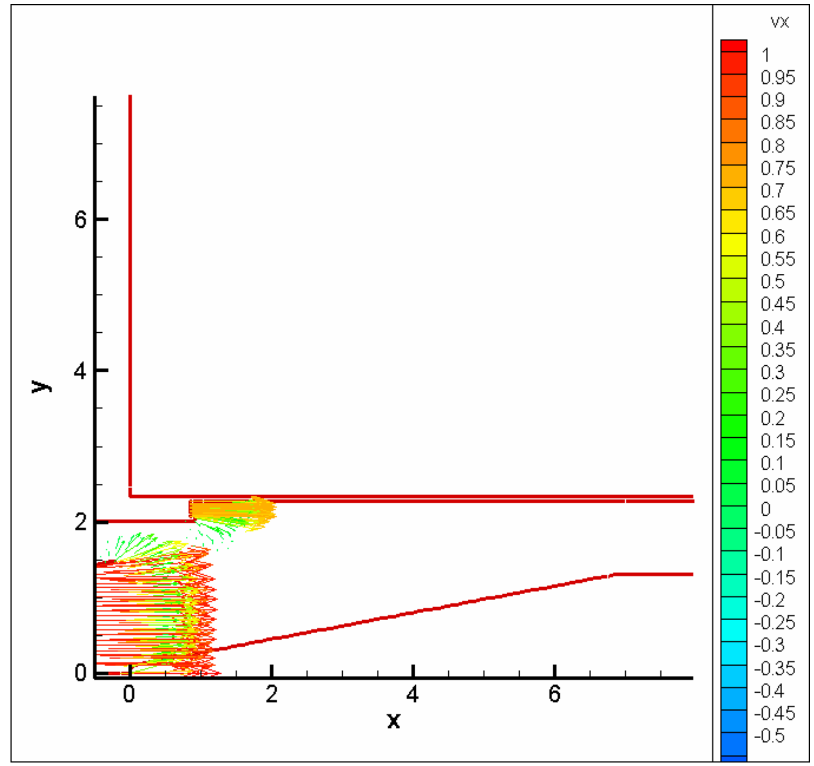
Fig. 6 CESE and NCC numerical results comparison.

Summary

- The J-2X engine exhaust in the center-body diffuser and spray chamber at B-2 facility is simulated using CESE method.
- The shock waves pattern was captured by CESE method agree well with the corresponding results obtained by using NCC code.
- Further analysis is needed to validate the design of the B-2 facility for testing rocket engines with up to 300,000 lbf thrust.

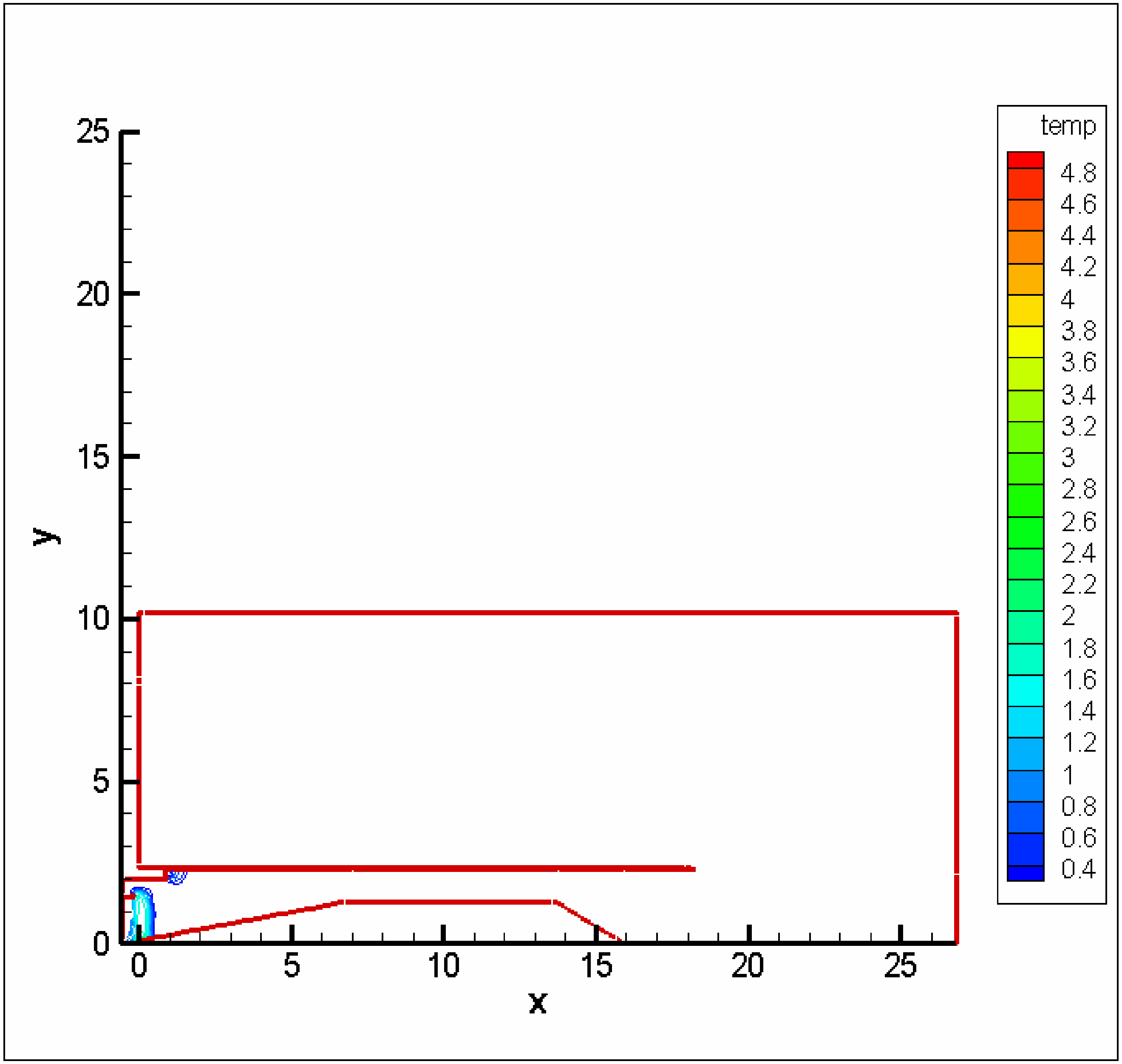
• animation of velocity vector

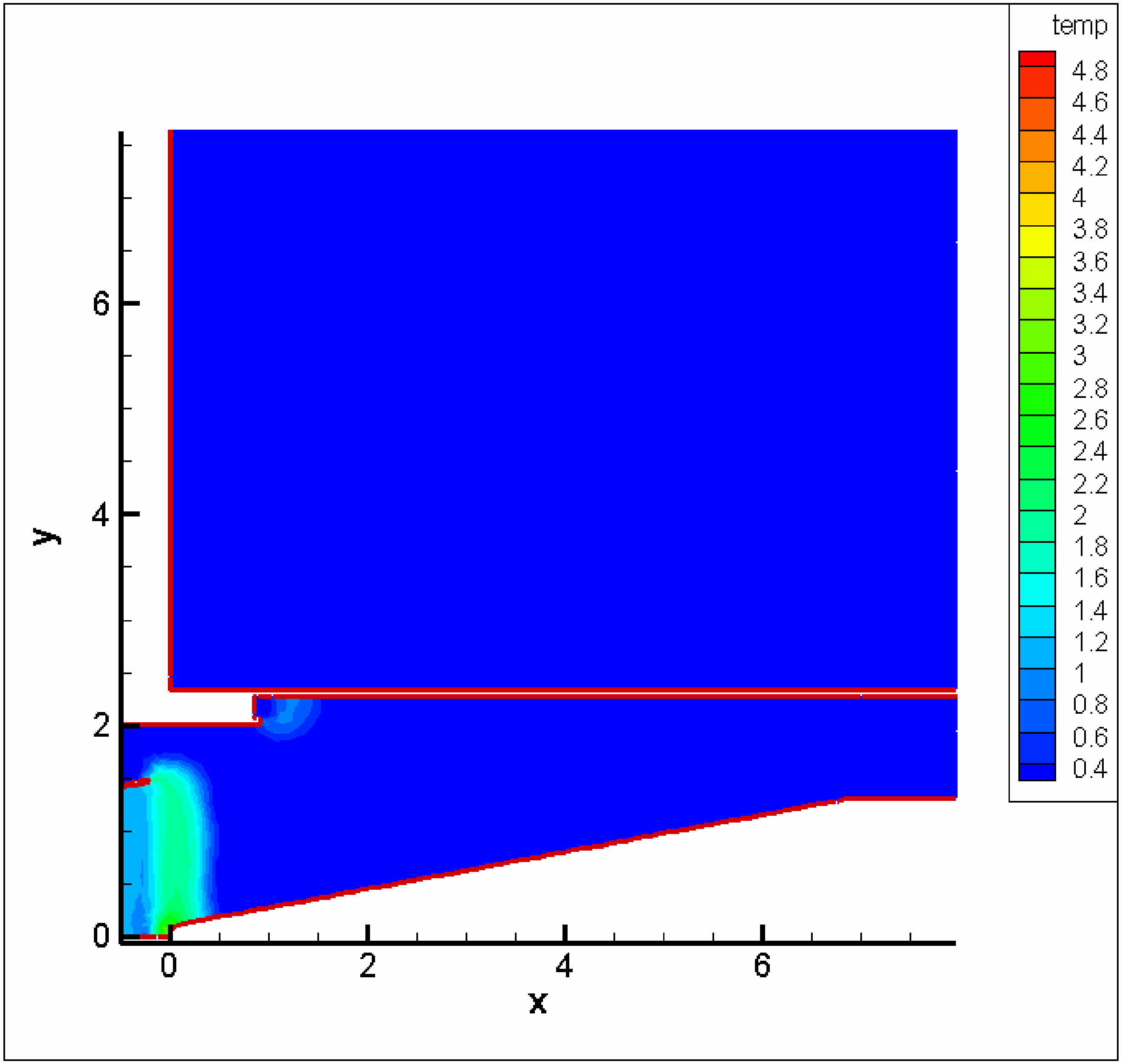




Close view at the inlet

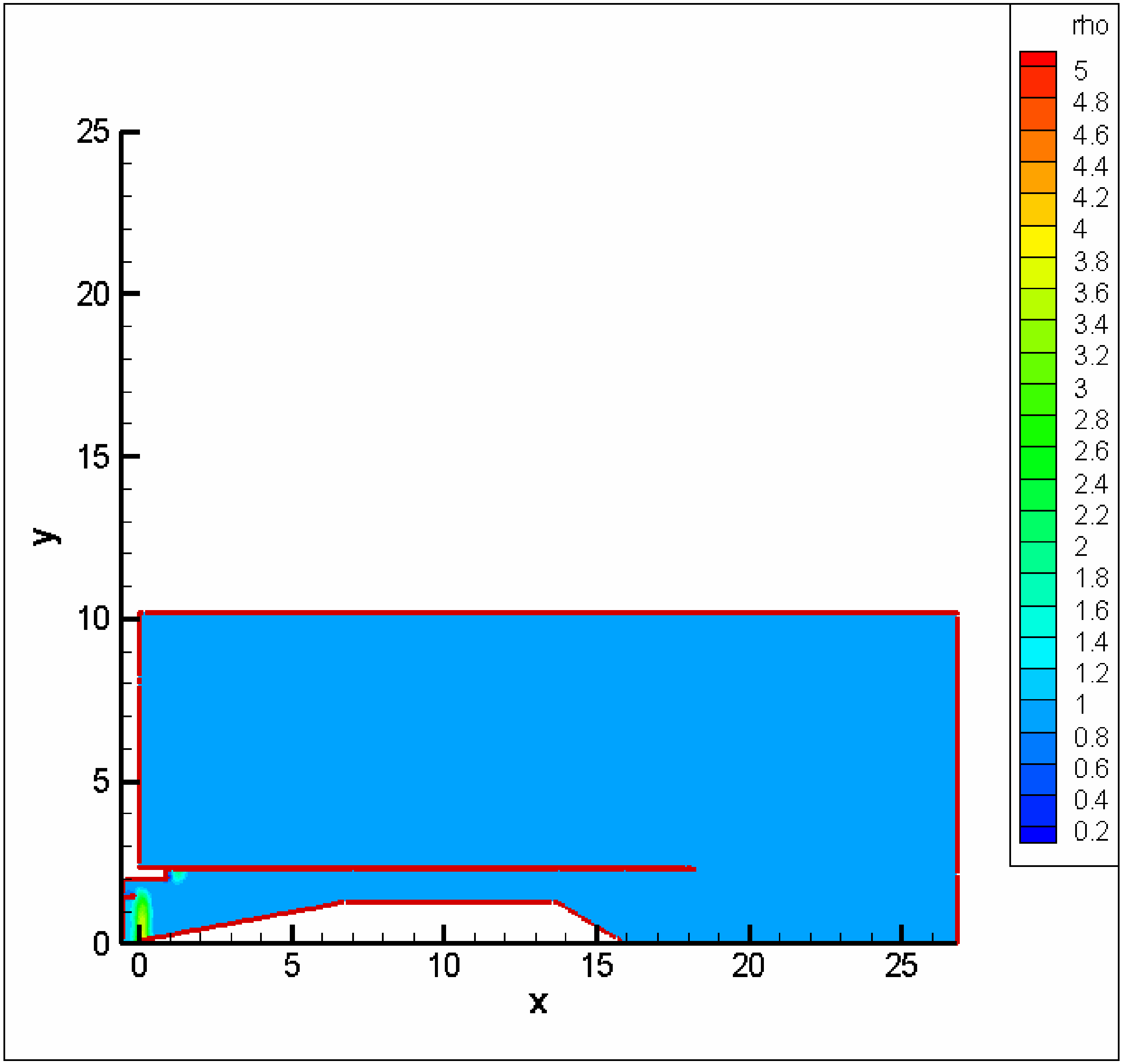
- animation of temperature

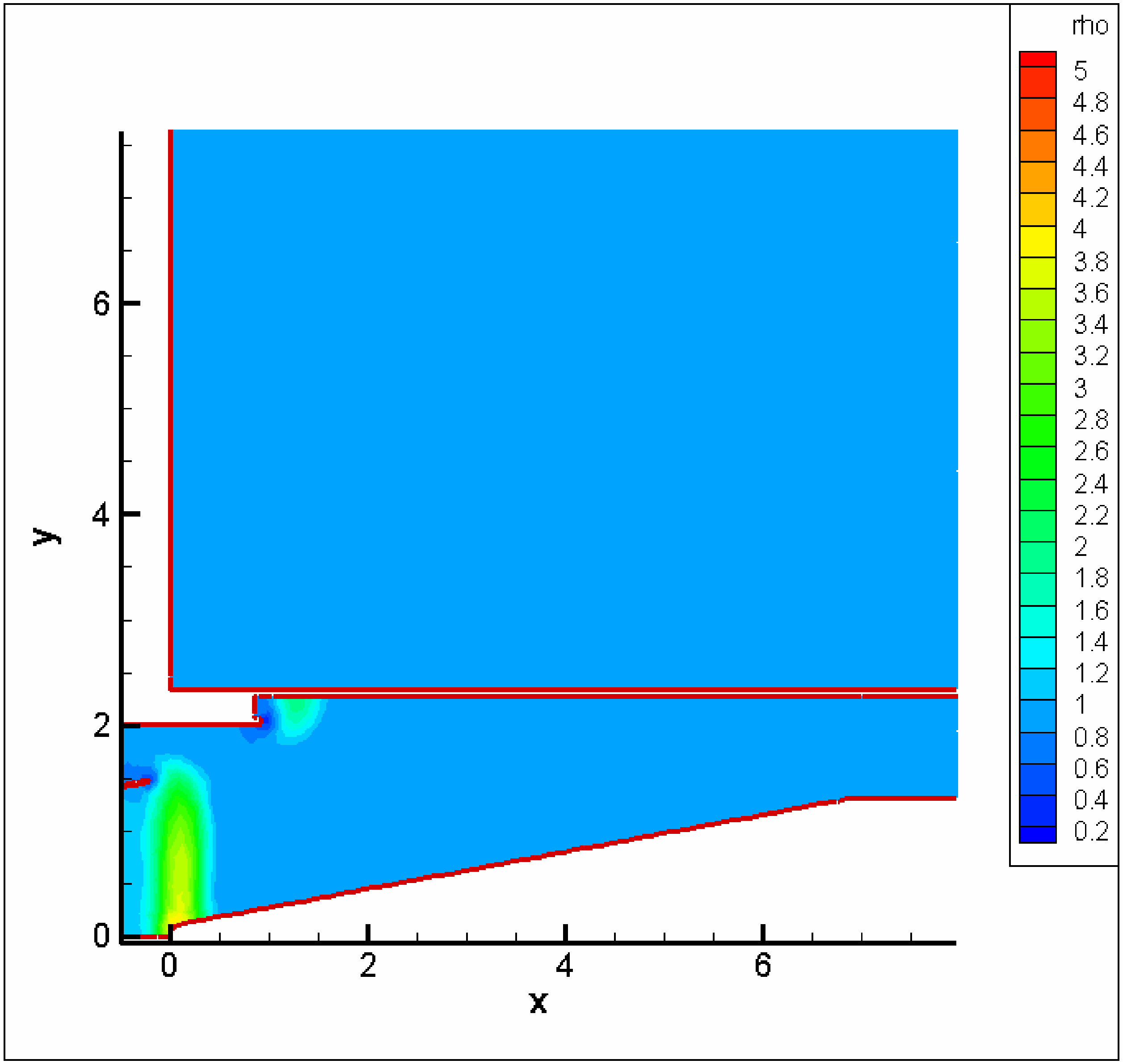




Close view
at the inlet

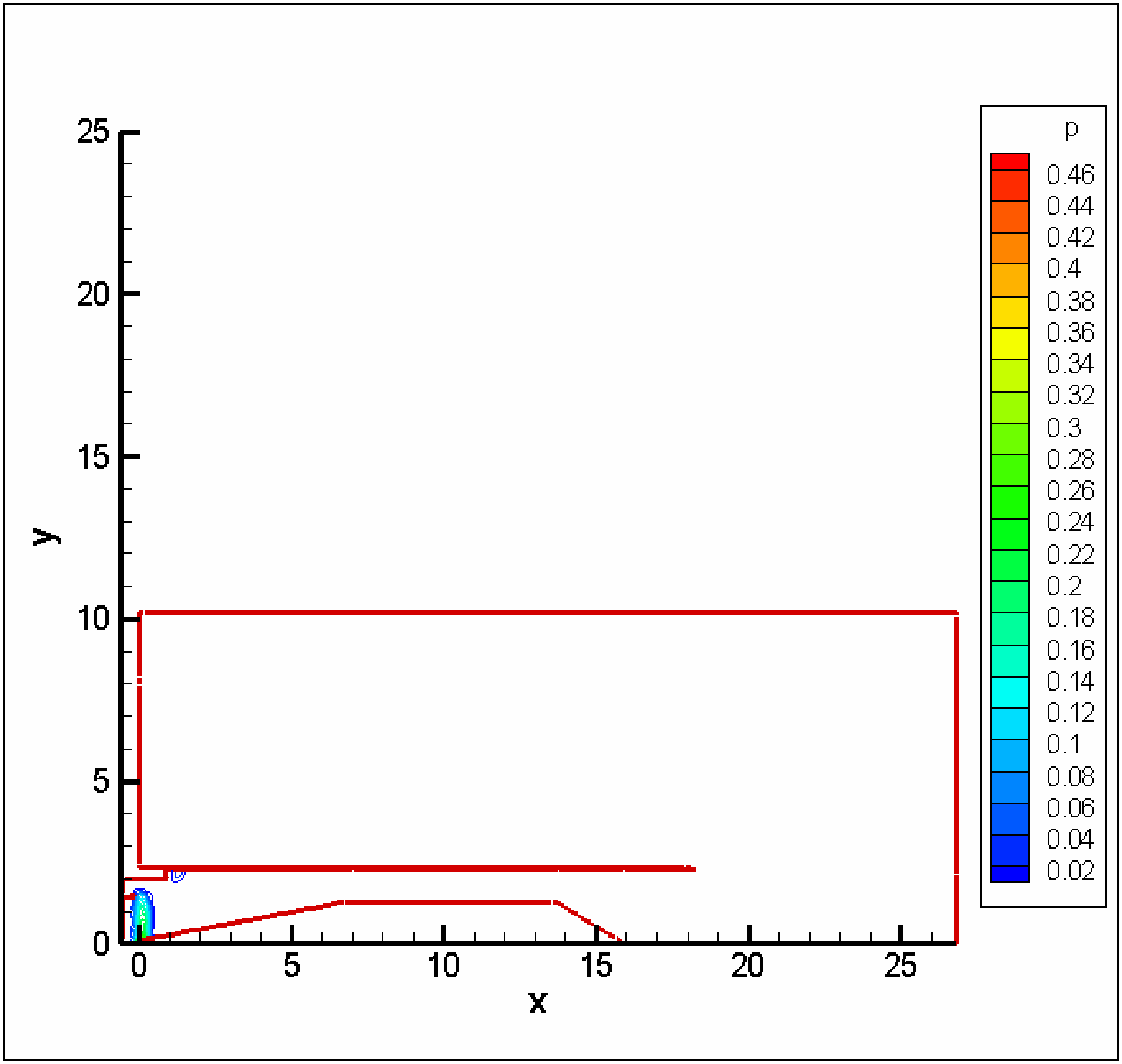
- animation of density

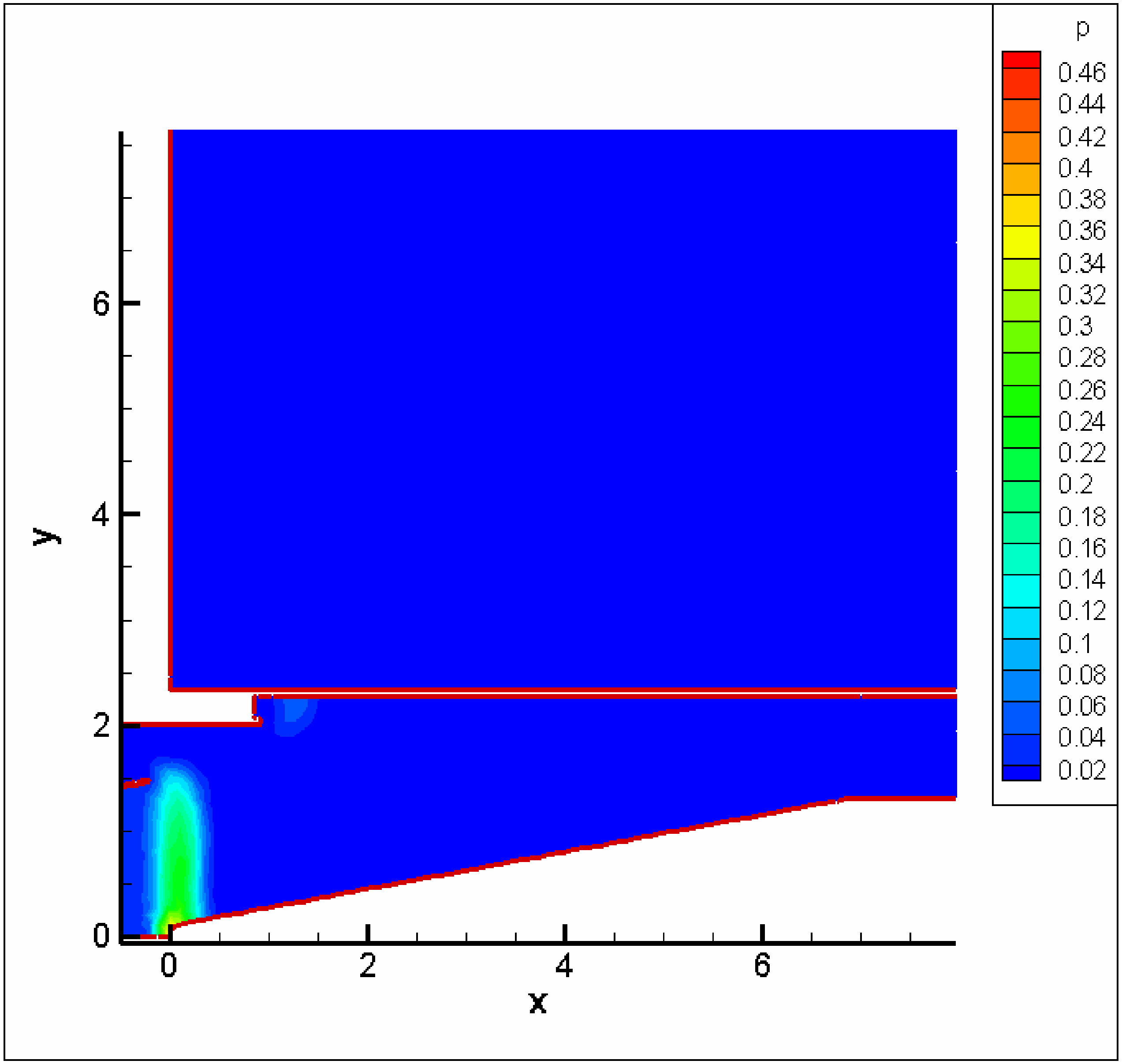




Close view
at the inlet

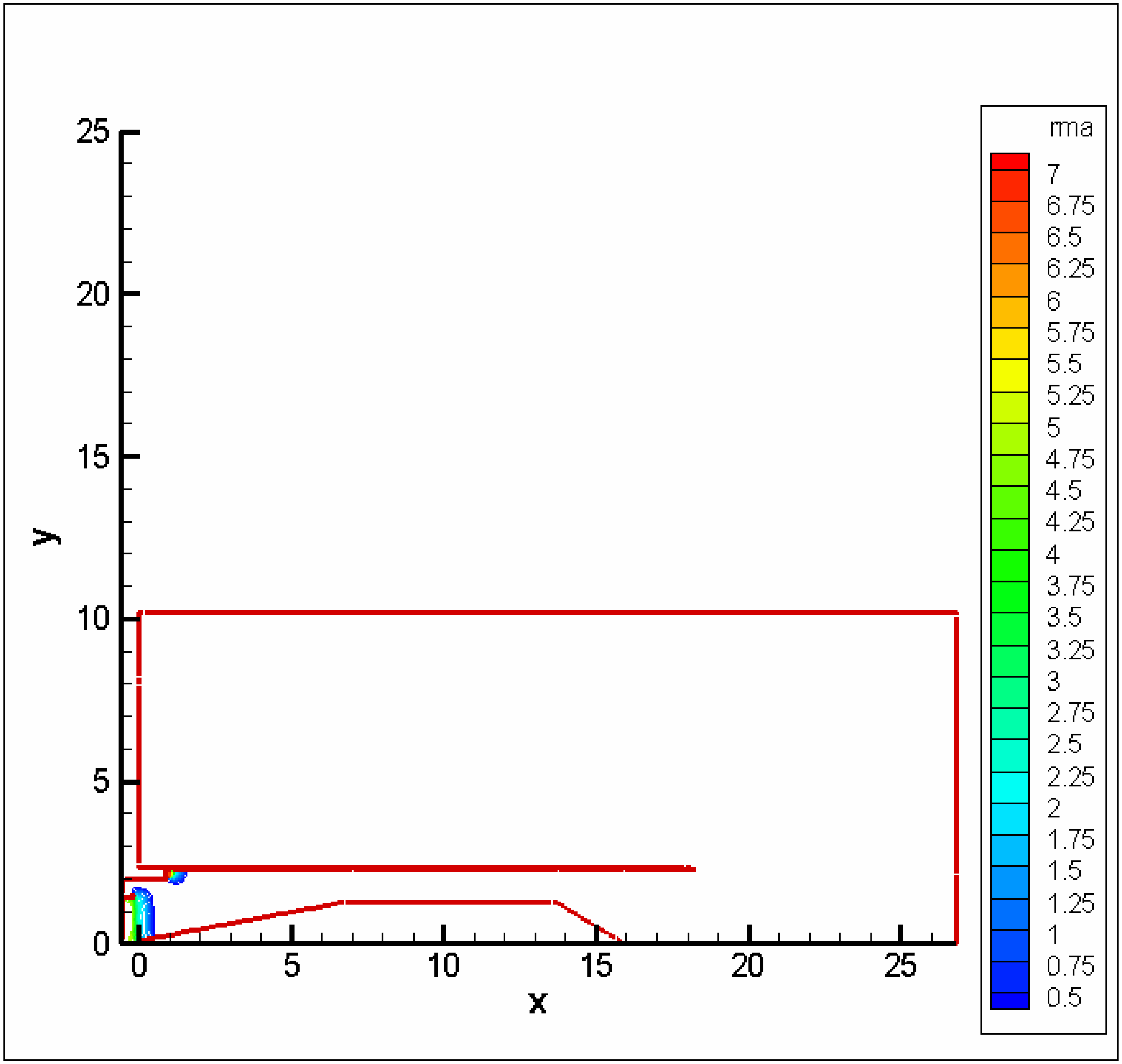
- animation of pressure

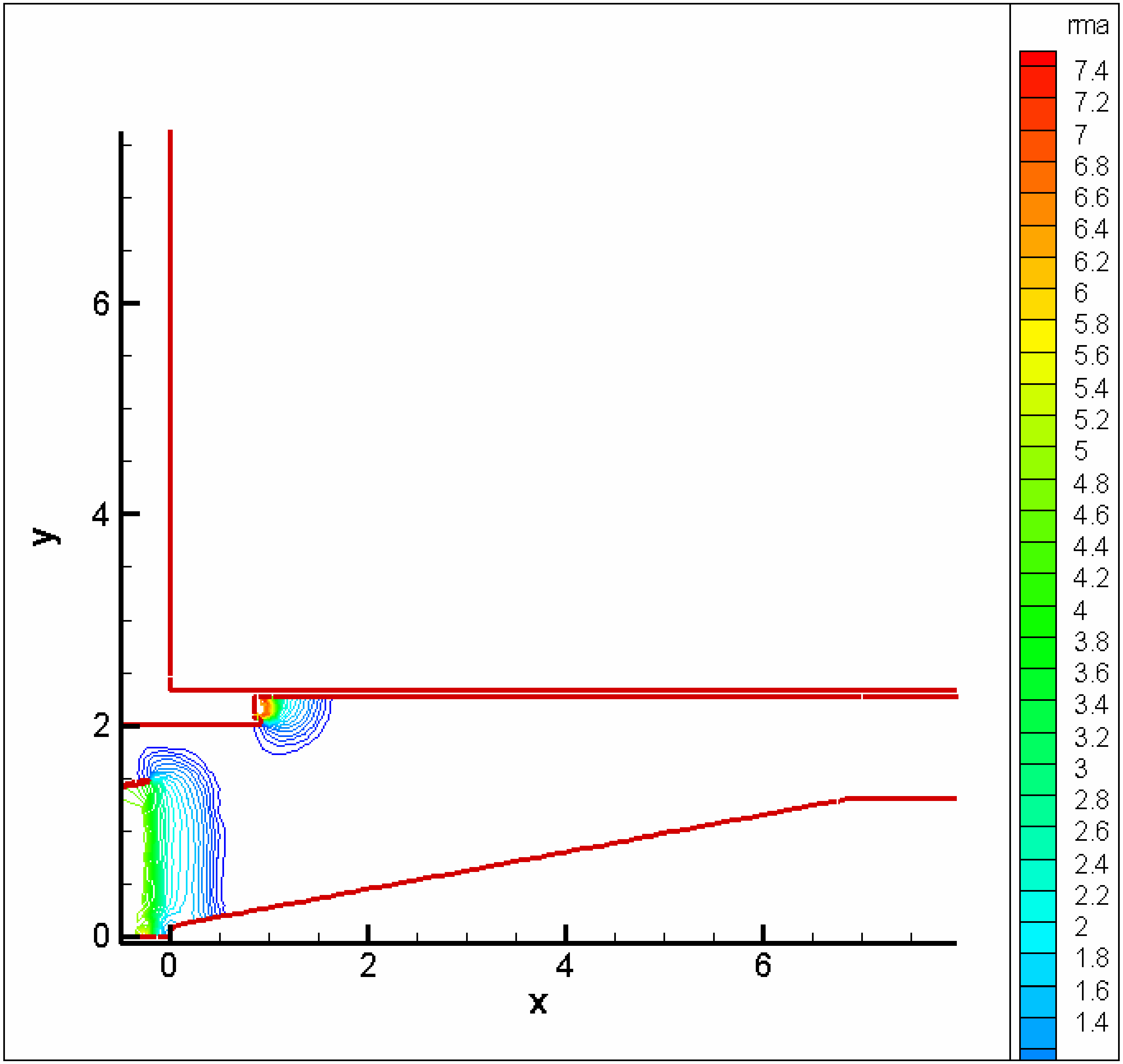




Close view
at the inlet

- Animation of Mach number





Close view
at the inlet