CHAM Limited

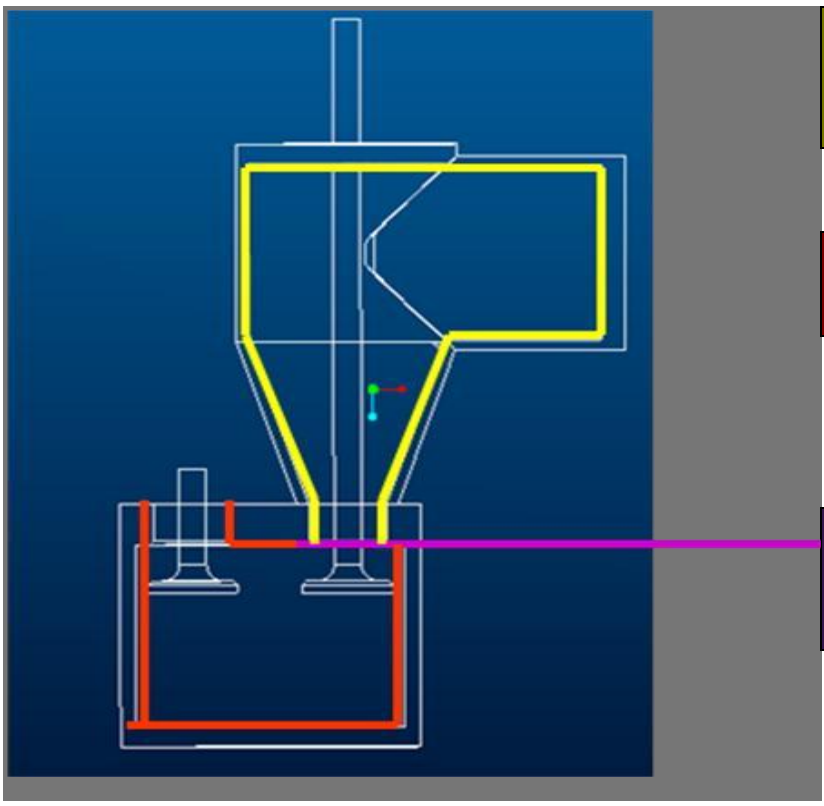
Pioneering CFD Software for Education & Industry

CHAM Case Study – Air Injector Model

Transient - PHOENICS demonstration case

Following unsuccessful attempts using another mainstream CFD product, EA Technical Solutions Ltd approached CHAM for the purpose of obtaining a CFD code suitable for investigating the gas mixing processes within various air injector models. The problem specified below involves the transient purging of a hot gas chamber with cold gas from a pressurized chamber. In this case, there are pre-set inlet and exit valve positions with impermeable membranes. The flow field is stagnant initially, but the flow is initiated by the instantaneous removal of the two (purple) membranes.

The geometry of the air injector was generated by PTC's Pro-Engineer and exported as a 3D solid model in STL format – a format readily accepted by PHOENICS.

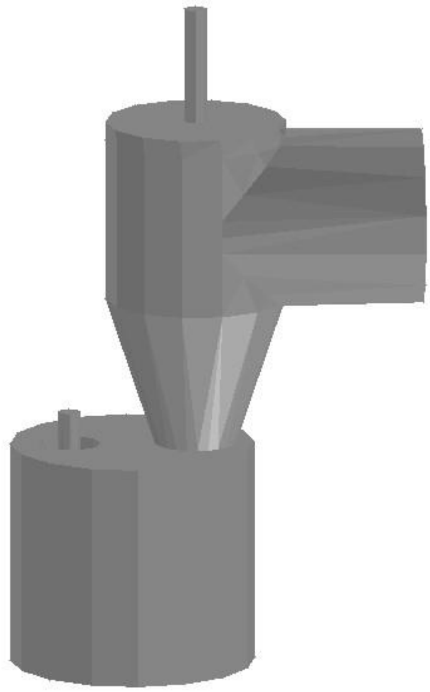


The yellow box holds stationary cold air at a few bars pressure.

The red box holds exhaust gas at about ambient pressure

The purple line is removed for a short time to discharge cold air into the hot gas.

The requirement was to find the time taken for the cold air to just start to exit through the exhaust valve. Different model variations have the inlet valve moved into the neck to vary the cold flow direction.



Air Injector imported from CAD

<u>CFD Model Description</u>

Initial Conditions:
Hot Gas Chamber - Pressure 4 bar, Temperature 500K.
Cold Gas Chamber - Pressure 1 bar, Temperature 800K.
Stagnant flow in both chambers.

Conservation & Transport Equations:
Continuity, three momentum equations, static temperature, marker variable for cold chamber gas, turbulent kinetic energy and its rate of dissipation.

Boundary conditions:
Adiabatic walls with empirical, equilibrium, log-law wall functions.

Fluid properties:
Working fluid is air.

Density: Ideal-gas law.
Specific heat: C_p = 1064 J/kgK
Thermal conductivity: k=0.0495 W/mK Kinematic molecular viscosity:

$$\nu = -4.9468.10^{-6} + 4.5839.10^{-8}T + 8.0924.10^{-11}T^2$$

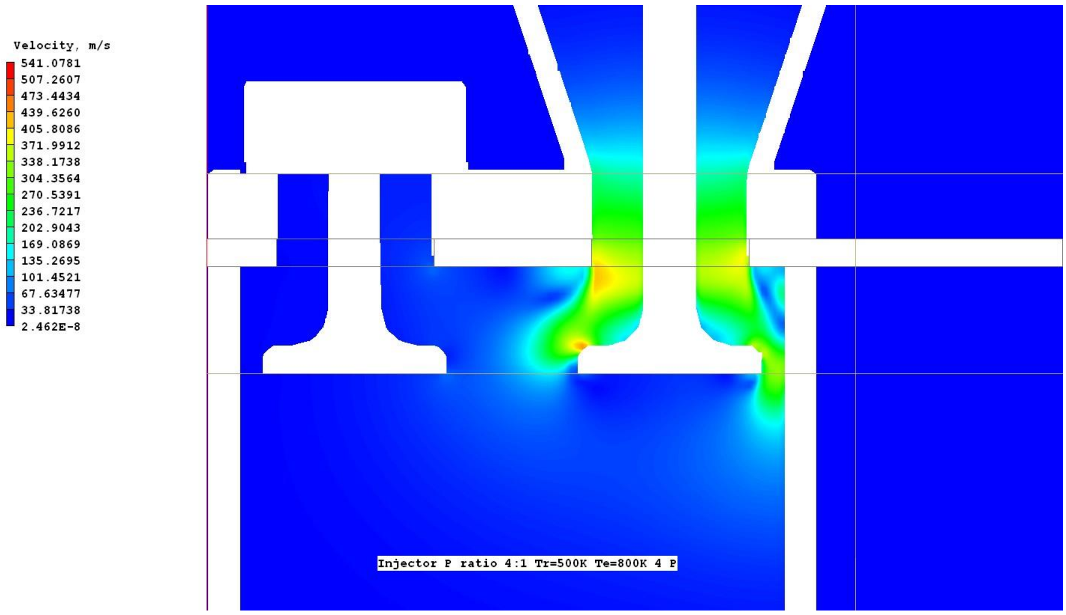
Numerical Parameters:
PARSOL Cartesian cut-cell solver with residual cut-cell volumes of 5%

Mesh: 179 * 66* 123 = 1.453 million cells
Time Duration of Simulation: 2ms
Time Step: 5µs (400 uniform time steps)
Typically 40 sweeps per time step

Note: As this was a demonstration case, there was no optimization made in respect of mesh, time stepping, relaxation practices, and iteration numbers.

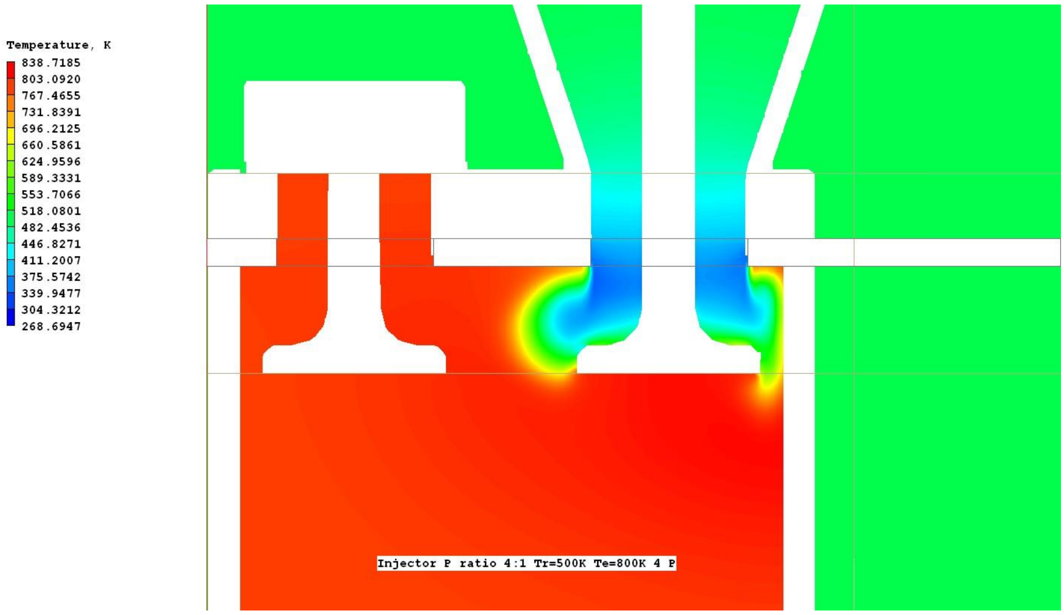
Version used: PHOENICS 2009 (64-bit Intel)
Elapsed run time: 78hrs Parallel with 4 processors.

<u>Result Images</u>

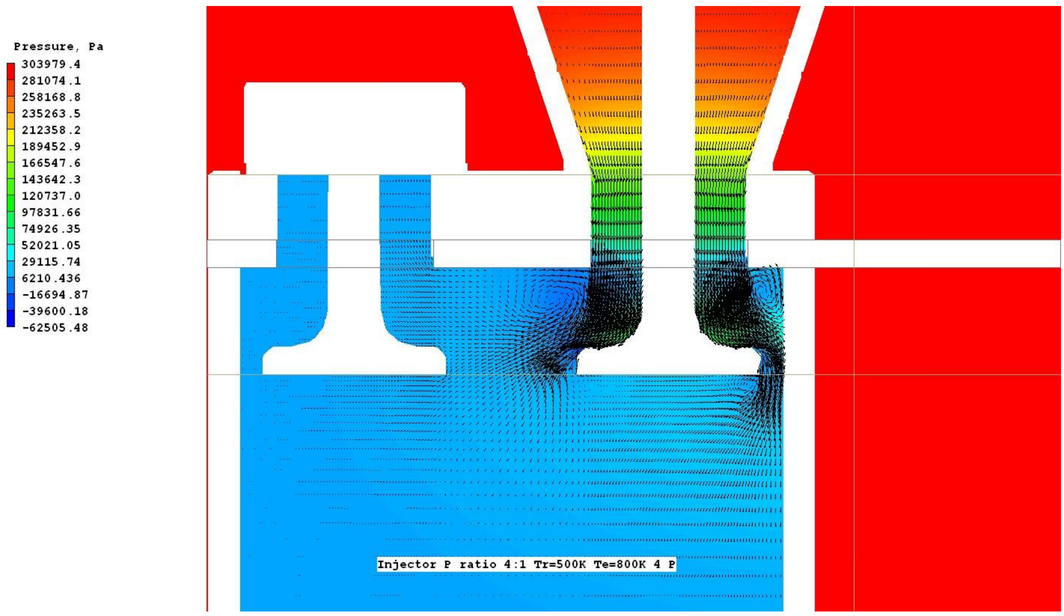


Velocity – Timestep 10

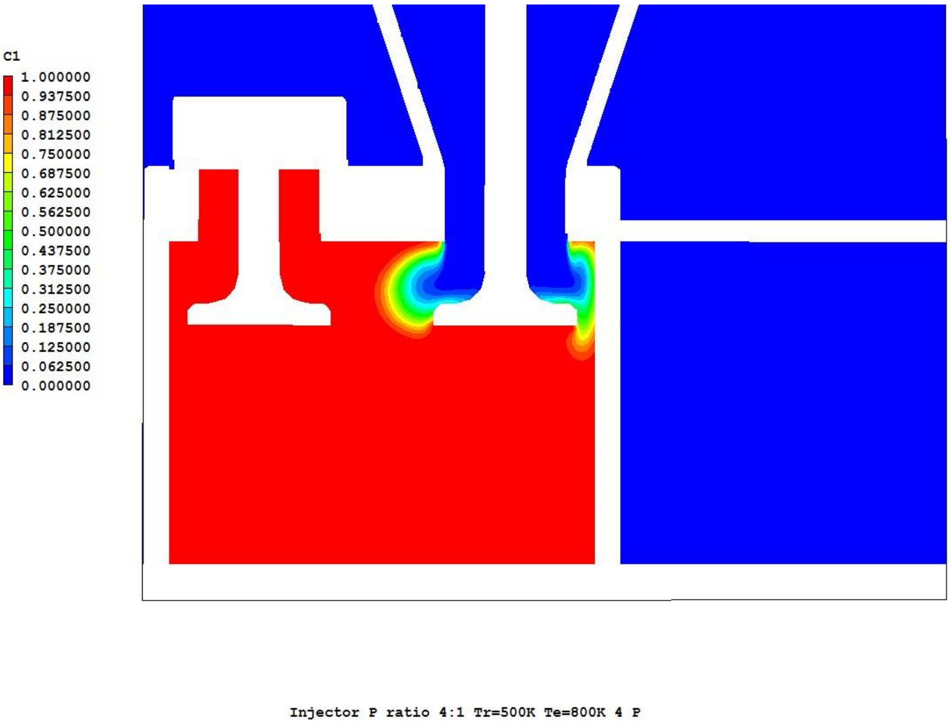
[Early in the process]



Temperature – Timestep 10

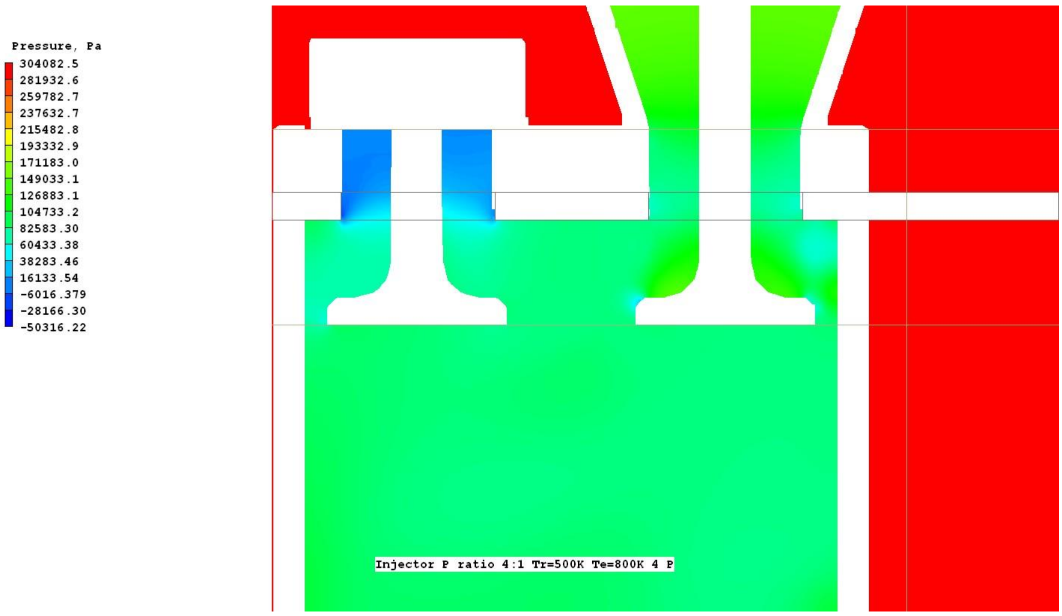


Pressure – Timestep 10 (+ velocity vectors)



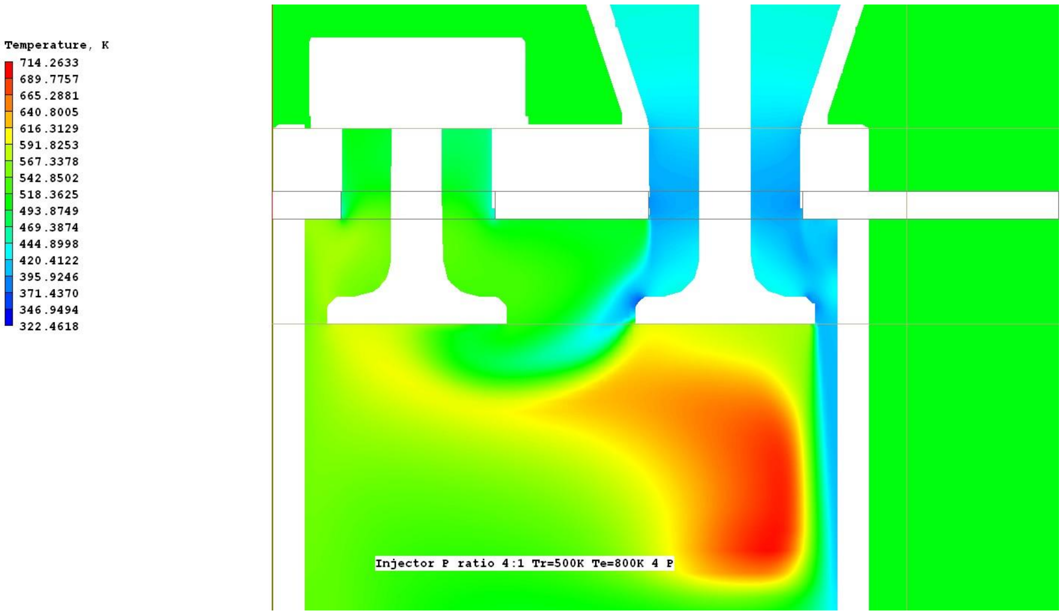
C1 (Hot gas marker) – Timestep 10

[Progression of cold gas flushing the hot]

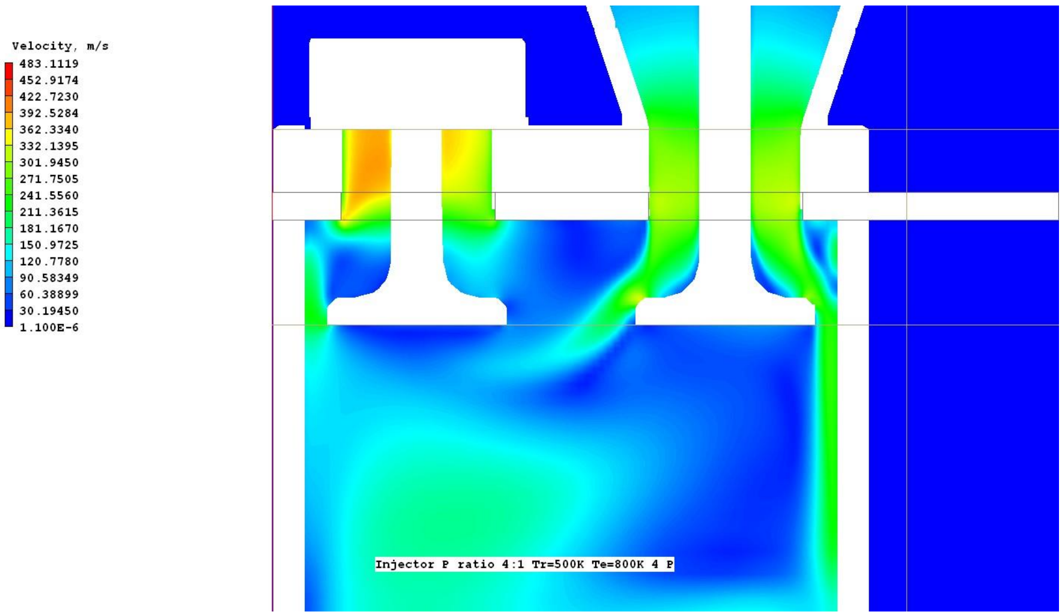


Pressure – Time step 220

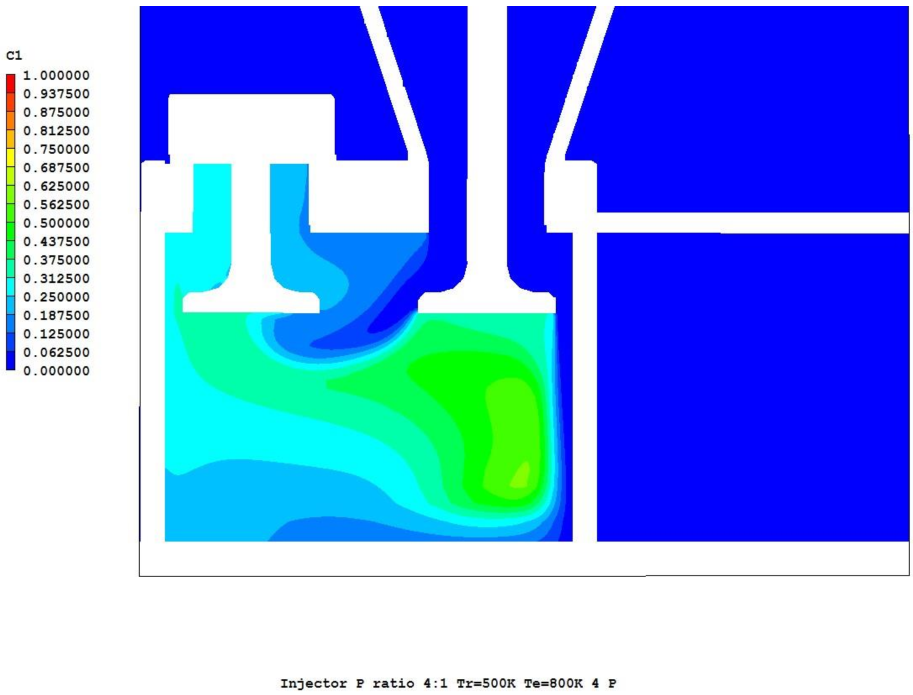
[Mid-way through the process]



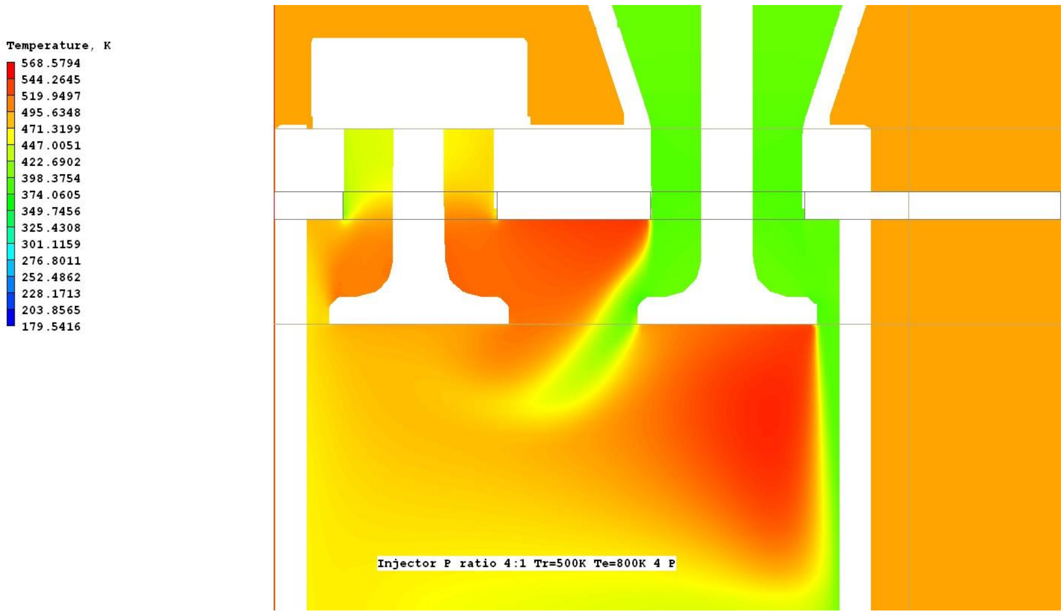
Temperature – Timestep 220



Velocity – Timestep 220

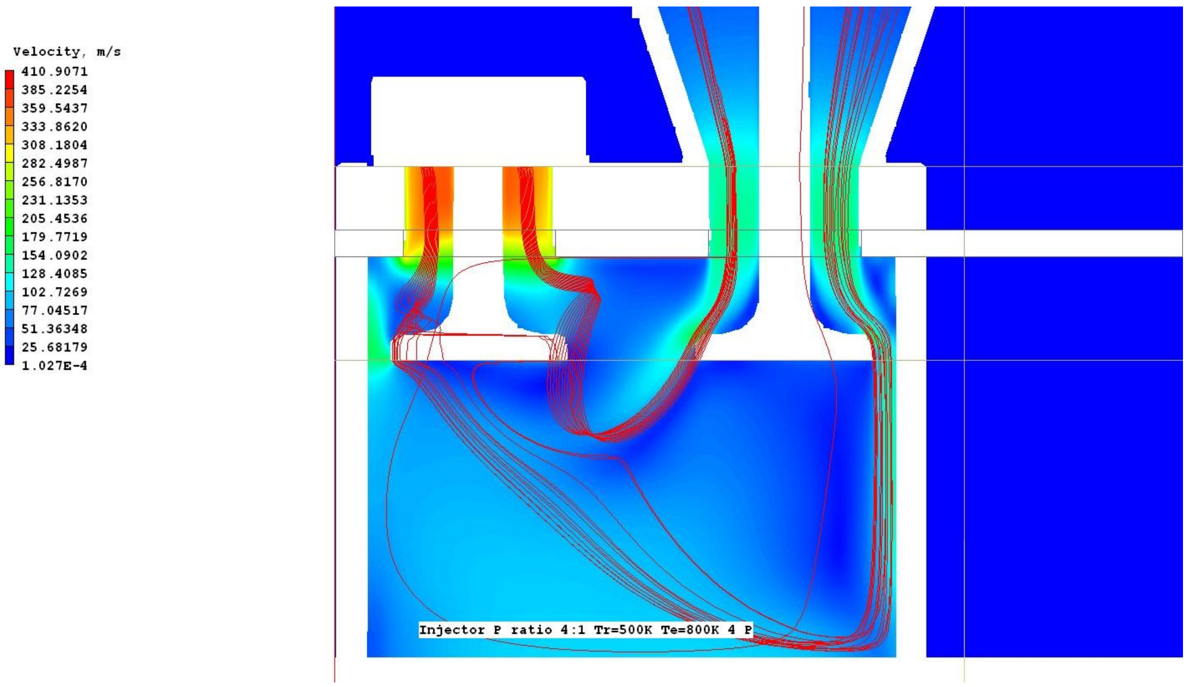


C1 (Hot gas marker) – Time step 220

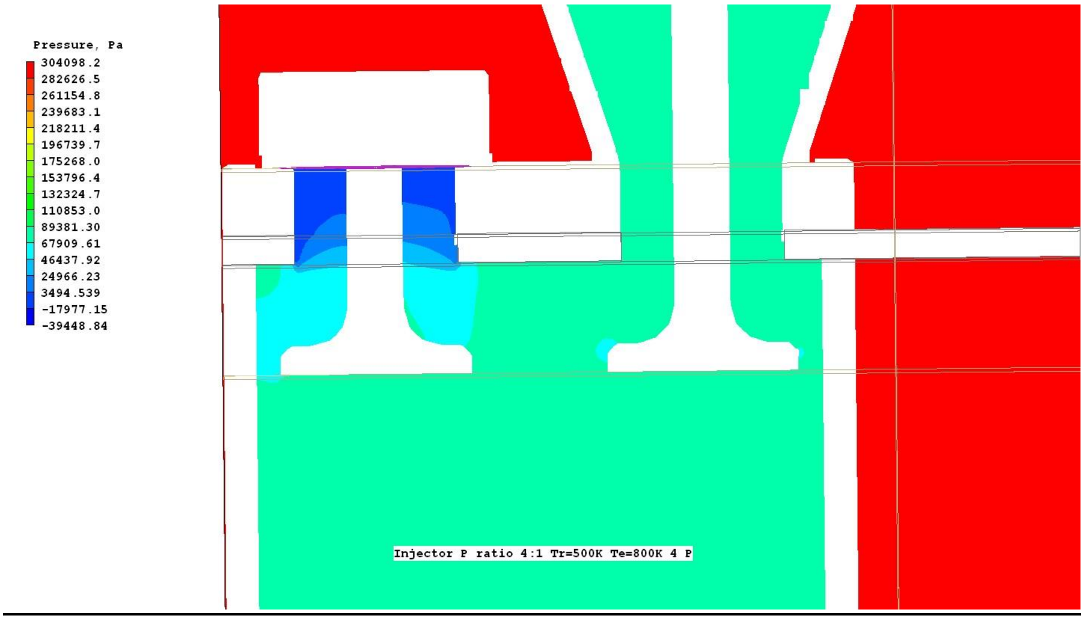


Temperature – Time step 400

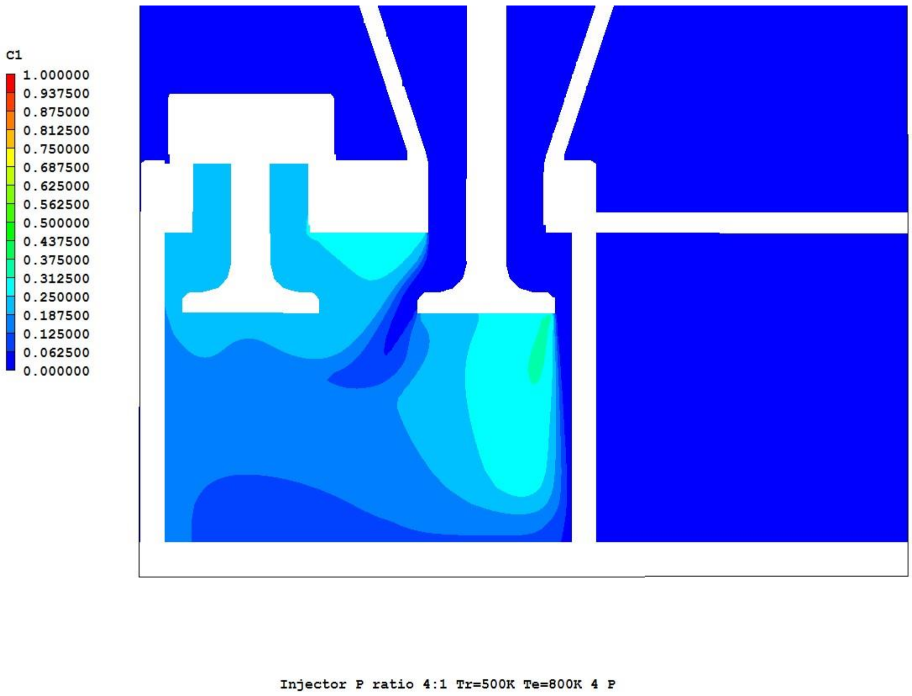
[Near-end of the process]



Velocity – Time step 400 (+ streamlines)



Pressure – Time step 400



C1 (Hot gas marker) – Time step 345 (T400 plot unavailable)

Note: Whilst the scale used in the animated results (see below) remains constant, the scales used in some of the images shown above do vary.

Conclusion

Complex geometry and boundary conditions, involving both high pressure, temperature and velocity gradients, are characterized in this example. It has been demonstrated that PHOENICS can adequately capture the instantaneous removal of the 'idealized' separating membrane and the subsequent gas-mixing and exhaust process.

Diaphragm rupture is a phenomenon that attracts a high interest in the scientific world as it is the mean feature characterizing shock tubes. These are widely employed when studying gas-phase combustion reactions or problems involving values of pressure and temperature that are not easily reproduced in a test rig. The main challenge for CFD codes in modelling this class of problems is the fast propagation of waves through the low-pressure zones.

Optimized results for this case could be obtained by localized refinement of the mesh where high- pressure ratios are present, adjustment of the time step, or greater attention to the relaxations factors. Such optimization can lower the computational cost while achieving better accuracy.

Contact: contact@arcofluidconsulting.com