Gas Jet Simulations

Paolo Magagnin and Ray Veness

Contents

Background and Objectives

- Considerable simulation work has been done for gas jet dynamics, most recently at Cockcroft*
- Two main reasons for continuing this work:
  - Need a design tool for optimising this instrument
    - Optimal spacing, diameter and geometry of nozzle skimmers
    - Pressure and hence pumping requirements from the vacuum system
    - Input for alignment precision of skimmers
  - Previous studies have focussed on the 'high pressure' end of the system
    - Tools and expertise are available at CERN to analyse the low pressure (molecular flow) regime
    - Could provide input for pumping system design, gas jet shaper and beam-gas interaction region

*M.Putignano, C.P. Welsch, Nuc. Inst. Methods A, 667 (2012) 44-52

Simulation issues

- Pressure range spans 11 orders of magnitude
  - Gas nozzle inlet at 10 Bar, Interaction chamber at ~10^{-7} mbar
  - Transition from viscous to molecular flow regimes mean the same physical models cannot be used over the whole flow
- Geometric details also range over 4 to 5 orders of magnitude
  - Nozzles from ~30 µm with transport over ~ 1 m
  - Tends to require numerical models with large numbers of elements, so computationally demanding

Simulation Strategy

- Separate the simulation by the 2 physics models
  - Preliminary analytical calculations and literature predict that the mean pressure in the volume after the first skimmer is ~10^{-5} mbar, so already molecular flow
  - Simple gas flow analysis also suggest that the volumes after the first skimmer can be pumped together
- High pressure (viscous flow) regime
  - Using Computational Fluid Dynamics Finite Element (CFD-FE) code (ANSYS-CFX) available at CERN
  - Simulations made by Paolo Magagnin
- Low pressure (molecular flow) regime
  - Using the MoFlow code
  - Developed (and used) by Roberto Kersevan

CFD Model (Paolo Magagnin)

- 30 µm diameter 'nozzle' with simple rectilinear geometry
  - 180 µm diameter 'Skimmer 1' added for later models
- 10 Bar of N_2 at 20 C expands into a volume with a pressure boundary condition
  - Pressure condition varied down to 0.88 Pa which is the pressure measured in the Cockcroft setup (limits to covergance)
- 'Simple' axisymmetric model
  - ~78'000 elements
  - Steady-state flow (ie, not directly simulating the pulsed nozzle operating at Cockcroft)

Benchmarking – theoretical Mach disc dimensions

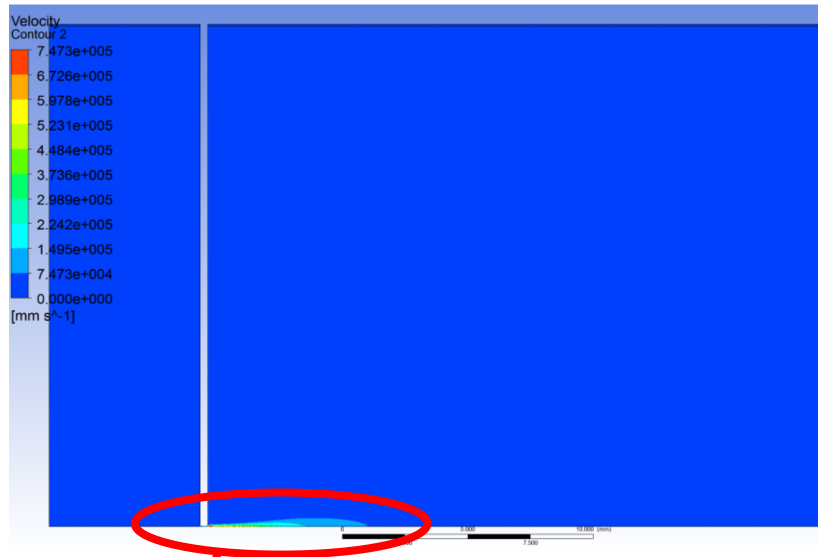
Nozzle diameter	Tank pressure	Nozzle chamber pressure	Mach disc position	Mach disc diameter	Barrel shock diameter
d_n [mm]	p_t [bar]	p_nc [bar]	x_M [mm]	d_M [mm]	d_Bs [mm]
3.00E-02	10	1.0E-01	0.20	0.10	0.15
		1.0E-02	0.64	0.32	0.48
		1.0E-03	2.01	1.01	1.51
		1.0E-04	6.36	3.18	4.77
		1.0E-05	20.10	10.05	15.08
		8.80E-06	21.43	10.71	16.07

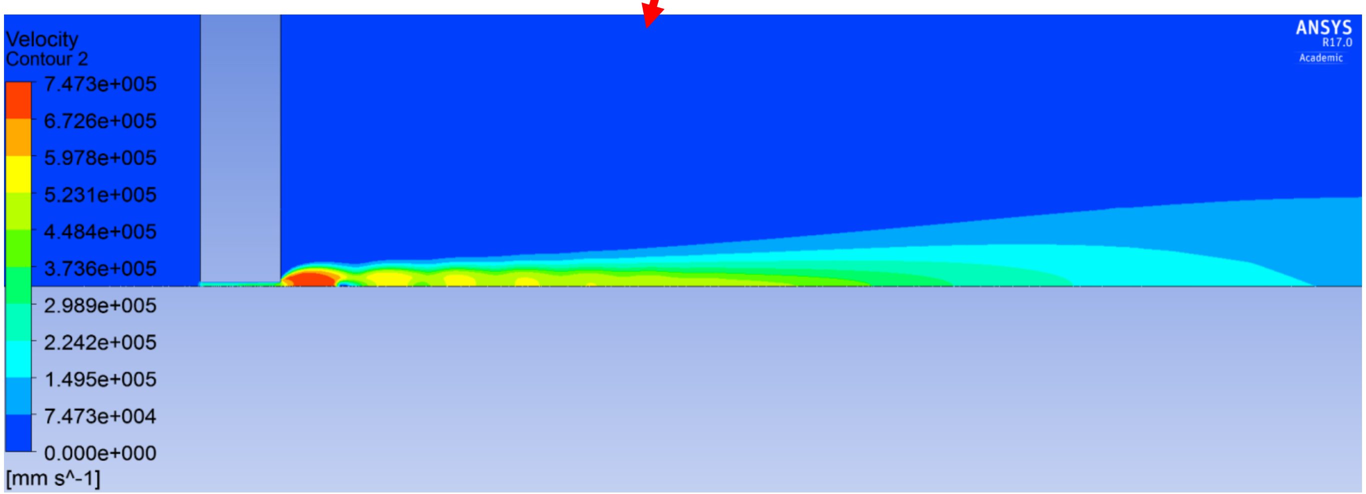
There are rather simple analytical solutions for the Mach disc position (transition from supersonic to subsonic flow), Mach disc diameter and barrel shock diameter (the transverse dimension of the super- to sub-sonic transition) as a function of inlet pressure and nozzle diameter given in [1].
These were used to benchmark the results from this model.

‘Supersonic Gas-Jet Based Beam Profile Monitor’. M. Putignano. PhD Thesis, University of Liverpool, 2012

CFD simulation with nozzle chamber pressure $p_{NC} = 0.1\ [bar]$

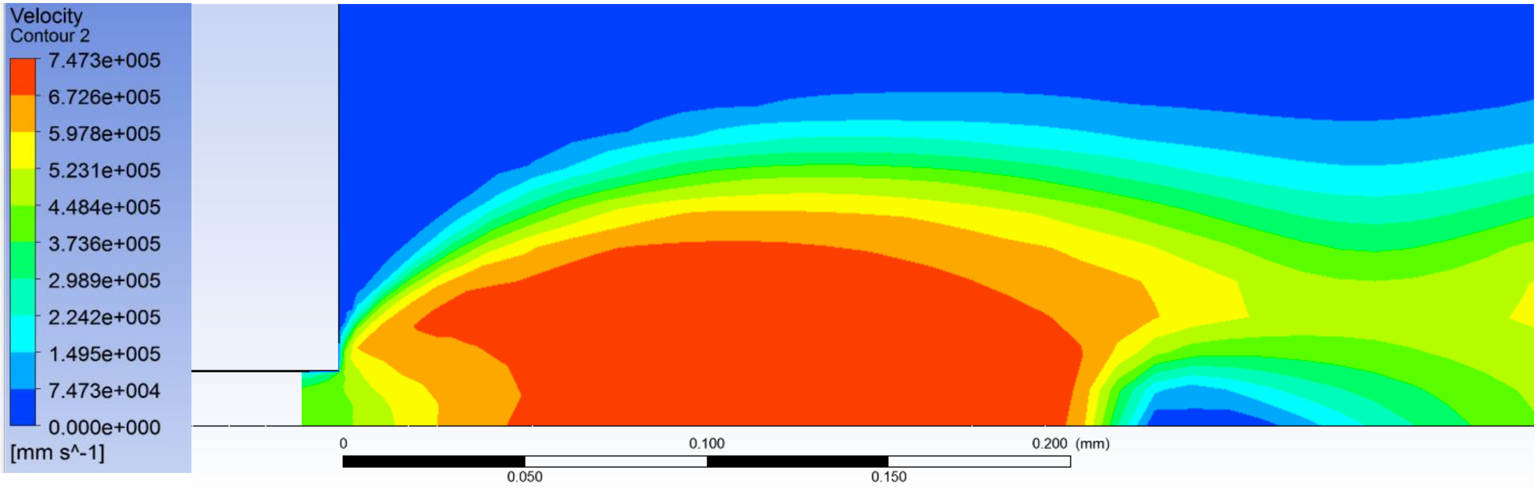
- Velocity distribution





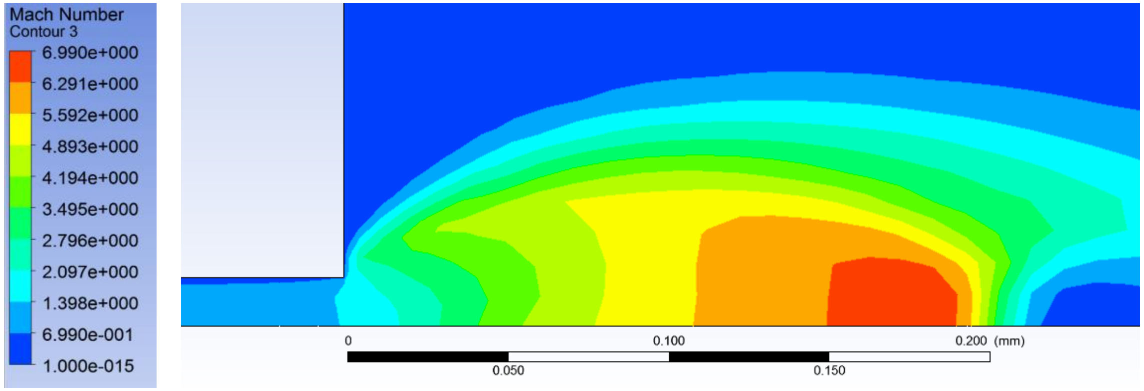
CFD simulation with nozzle chamber pressure $p_{NC} = 0.1\ [bar]$

- Validation: velocity distribution



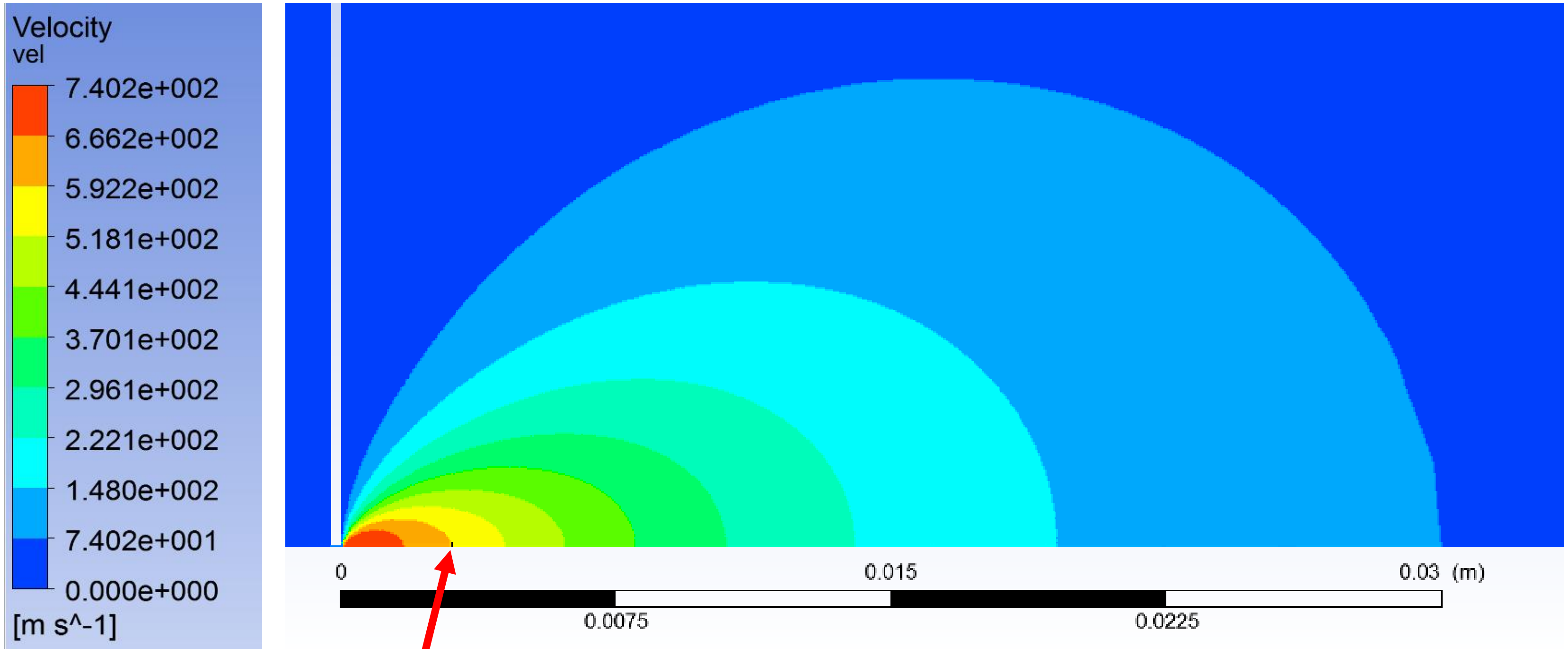
Nozzle chamber pressure	Mach disc position	Mach disc diameter	Barrel shock diameter
p_nc [bar]	x_M [mm]	d_M [mm]	d_Bs [mm]
1.0E-01	**0.20**	**0.10**	**0.15**
1.0E-02	0.64	0.32	0.48
1.0E-03	2.01	1.01	1.51
1.0E-04	6.36	3.18	4.77
1.0E-05	20.10	10.05	15.08
8.80E-06	21.43	10.71	16.07

- Mach number distribution



CFD simulation with nominal nozzle chamber pressure $p_{NC} = 0.88\ [Pa]$

- Validation: velocity distribution



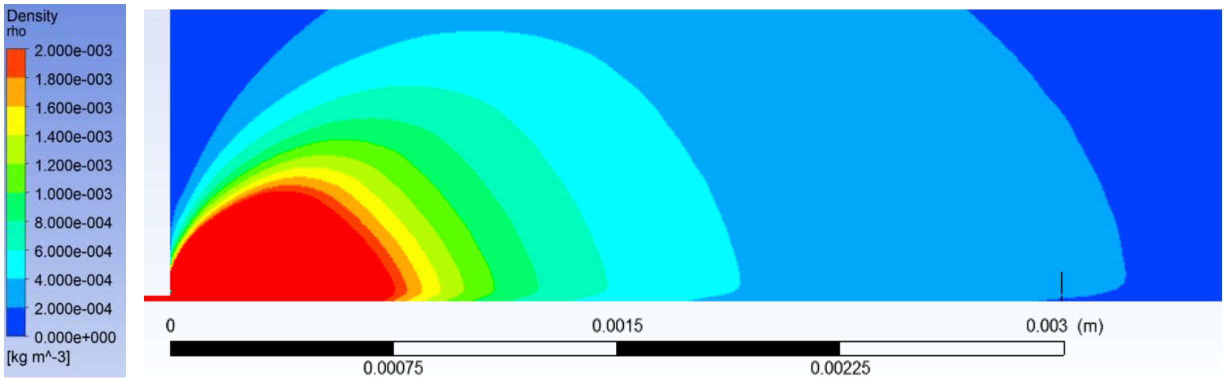
1st skimmer entrance section

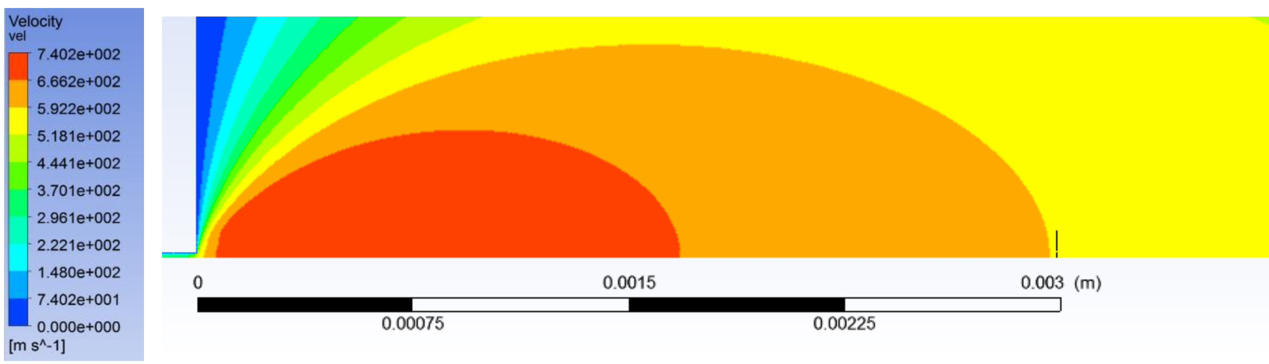
Nozzle chamber pressure	Mach disc position	Mach disc diameter	Barrel shock diameter
p_nc [bar]	x_M [mm]	d_M [mm]	d_Bs [mm]
1.0E-01	0.20	0.10	0.15
1.0E-02	0.64	0.32	0.48
1.0E-03	2.01	1.01	1.51
1.0E-04	6.36	3.18	4.77
1.0E-05	20.10	10.05	15.08
8.80E-06	**21.43**	**10.71**	**16.07**

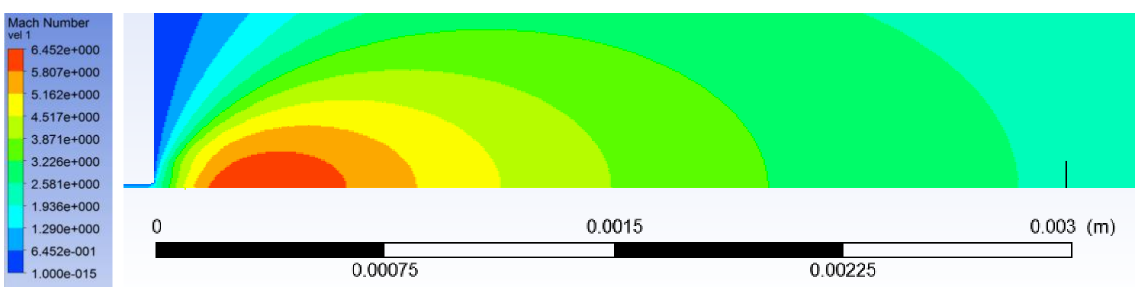
Benchmark for current Cockcroft prototype operating conditions

CFD simulation with nominal nozzle chamber pressure $p_{NC} = 0.88\ [Pa]$

- Density vs velocity distribution vs Mach number.





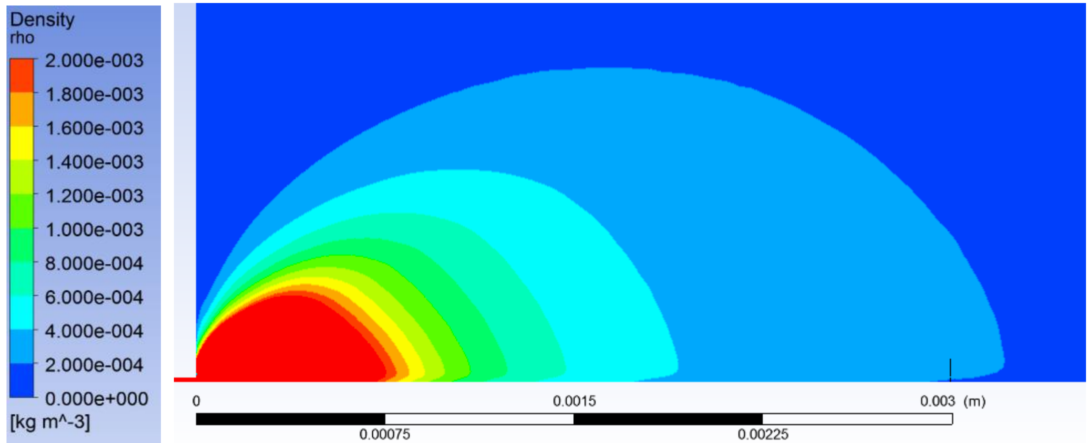


Gas jet at first skimmer location is already quite broad – alignment less critical?

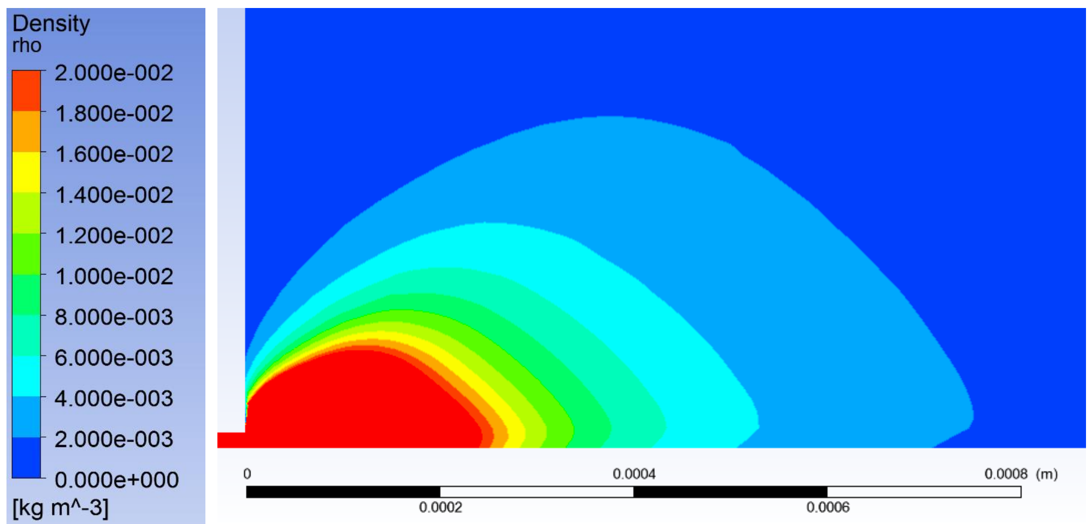
CFD simulation with nominal nozzle chamber pressure $p_{NC} = 0.88\ [Pa]$

- Density distribution [kg/m3]: zoom in the high density region.

- $\rho_{Max} = 2 \cdot 10^{-3}\ \frac{kg}{m^3}$



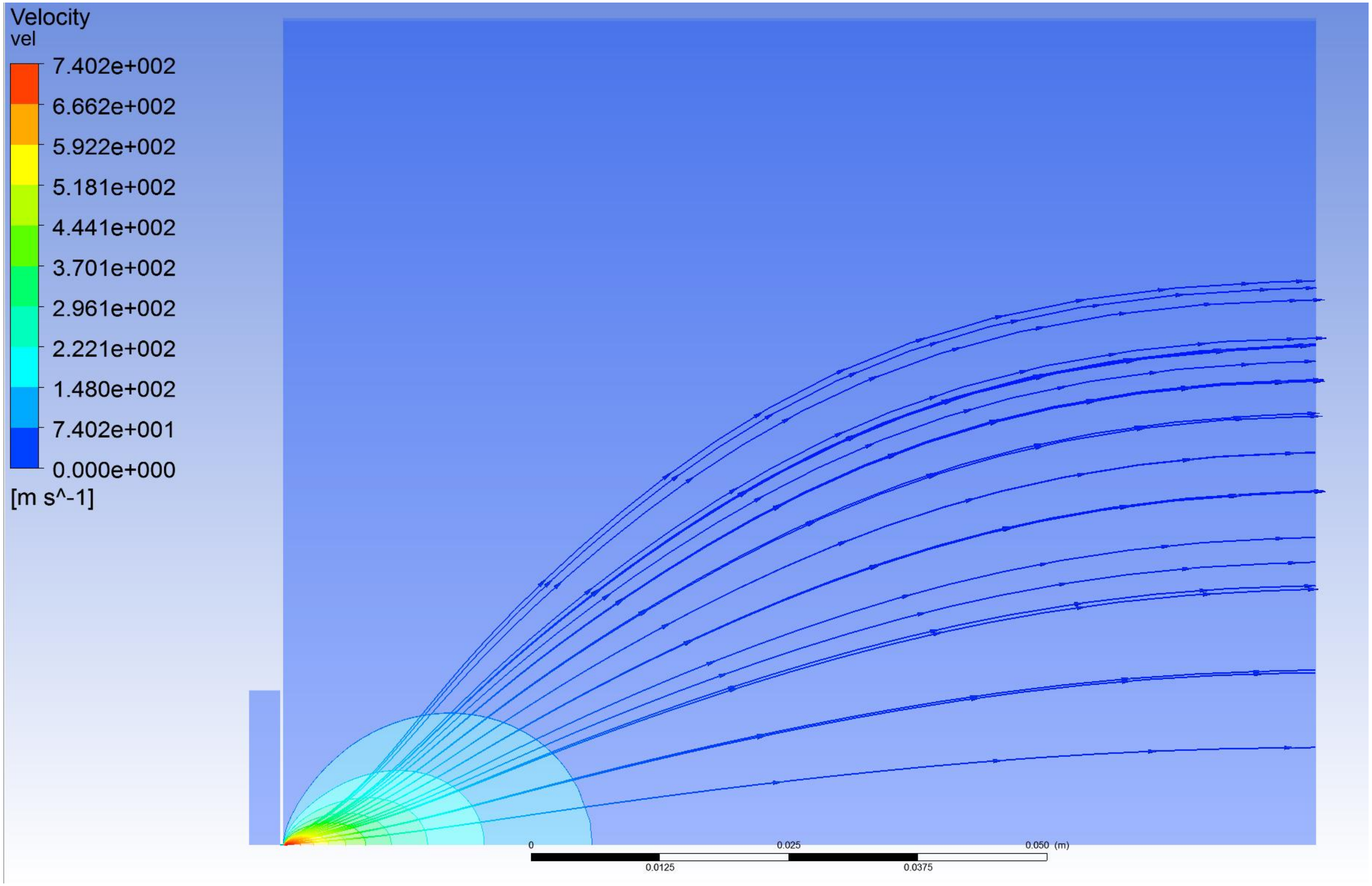
- $\rho_{Max} = 2 \cdot 10^{-2}\ \frac{kg}{m^3}$



Input for mass flow at the nozzle location

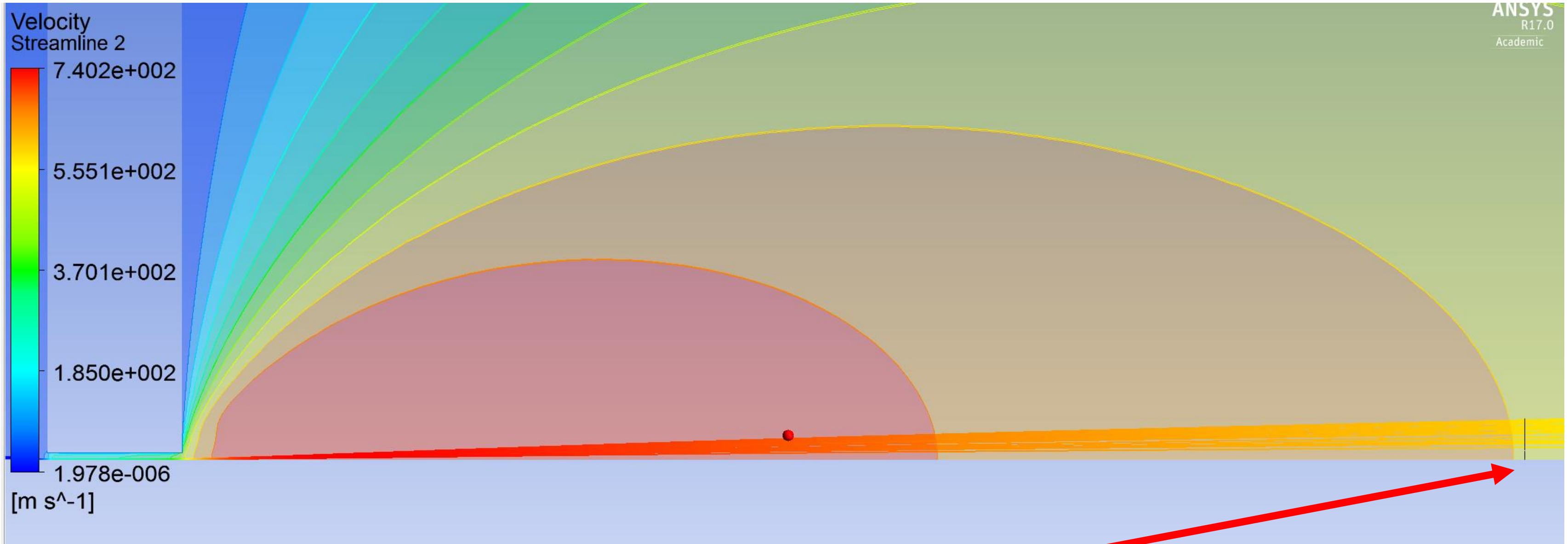
CFD simulation with nominal nozzle chamber pressure $p_{NC} = 0.88\ [Pa]$

- Flow direction

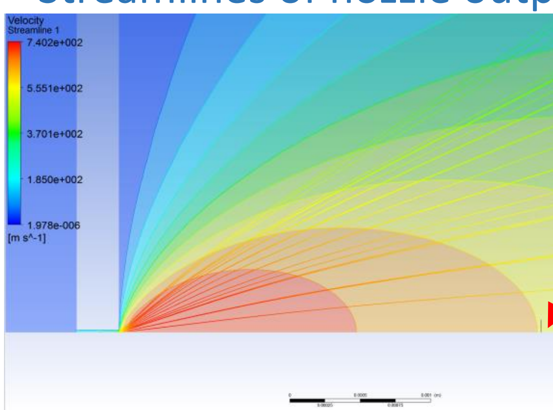


CFD simulation with nominal nozzle chamber pressure $p_{NC} = 0.88\ [Pa]$

- Streamlines of 1st skimmer input

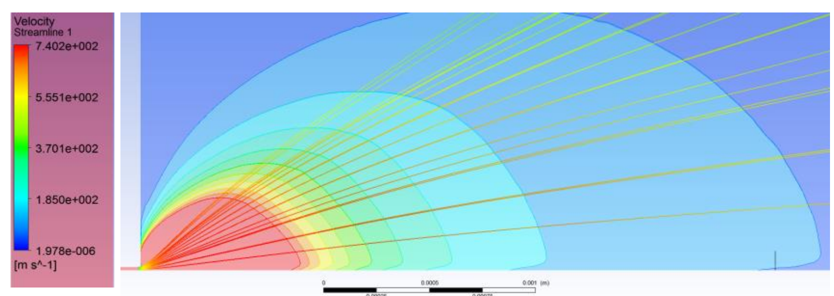


- Streamlines of nozzle output



1st skimmer entrance section

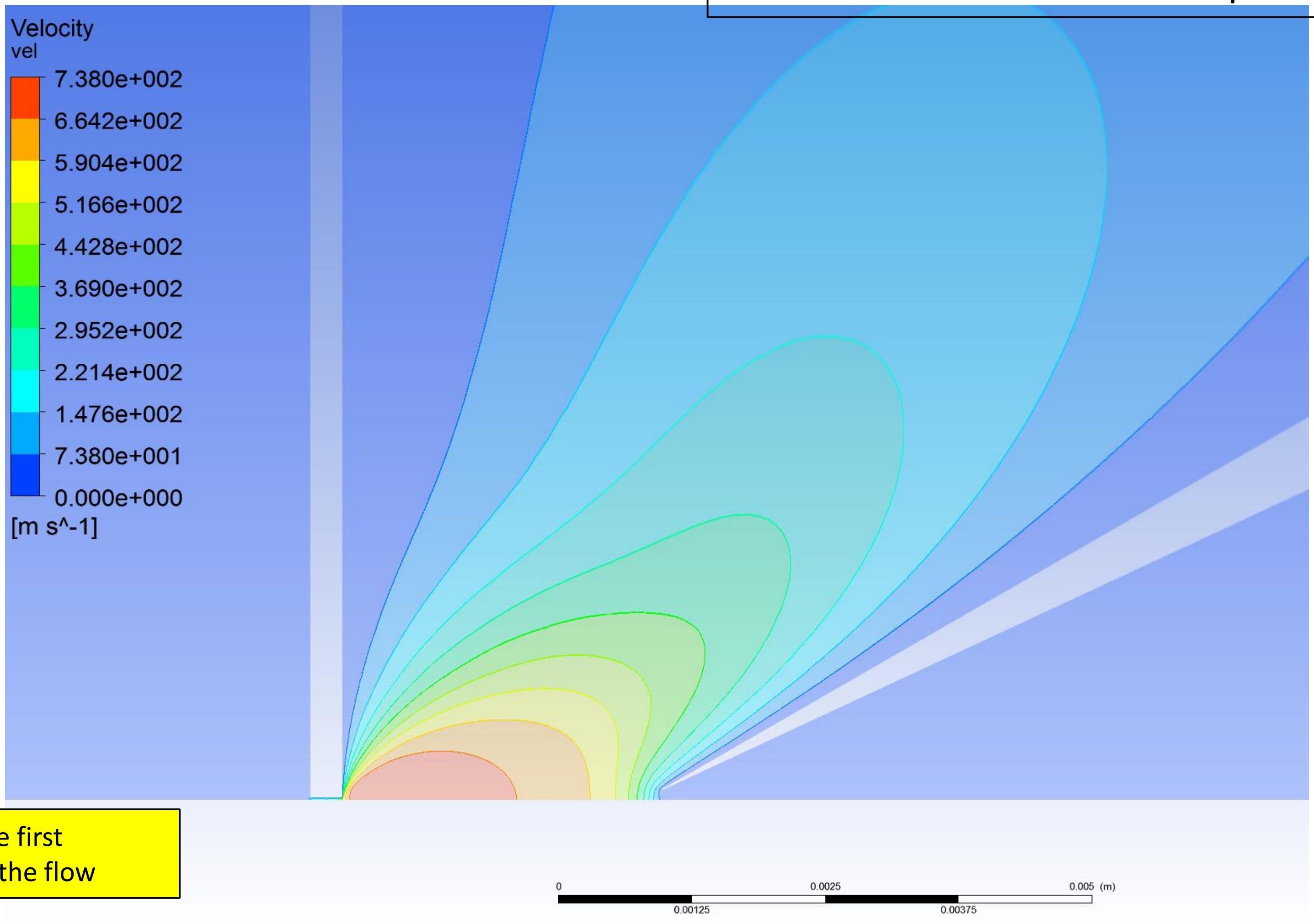
- Streamlines (velocity [m/s]) vs density [kg/m3]



CFD simulation with 1st skimmer and $p_{NC} = 0.88\ [Pa]$ everywhere

- Velocity distribution

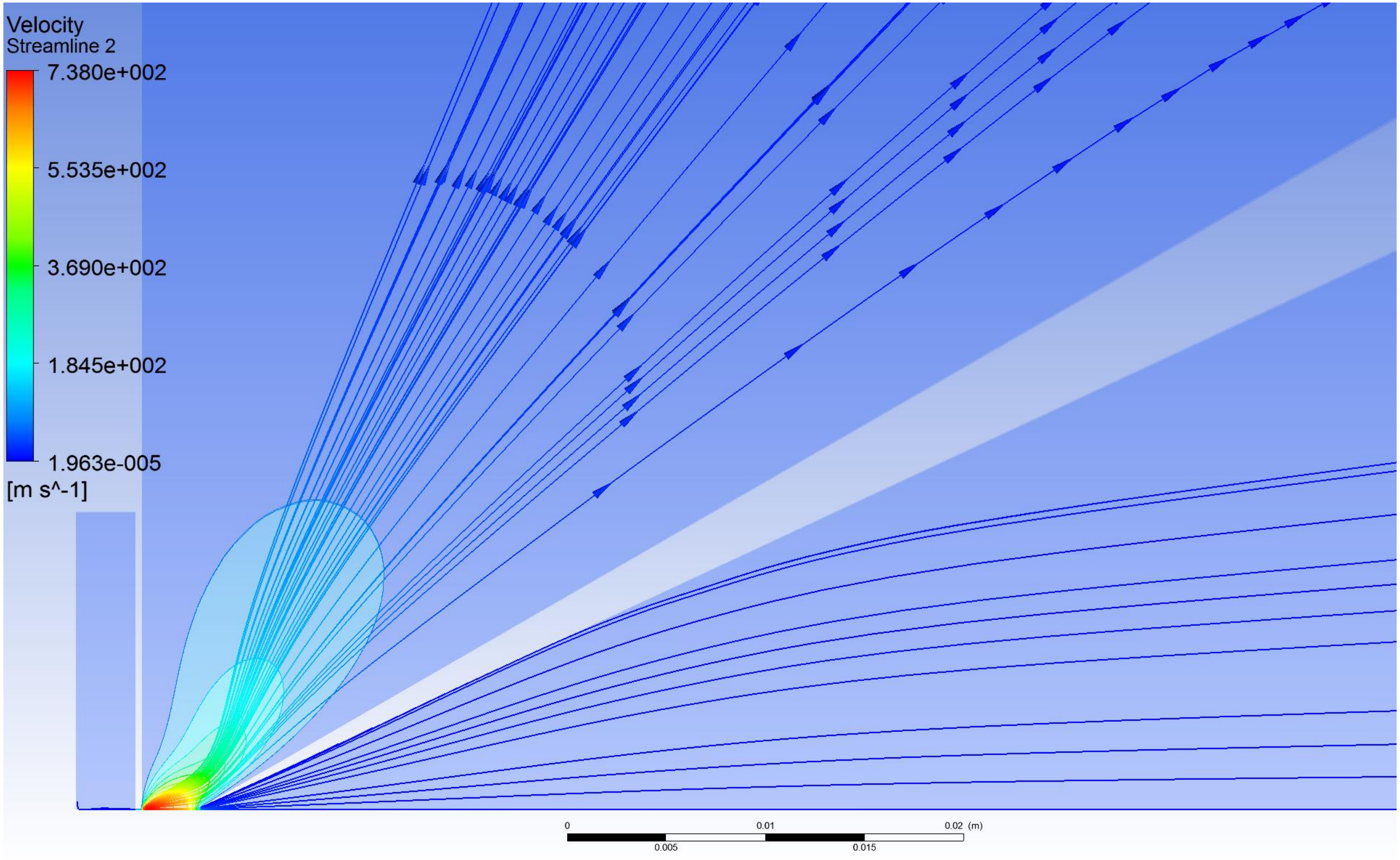
Note: The background pressure on the far side of skimmer is also 0.88 Pa – probably too high



Impact of the first skimmer on the flow

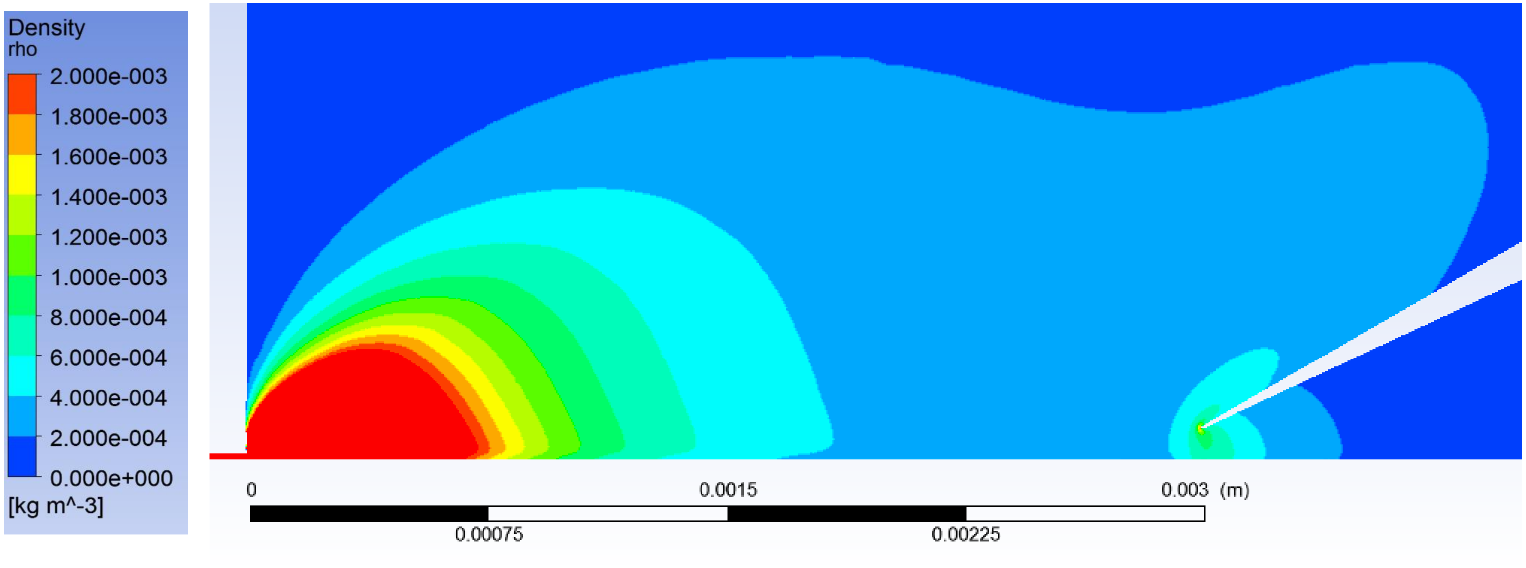
CFD simulation with 1st skimmer and $p_{NC} = 0.88\ [Pa]$ everywhere

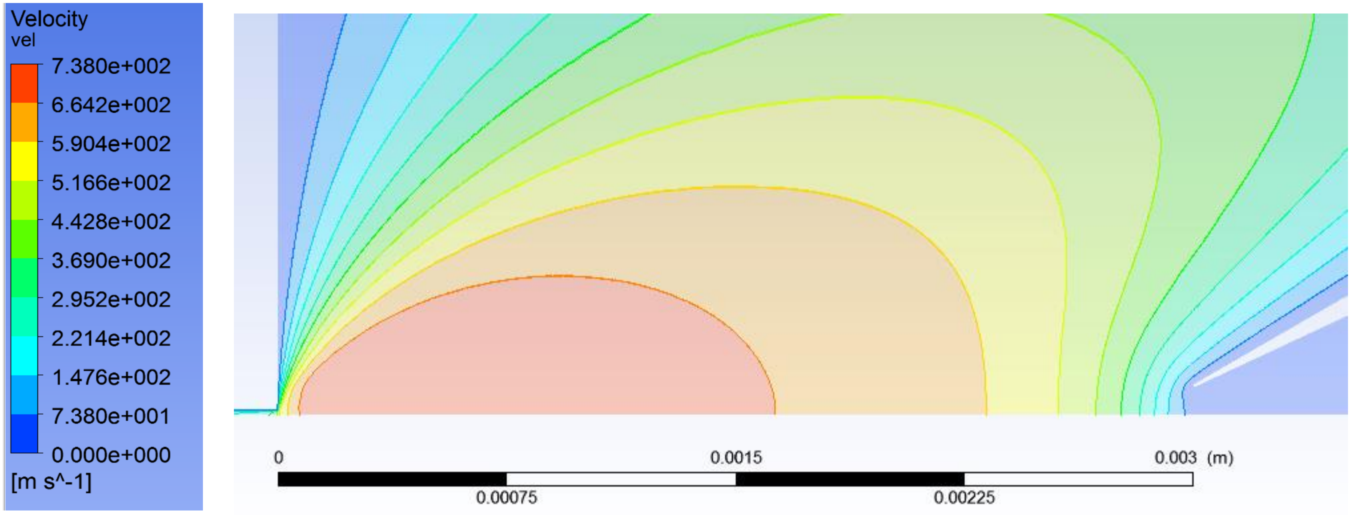
- Velocity distribution and streamlines



CFD simulation with 1st skimmer and $p_{NC} = 0.88\ [Pa]$ everywhere

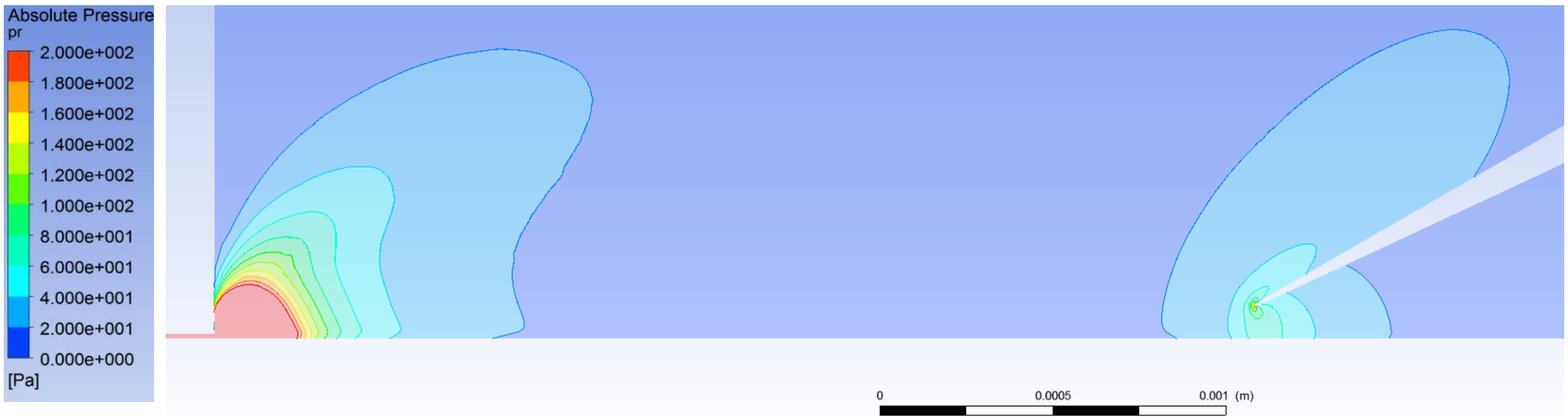
- Density vs velocity distribution. Attention: scale between 2E-3 kg/m3 and 0.

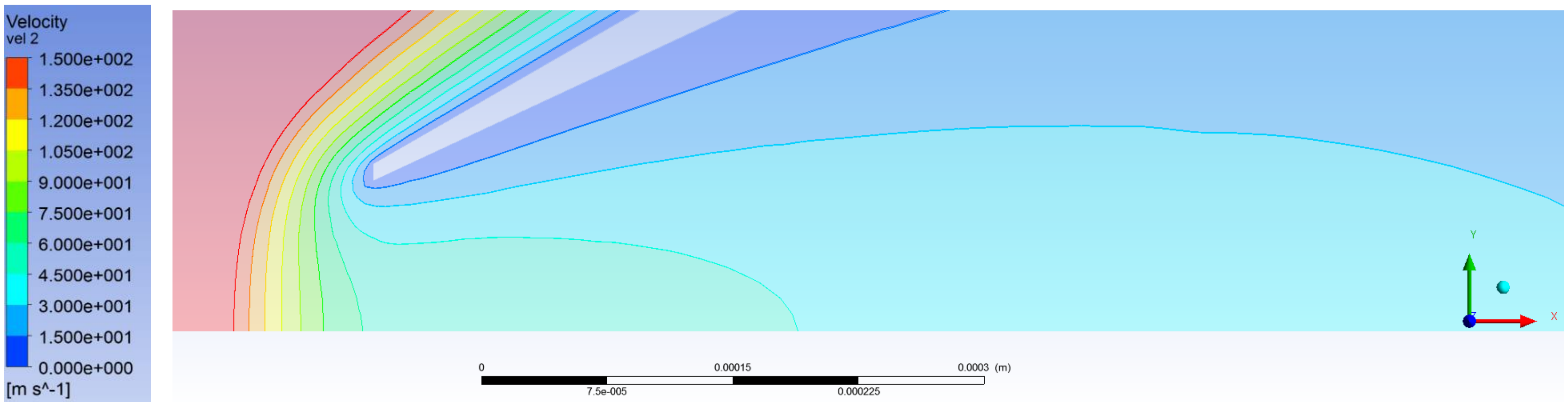




CFD simulation with 1st skimmer and $p_{NC} = 0.88\ [Pa]$ everywhere

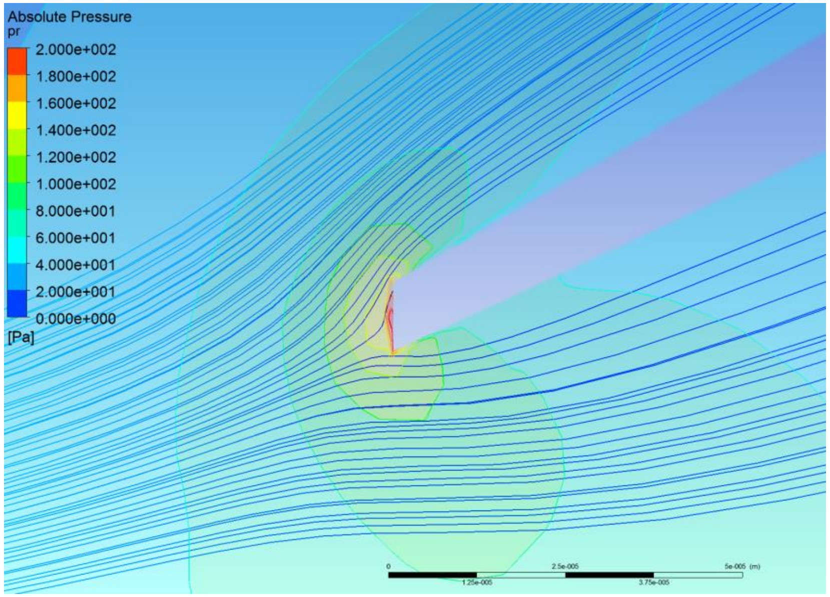
- Skimmer effect on the flow: generation of an overpressure in the skimmer entrance, which decelerate the flow to 50 m/s.



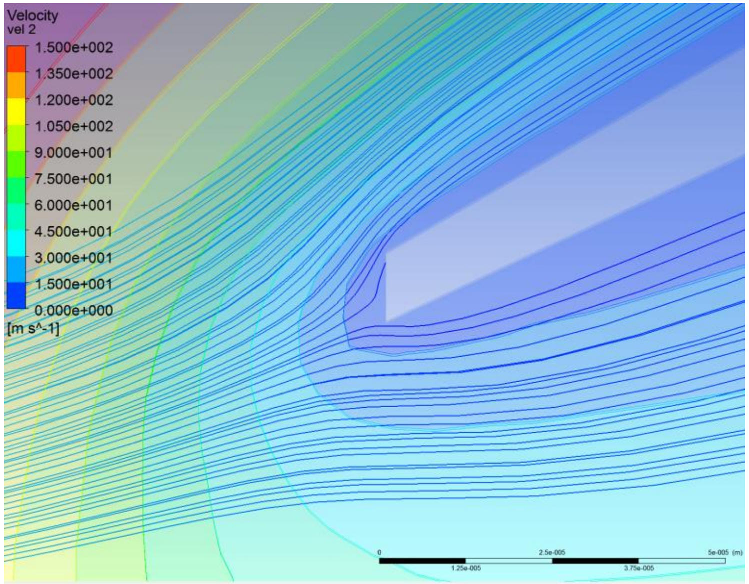


CFD simulation with 1st skimmer and $p_{NC} = 0.88\ [Pa]$ everywhere

- Streamlines and absolute pressure on background



- Streamlines and velocity on background



- Effect of skimmer's entrance thickness: the flow become more straight in proximity of the skimmer, but the overpressure generated decelerate the flow.
- The entrance thickness has to be optimised, choosing the best trade off between the generated effects and machining feasibility.

What have we achieved so far?

- Produced a CFD model that benchmarks well with analytical solutions from Putignano's thesis
  - Produced a data set that can be used as input for the MoFlow calculation
- Simulations with different pressure in the nozzle chamber
  - confirm that lower pressures extend the length of the supersonic region, where the first skimmer should be located
- The model shows that alignment in offset/angle of the first skimmer should not be critical in the range of ~ diameters
  - First to second skimmer alignment may be more critical
- Insertion of the first skimmer has a significant impact on the gas flow
  - Gas flow lines are re-directed, but velocity appears significantly retarded

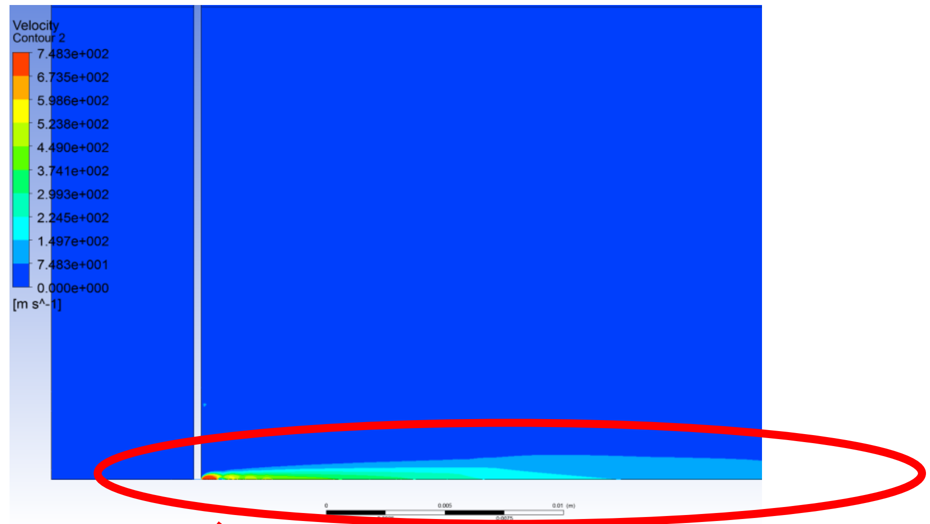
Possible issues and improvements

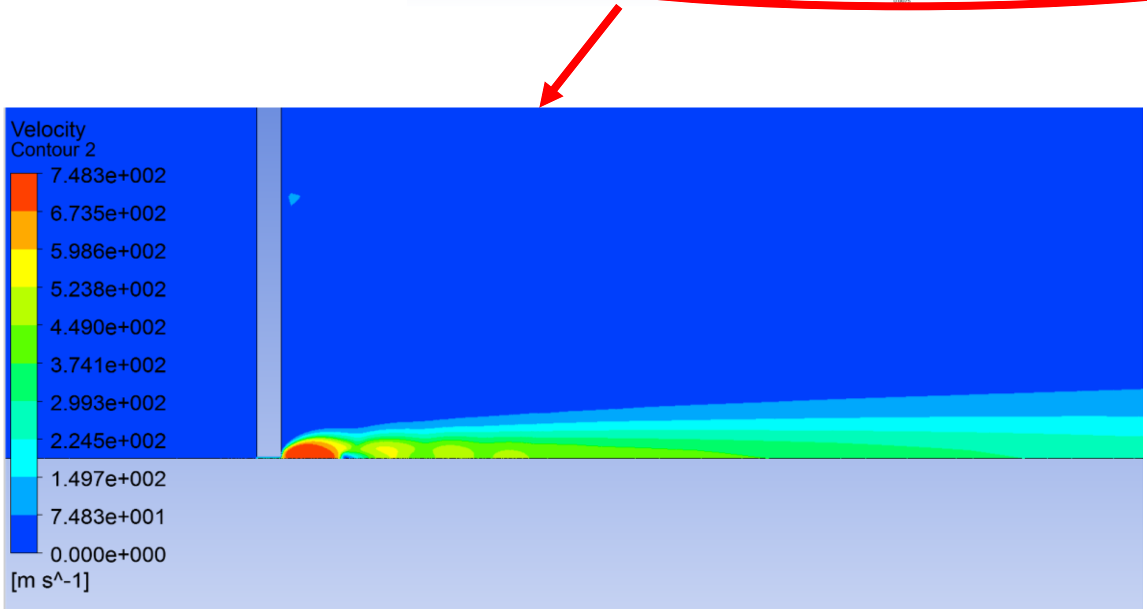
- Possible improvements to the model:
  - The Cockcroft prototype is operated in a 'pulsed' mode. Does the pressure in the nozzle have time to fully develop? Can this be modelled?
  - A 3D model would allow the pumping environment and skimmer alignment to be modelled. Is this justified?
- Possible improvements to the design:
  - Benefits from reducing the gas temperature to increase density? This is a solution used in gas jet targets
  - Details of the nozzle – would a micro 'de Laval' or other nozzle geometry help? This is a current research field for satellite propulsion systems
  - Details of the first skimmer are important – significant impact on gas velocity and direction
- CERN are now missing personnel to do this work…

Additional Material

CFD simulation with nozzle chamber pressure $p_{NC} = 0.01\ [bar]$

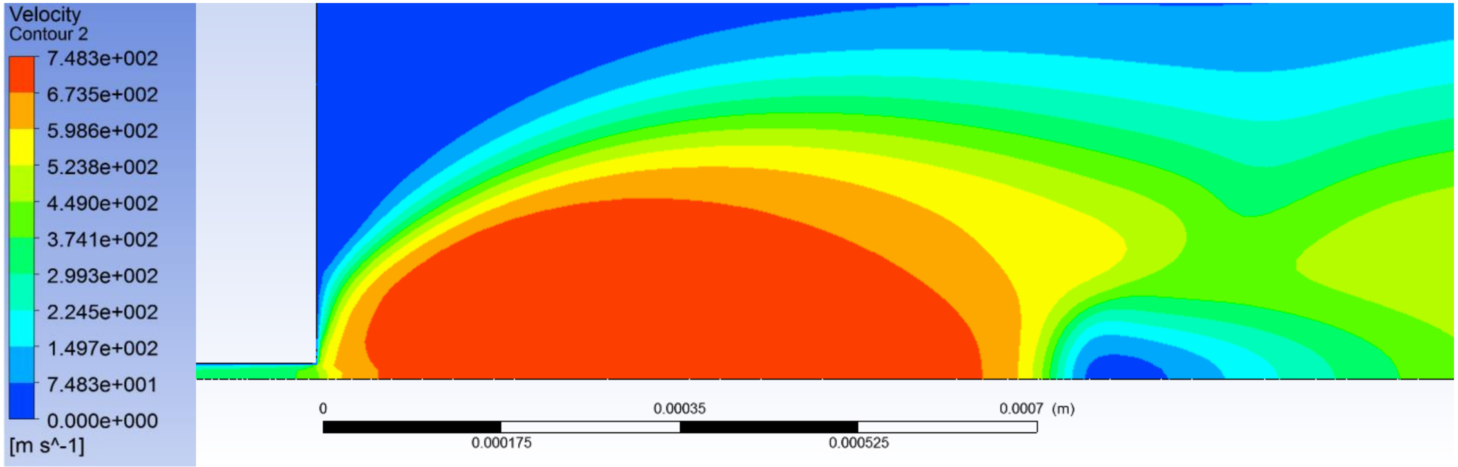
- Velocity distribution





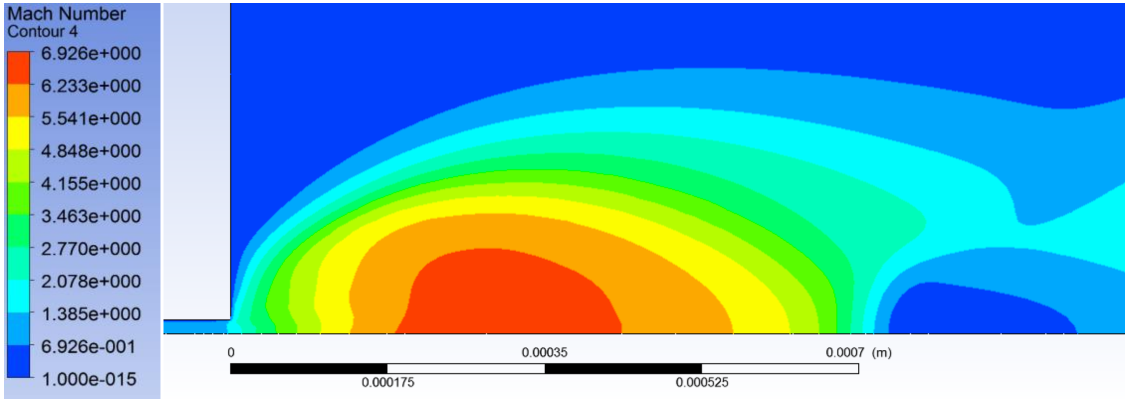
CFD simulation with nozzle chamber pressure $p_{NC} = 0.01\ [bar]$

- Validation: velocity distribution



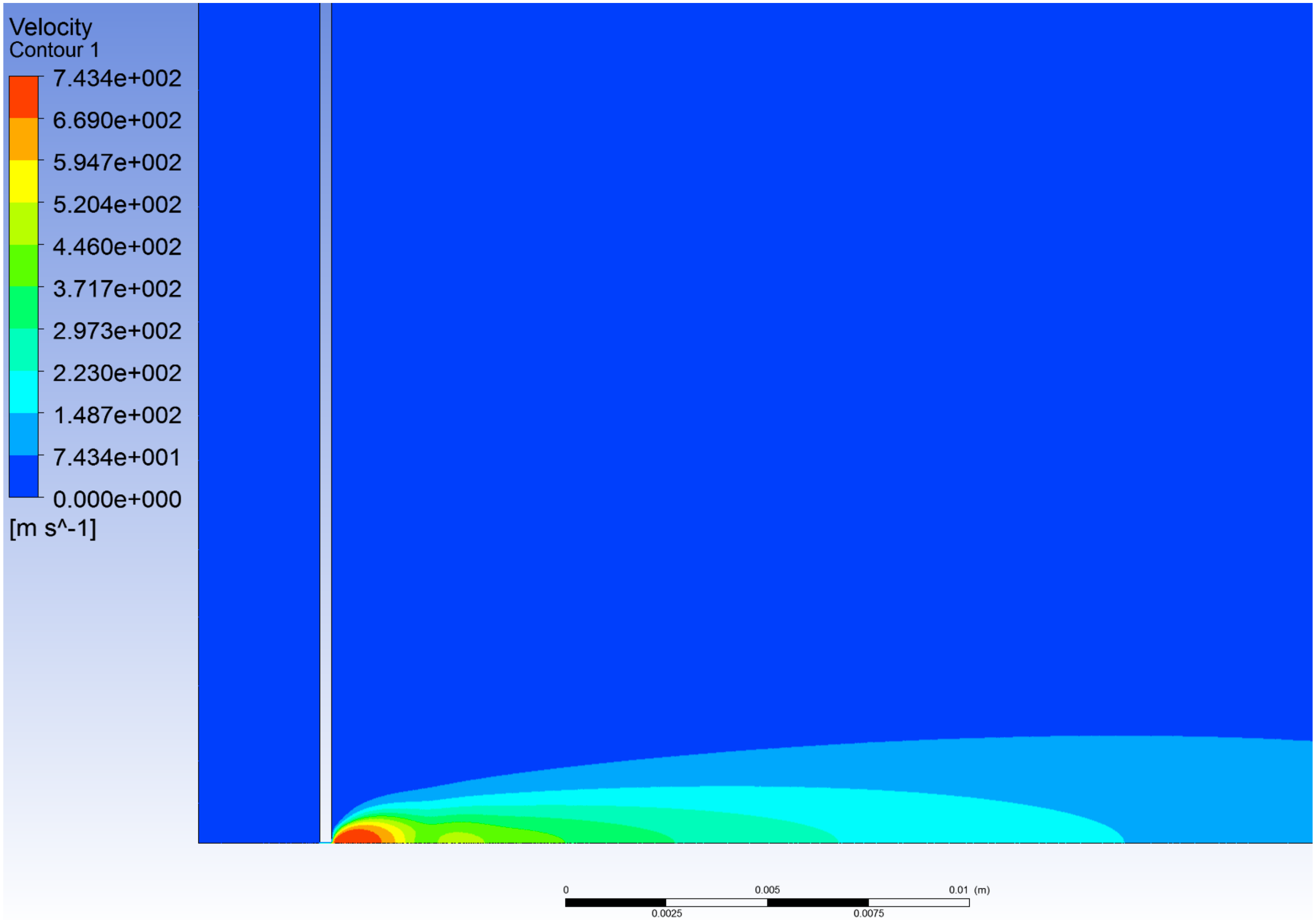
Nozzle chamber pressure	Mach disc position	Mach disc diameter	Barrel shock diameter
p_nc [bar]	x_M [mm]	d_M [mm]	d_Bs [mm]
1.0E-01	0.20	0.10	0.15
1.0E-02	**0.64**	**0.32**	**0.48**
1.0E-03	2.01	1.01	1.51
1.0E-04	6.36	3.18	4.77
1.0E-05	20.10	10.05	15.08
8.80E-06	21.43	10.71	16.07

- Mach number distribution



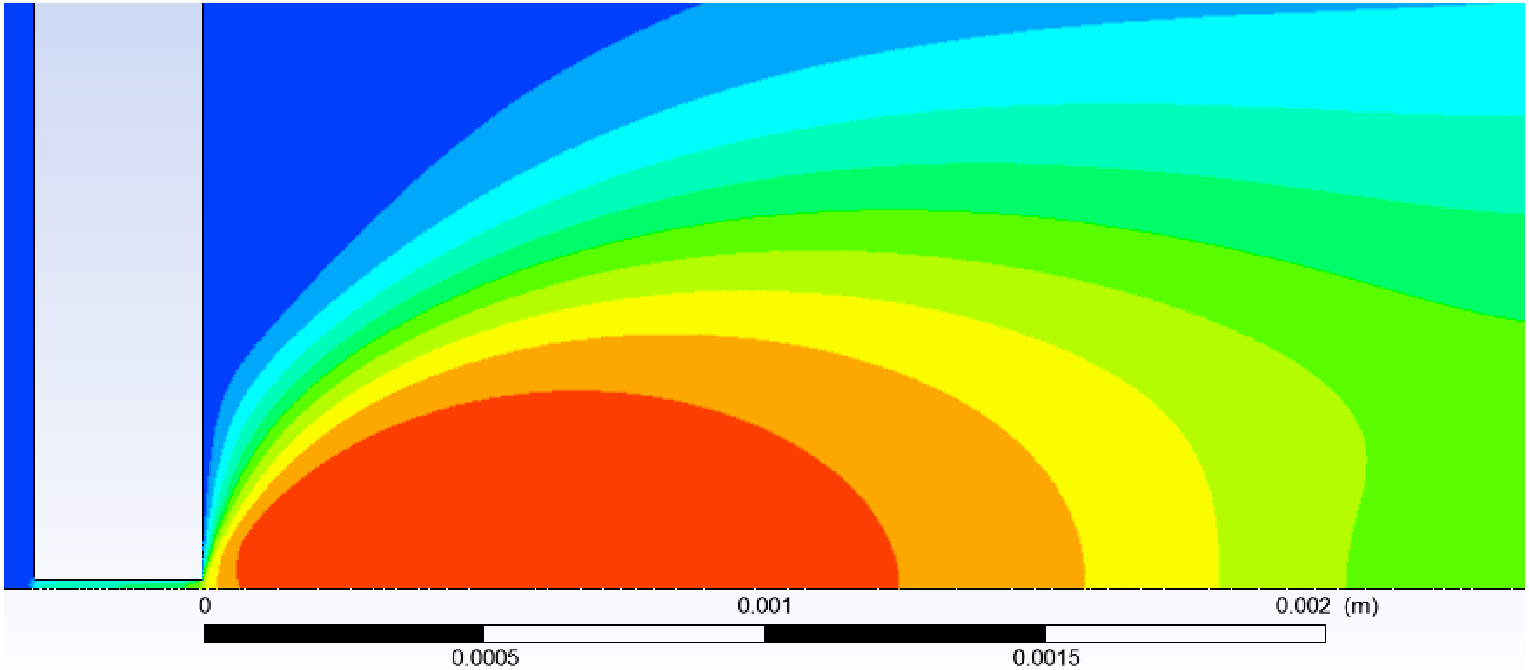
CFD simulation with nozzle chamber pressure $p_{NC} = 1\ [mbar]$

- Velocity distribution



CFD simulation with nozzle chamber pressure $p_{NC} = 1\ [mbar]$

- Validation: velocity distribution

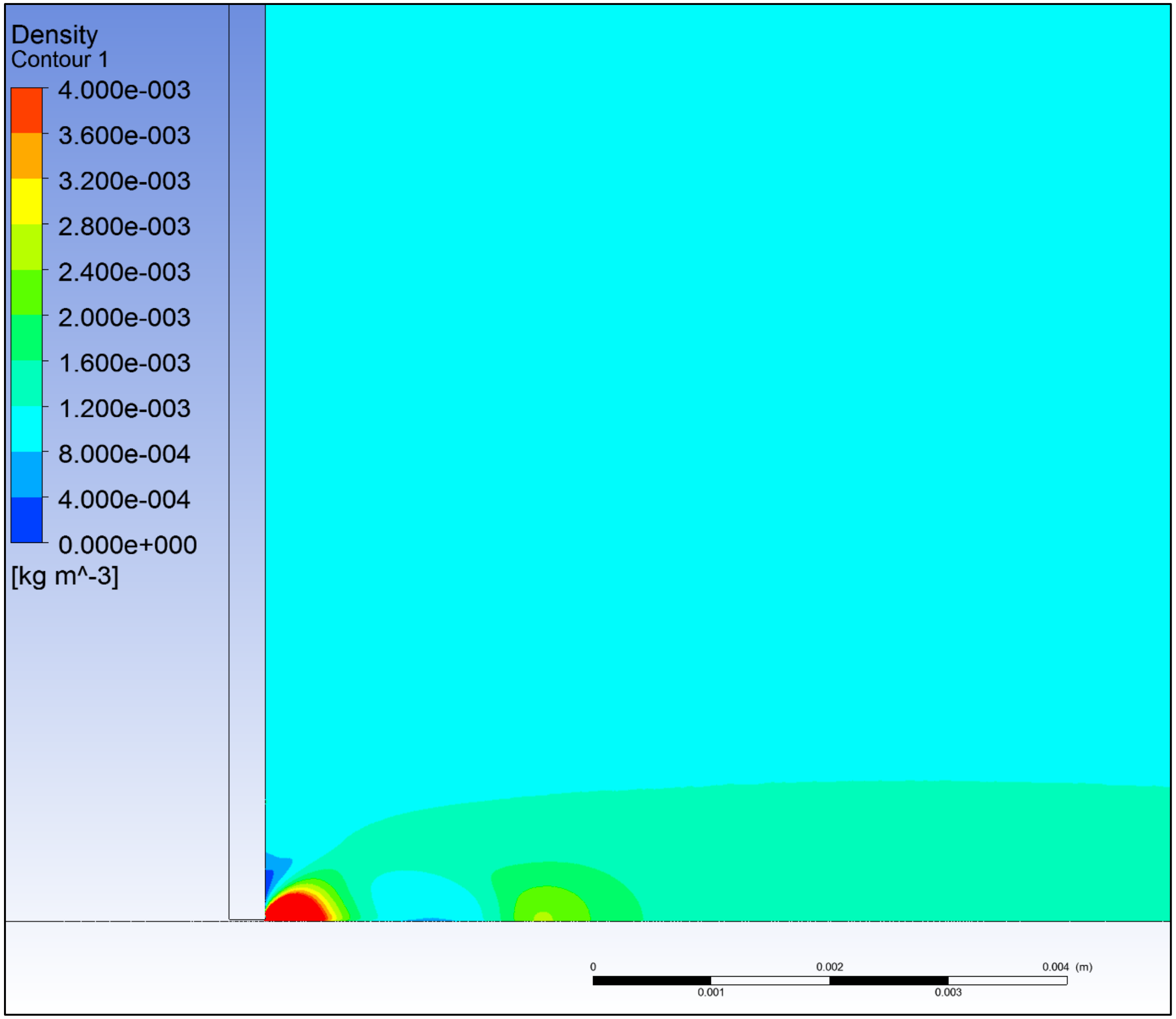


Nozzle chamber pressure	Mach disc position	Mach disc diameter	Barrel shock diameter
p_nc [bar]	x_M [mm]	d_M [mm]	d_Bs [mm]
1.0E-01	0.20	0.10	0.15
1.0E-02	0.64	0.32	0.48
1.0E-03	**2.01**	**1.01**	**1.51**
1.0E-04	6.36	3.18	4.77
1.0E-05	20.10	10.05	15.08
8.80E-06	21.43	10.71	16.07

At low pressure the shock become less abrupt and pressure change occurs over a longer distance.

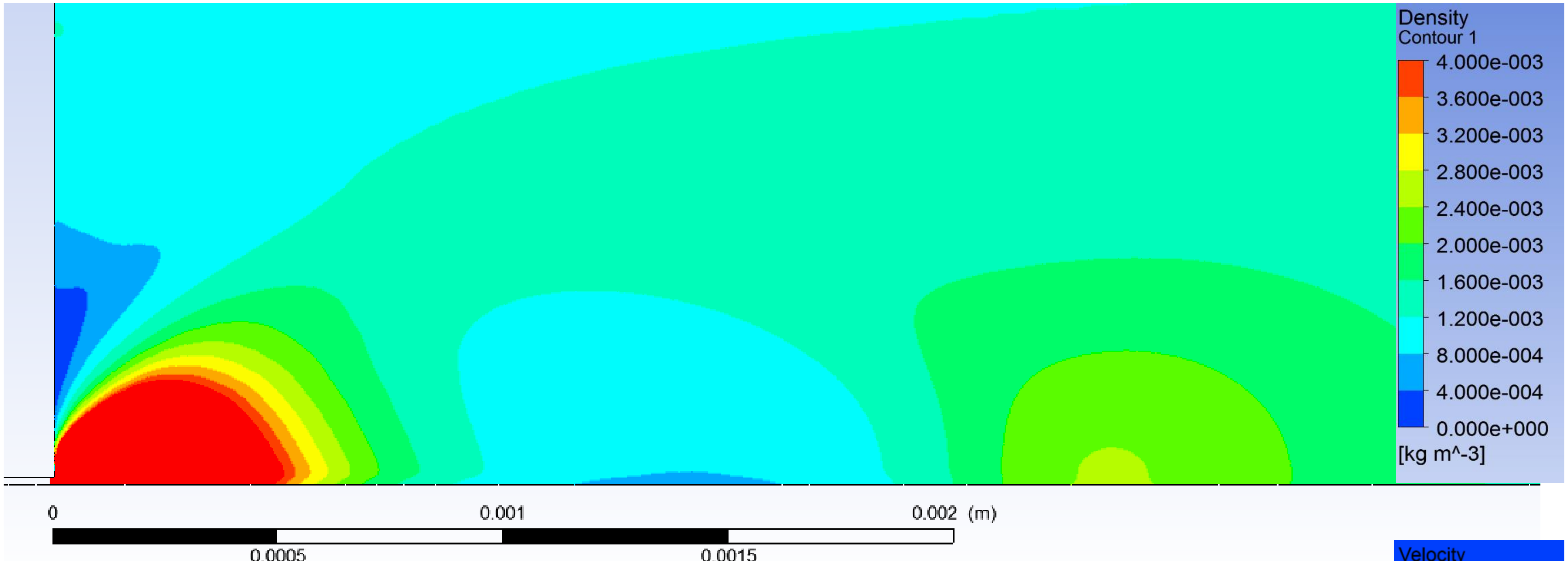
CFD simulation with nozzle chamber pressure $p_{NC} = 1\ [mbar]$

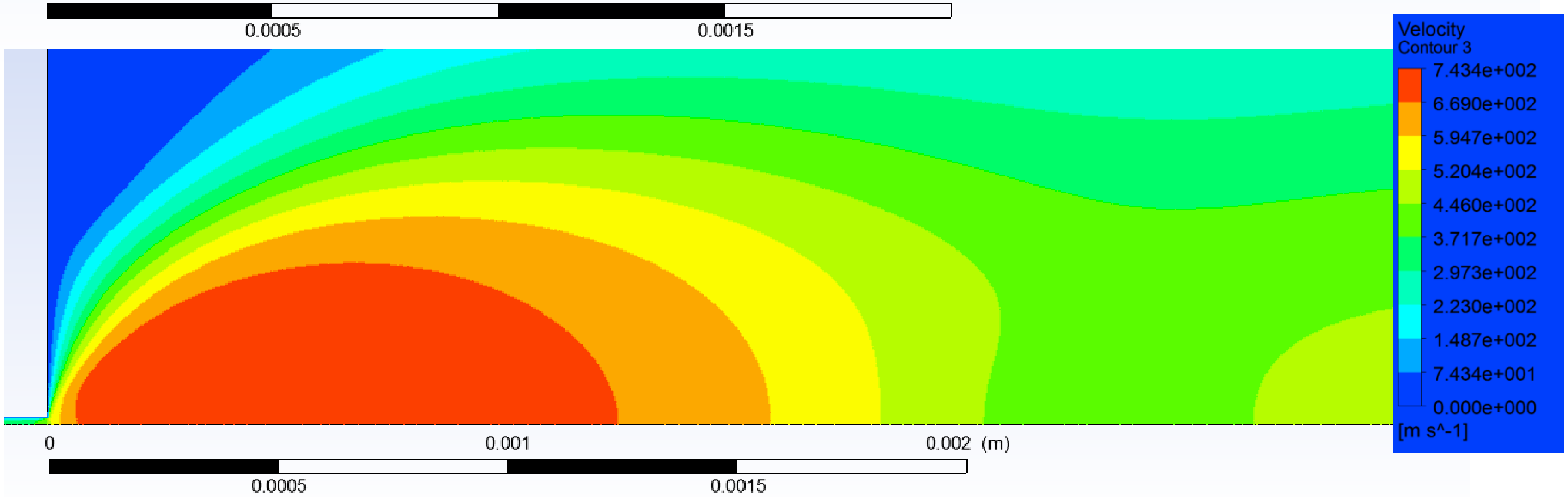
- Density distribution [kg/m3]
  Attention: scale between 4E-3 kg/m3 and 0.



CFD simulation with nozzle chamber pressure $p_{NC} = 1\ [mbar]$

- Density vs velocity distribution. Attention: scale between 4E-3 kg/m3 and 0.

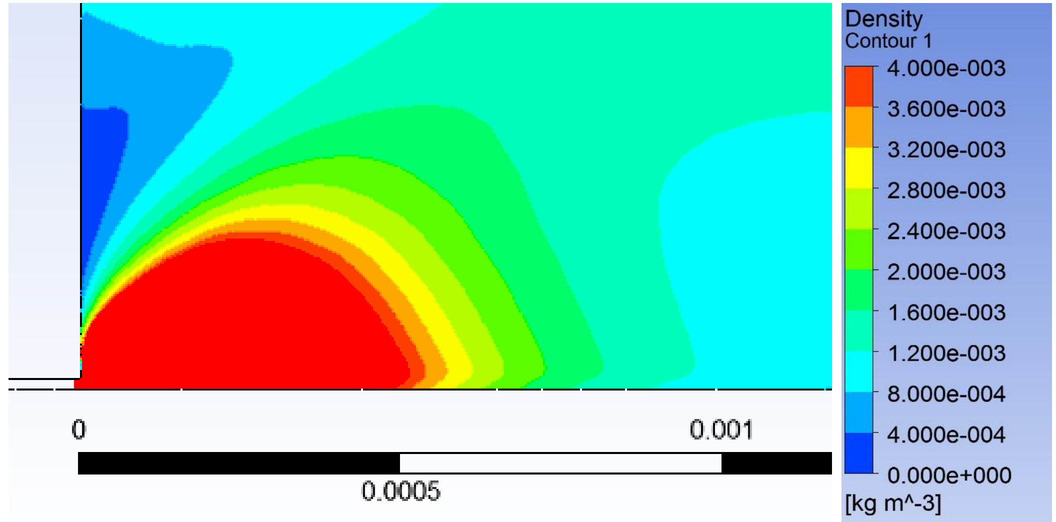




CFD simulation with nozzle chamber pressure $p_{NC} = 1\ [mbar]$

- Density distribution [kg/m3]: zoom in the high density region.

- $\rho_{Max} = 4 \cdot 10^{-3}\ \frac{kg}{m^3}$



- $\rho_{Max} = 4 \cdot 10^{-2}\ \frac{kg}{m^3}$