# Investigation of the Hydraulic Flip Phenomenon for Diesel Fuel Cavitation in a Planar Injection Nozzle



Thomas Frank
Ingenieurbüro Dr. Frank – CFD Consulting –


Tim Wittmann
Technische Universität Braunschweig

Realize Your Product Promise®

# Investigation of the Hydraulic Flip Phenomenon for Diesel Fuel Cavitation in a Planar Injection Nozzle

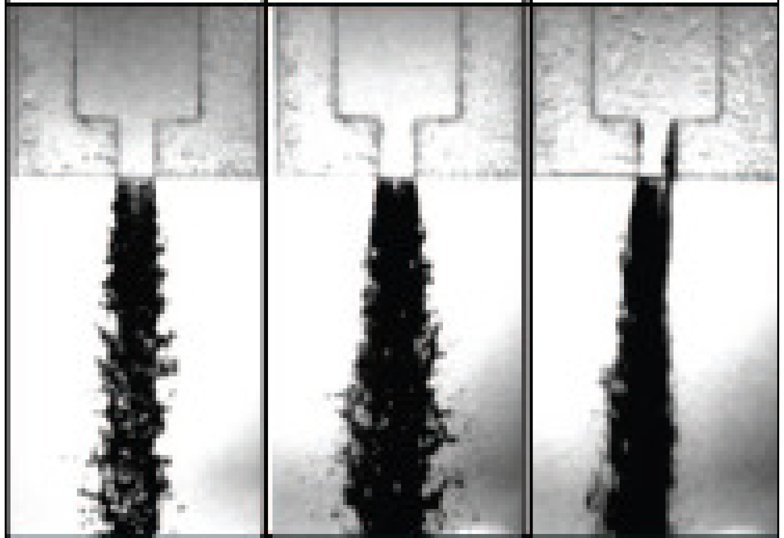

Thomas Frank [(*)], Tim Wittmann, Daniel Langmeyer

[(*)] ANSYS Fluid Solver Validation Manager

MFBU, ANSYS Germany, Otterfing

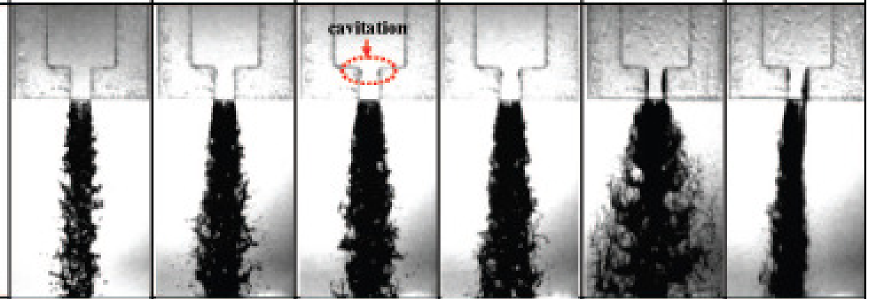

# Outline

- The Injector Experiment
- Objectives of the CFD Investigation
- Geometry, Mesh and CFD Setup
- Preliminary Investigations, Observations, Hysteresis Effects
- Target Flow Characteristics and the Hydraulic Flip
- Results with ANSYS CFX 16.0 and ANSYS Fluent 16.0
- Solver Comparison & Conclusions

| Turbulent flow | | Hydraulic flip |
|---|---|---|
| 0.13 | 0.19 | 0.40 |
| 5.2 | 9.2 | 14.4 |
|  | | |
| 4.6217 | 1.4765 | 0.6027 |

# Motivation

- **Injector systems in ICE (internal combustion engines)**
- **Usual fuel spray flow pattern is a symmetric conical jet/spray**

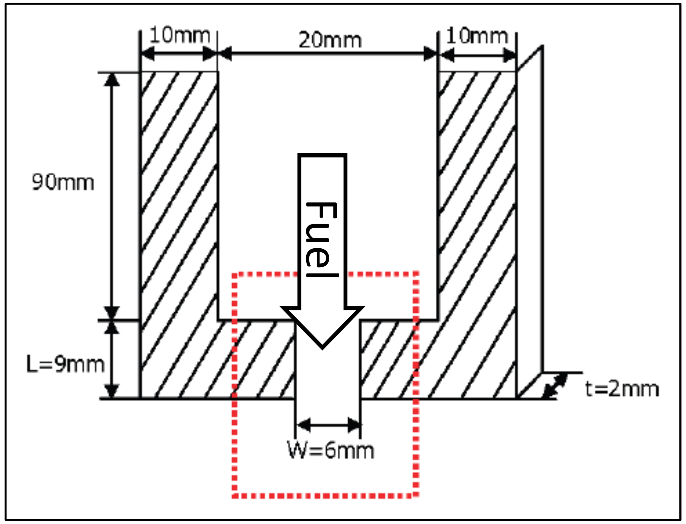
| | Turbulent flow | | Beginning point of cavitation | Growth of cavitatioin | | Hydraulic flip |
|---|---|---|---|---|---|---|
| Injection pressure($P_{inj}$,MPa) | 0.13 | 0.19 | 0.20 | 0.21 | 0.35 | 0.40 |
| Flow rate (Q,L/min) | 5.2 | 9.2 | 9.7 | 10.0 | 14.6 | 14.4 |
| Internal and External images |   | | | | | |
| Cavitation number(K) | 4.6217 | 1.4765 | 1.3282 | 1.2497 | 0.5863 | 0.6027 |

- **Under certain circumstances the fuel injection leads to the so-called “hydraulic flip”, i.a. flow attachment to one side of the injector wall or combustion chamber**
  **→ non-symmetric fuel jet/spray**
  **→ adverse effects on mixture formation and ICE engine performance**

# The Injector Experiment

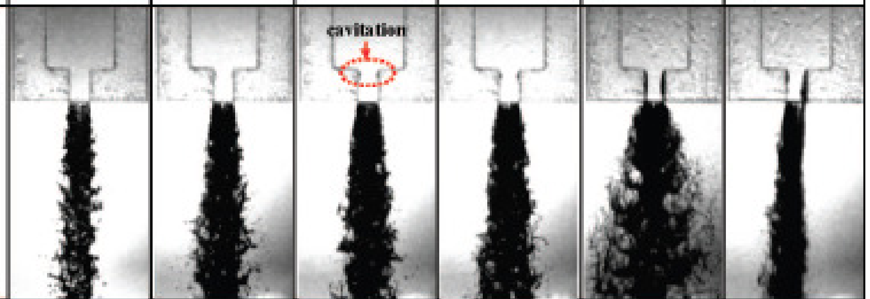
S.H. Park, H.K. Suh, Ch.S. Lee: “Effect of Cavitating Flow on the Flow and Fuel Atomization Characteristics of Biodiesel and Diesel Fuels”, *Energy & Fuels*, **2008**, Vol. *22*, pp. 605-613




Fuel injection pressure: 130 - 450 kPa
Ambient pressure: 100 kPa
Ambient temperature: 293 K

4 different flow patterns in dependence on inlet pressure:

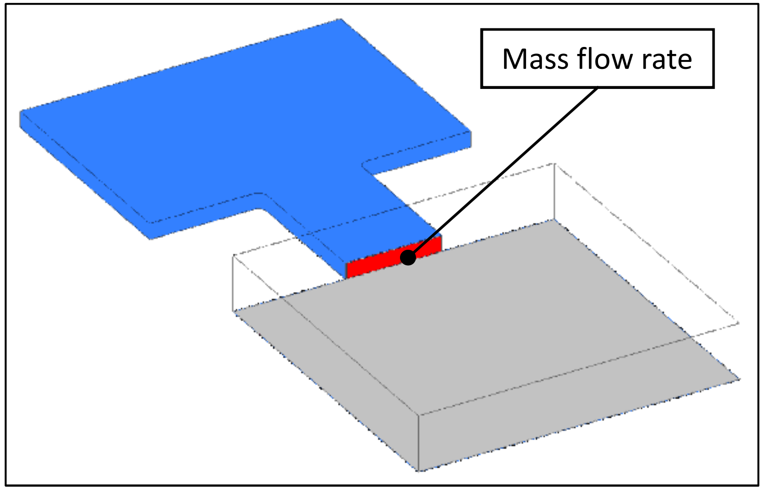
- Turbulent flow
- Beginning point of cavitation
- Growth of cavitation
- Hydraulic flip

| | Turbulent flow | | Beginning point of cavitation | Growth of cavitatioin | | Hydraulic flip |
|---|---|---|---|---|---|---|
| Injection pressure($P_{inj}$, MPa) | 0.13 | 0.19 | 0.20 | 0.21 | 0.35 | 0.40 |
| Flow rate (Q,L/min) | 5.2 | 9.2 | 9.7 | 10.0 | 14.6 | 14.4 |
| Internal and External images |   | | | | | |
| Cavitation number(K) | 4.6217 | 1.4765 | 1.3282 | 1.2497 | 0.5863 | 0.6027 |

# Objectives of the CFD Investigation

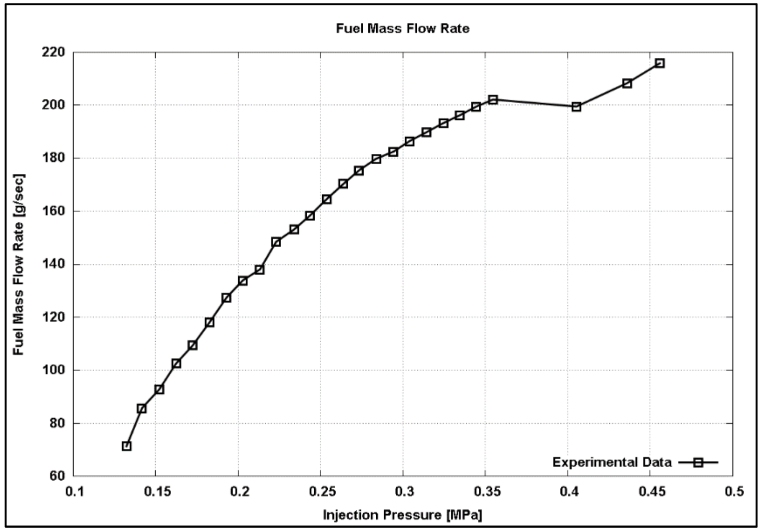
## Experimental information:

- **Injector mass flow rate**
- **Flow patterns from flow visualization**




## Questions to be addressed:

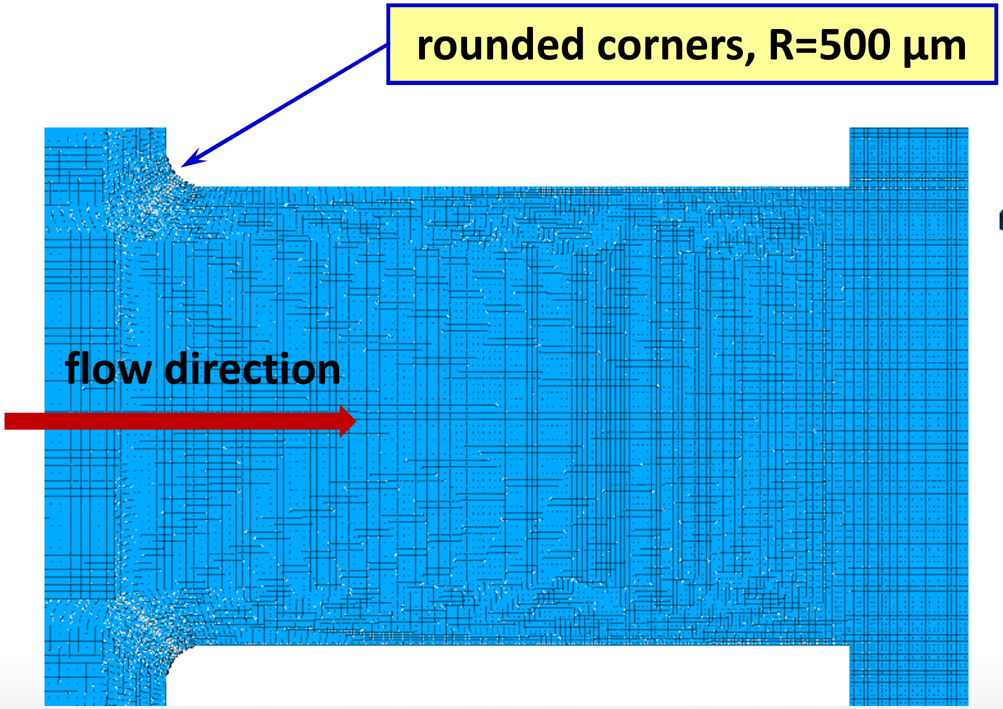
- **Can CFD predict and forecast the occurrence of a hydraulic flip?**
- **Can CFD predict the occurring injector mass flow limitation under the conditions of injector cavitation and hydraulic flip?**
- **Identification of a stable and reproducible CFD work flow**

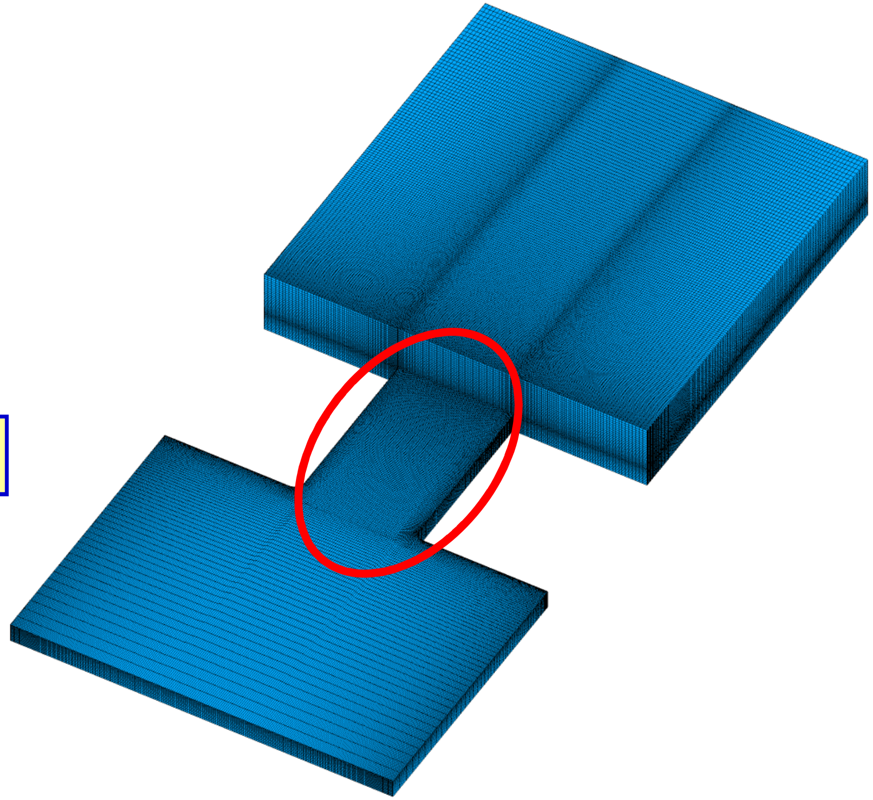


# Geometry and Computational Mesh

| | |
|---|---|
| Elements | 2 007 585 |
| Nodes | 2 095 488 |
| Max. Aspect Ratio | 17.4 |
| Min. Angle | 54.6 |



# Boundary Conditions

**Inlet:**

- Total pressure = **125 - 450 kPa**
- Turbulent intensity = 1%
- Hydraulic diameter = 3.5 mm
- Diesel Volume Fraction = 1

**Outlet:**

- Static pressure = 100 kPa
- Backflow turbulent intensity = 1%
- Backflow hydraulic diameter = 13.5 mm
- Air Backflow Volume Fraction = 1

**Upstream & Downstream Chamber Walls:**

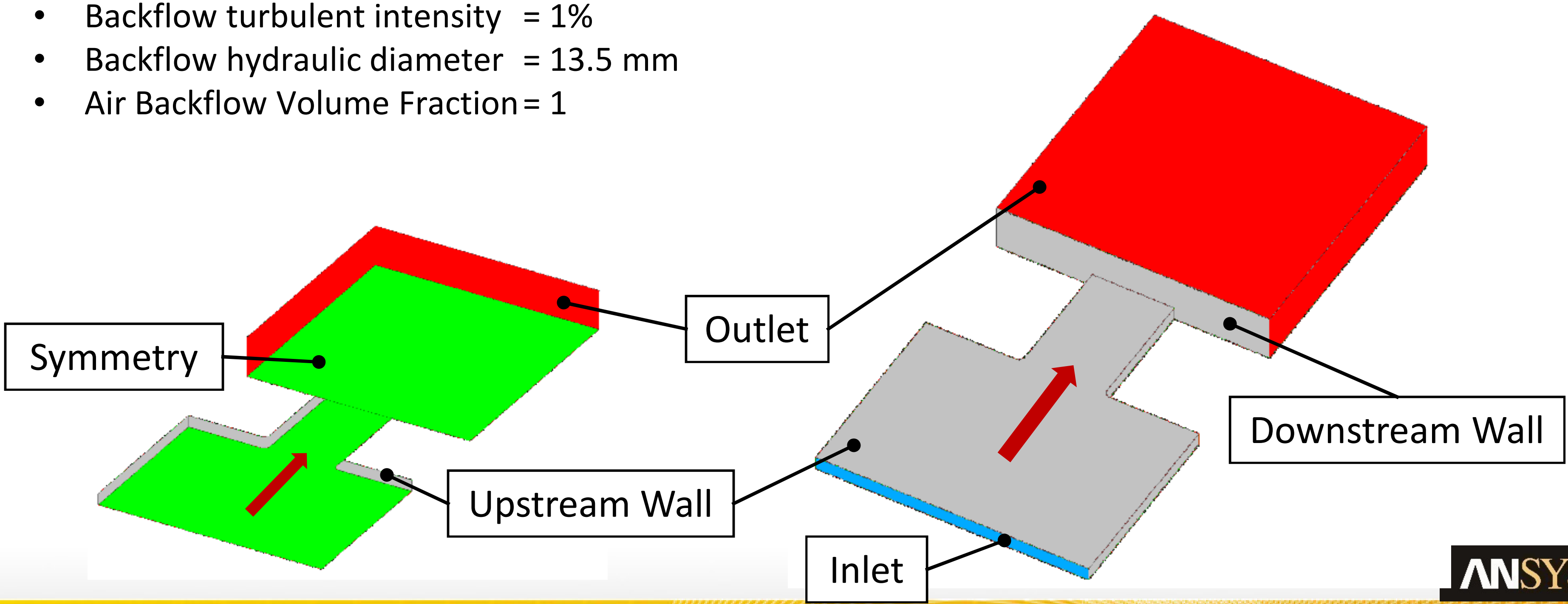
- No Slip Wall

**Symmetry Plane**

- Taking advantage of symmetry in z-direction

**Fluid:**

- Operating pressure = 100000 Pa



# CFD Setup & Fluid Material Properties

- 3-phase flow: Diesel liquid - Diesel vapor - Air
- ANSYS CFX : Homogenous Eulerian multiphase flow with cavitation
  ANSYS Fluent: 3-phase VOF with cavitation (homog. Eulerian mixture)
- Phase transition due to cavitation:
  - Diesel liquid – Diesel vapor: Zwart-Gerber-Belamri cavitation model
- Phase pairs without mass transfer:
  - Diesel liquid – air
  - Diesel vapor – air

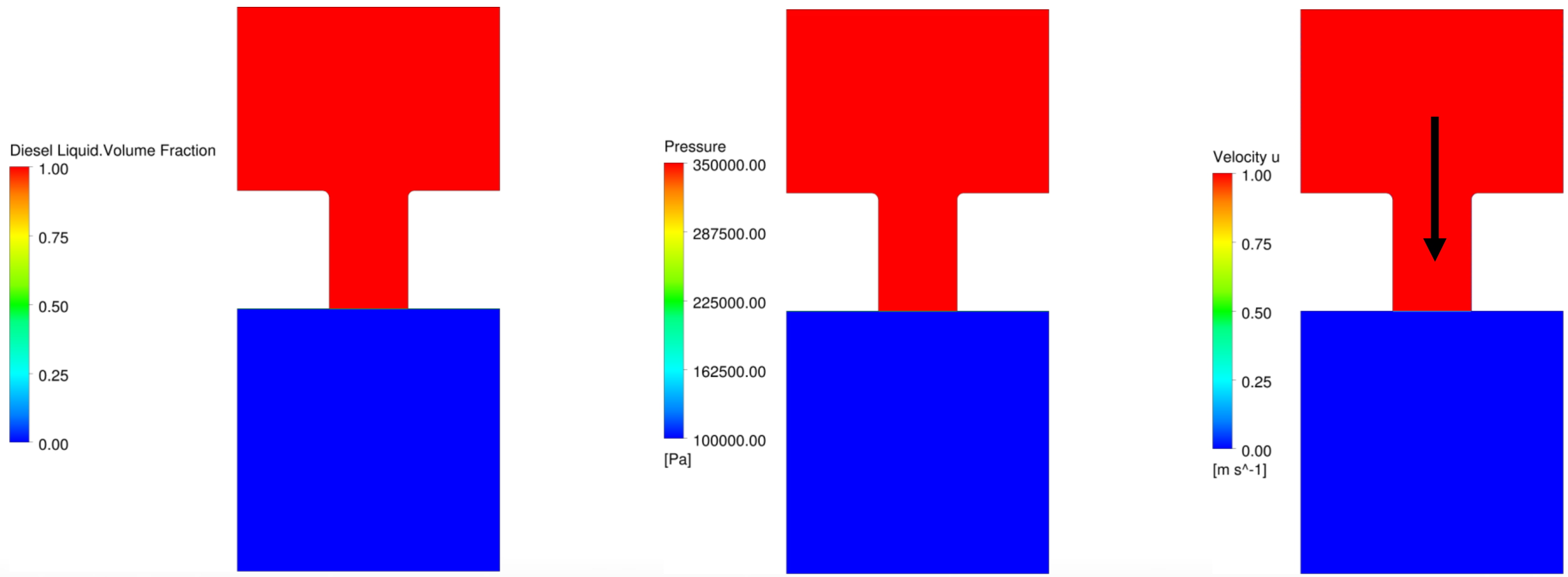
| | **Air** | **Diesel liquid** | **Diesel vapor** |
|---|---|---|---|
| **Density (kg/m³)** | 1.225 | 830 | 1 |
| **Viscosity (kg/m s)** | $1.7894 \cdot 10^{-6}$ | 0.00223 | $7 \cdot 10^{-6}$ |

| | **Diesel liquid / Air** | **Air / Diesel vapor** | **Diesel liquid / Diesel vapor** |
|---|---|---|---|
| **Surface tension (N/m)** | 0.026 | 0 | 0.026 |

# Flow Initialization
# Steady-state vs. Transient Flow Patterns

- **Preliminary investigations used a symmetric initialization:**
  - Applied for not or slightly cavitating flow regimes ($P_{Inlet}$=125 kPa,…,350 kPa)
  - To save computational effort for flow development in the injector, most CFD simulations have been initialized from a fully developed flow pattern corresponding to inlet pressure $P_{Inlet} \in$ [125 kPa,…,200 kPa]
  - Observation of steady-state flow patterns for $P_{Inlet} \in$ [125 kPa,…,350 kPa]
  - Observation of strong flow instability for $P_{Inlet} \in$ [350 kPa,…,450 kPa] → transient

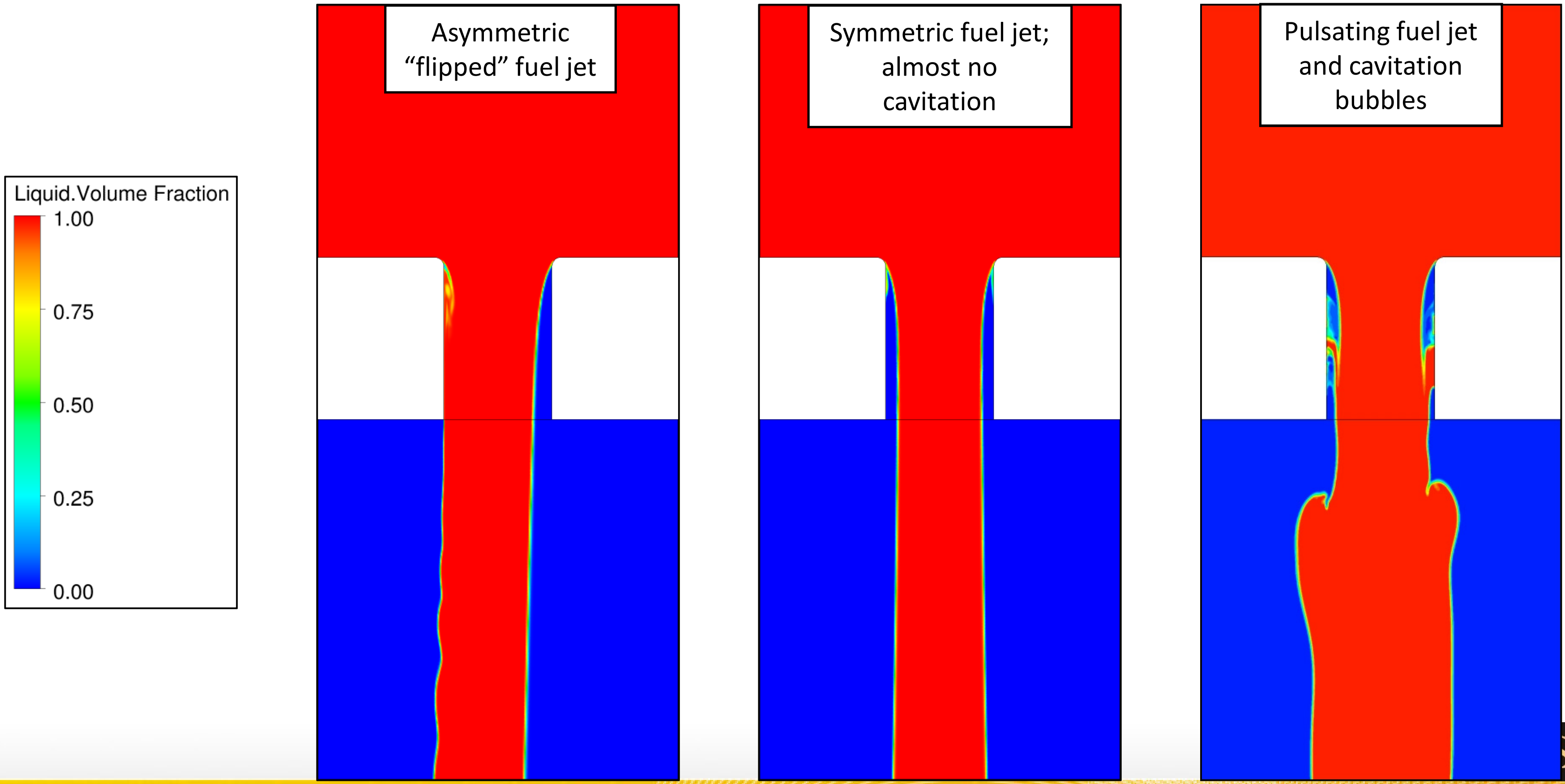
**Diesel Volume Fraction** **Inlet pressure** **Inlet Velocity**




**Outlet pressure**

# $1^{st}$ Preliminary CFD Results
# Flow Patterns, Flow Instability & The Hydraulic Flip

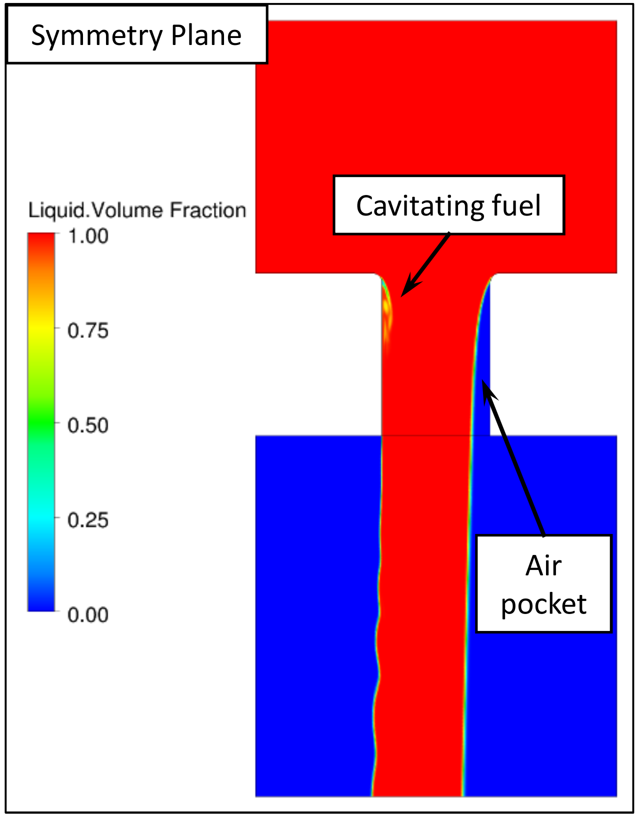
- The researchers Park et al. (2007) observed occurrence of hydraulic flip starting from $P_{Inlet}$=400 kPa
- Occurrence in CFD simulations so far depends on small flow disturbances, e.g. induced by certain solver, BC settings and flow initialization.
- In general there were three observed characteristic flow patterns for applied higher inlet pressure:

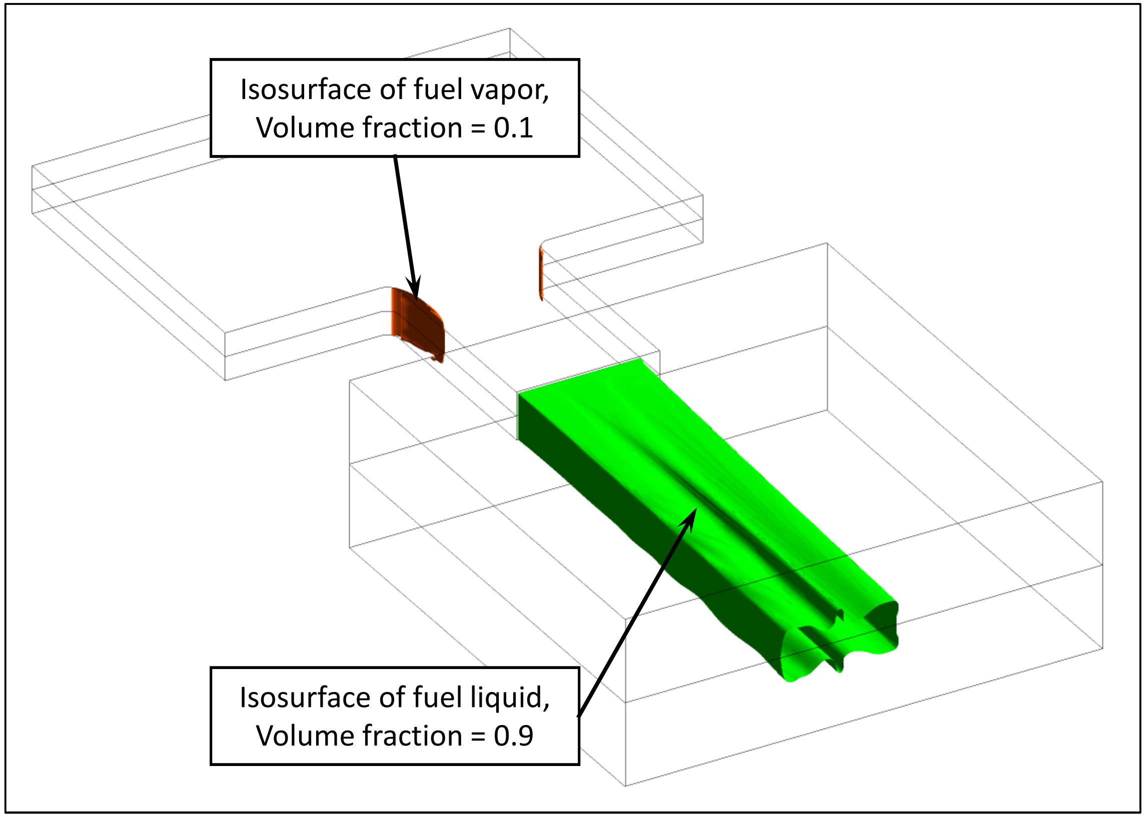


# 1st Preliminary CFD Results
# The Hydraulic Flip – Diesel Volume Fraction

- Once the hydraulic flip occurred, it appeared to be very stable
  → almost quasi steady-state
- Images show the Diesel volume fraction distribution for a transient calculation with occurrence of the hydraulic flip and $P_{Inlet}$=450 kPa.

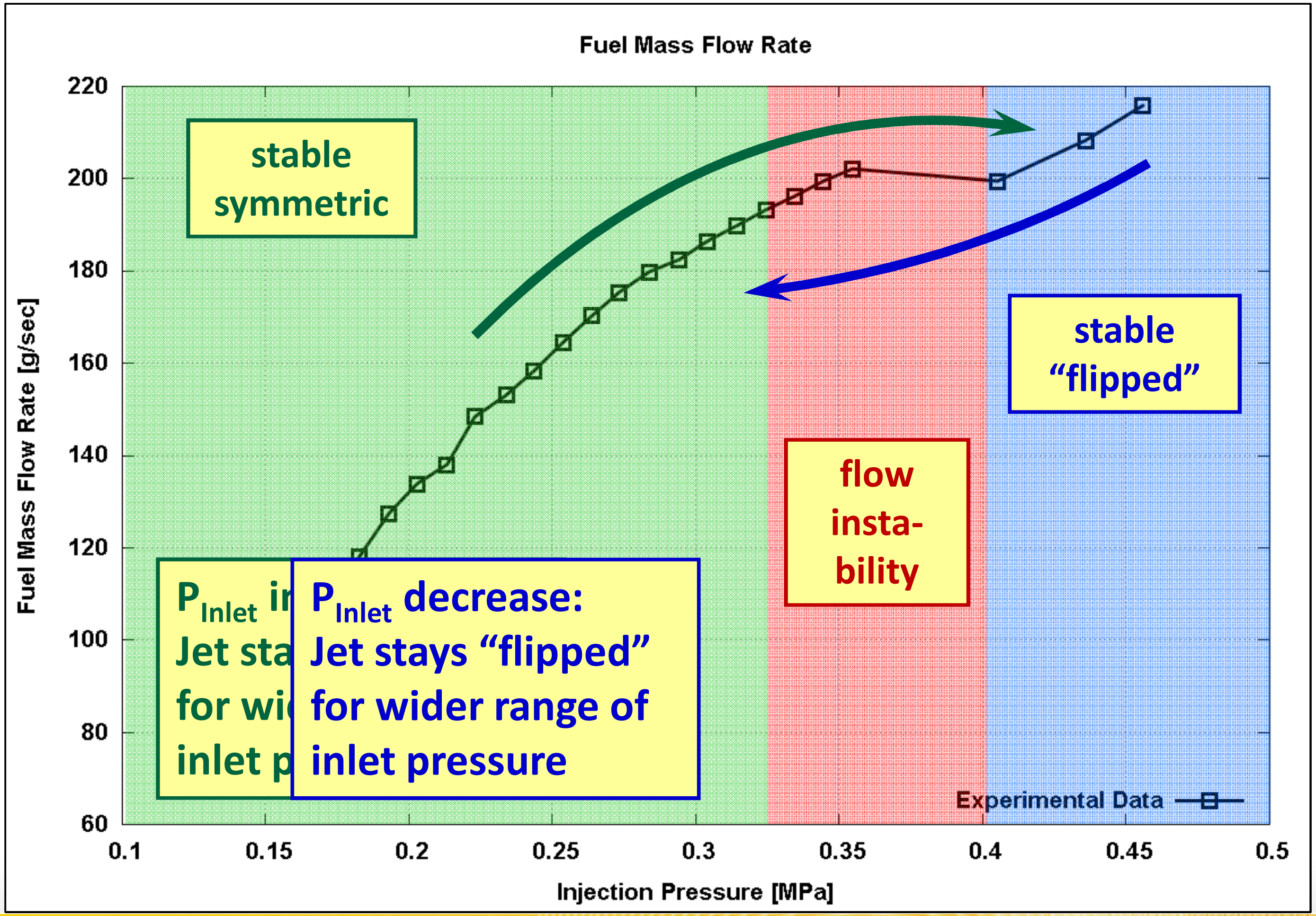


# $1^{st}$ Preliminary CFD Results
# Hysteresis Effects by Variation of $P_{Inlet}$

- Observed strong hysteresis effects in flow regime transitions from a stable symmetric fuel jet/spray to a stable hydraulic flip with inclined fuel jet/spray

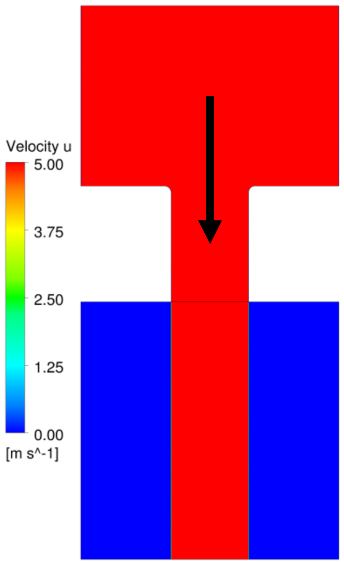



  November 27, 2015 

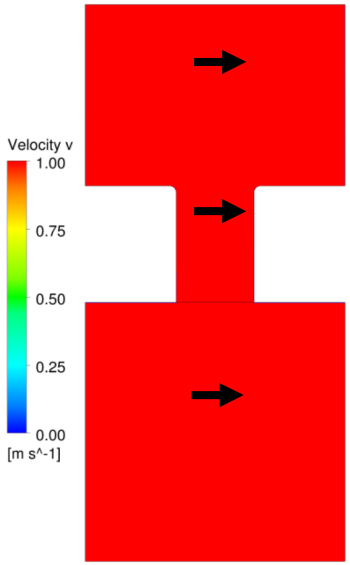
# Change in Flow Initialization

- **Attempt to avoid this kind of hysteresis and unstable CFD prediction**
- **Using domain initialization with a horizontal velocity component**
  → Reproducible hydraulic flip
  → Occurs only for inlet pressures which show the hydraulic flip in experiment as well
  → CFD simulations became reproducible and indepen-dent from numerics

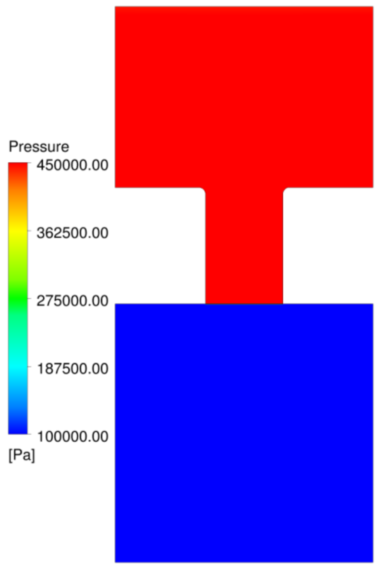
**Vertical velocity**



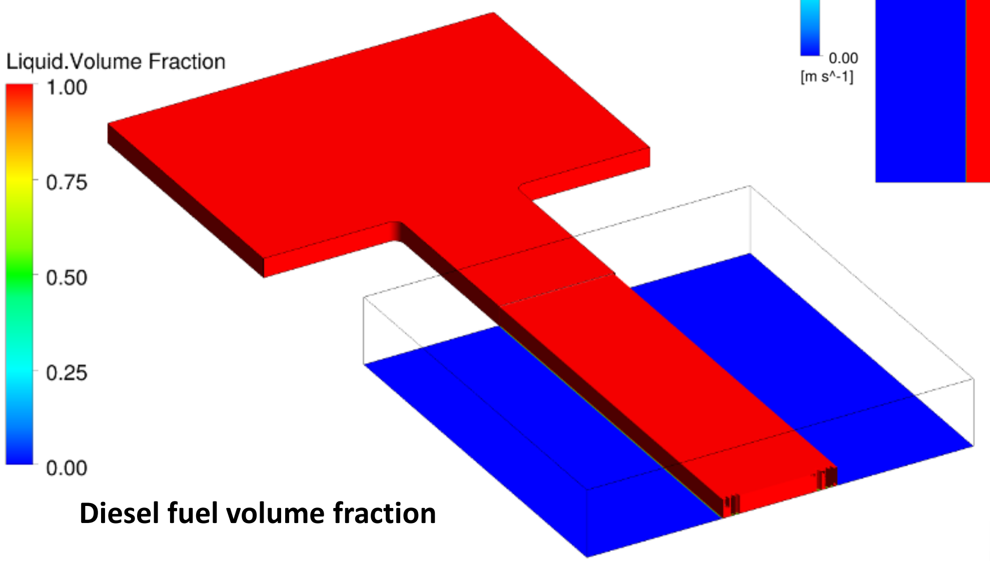

**Horizontal velocity**




**Inlet pressure**

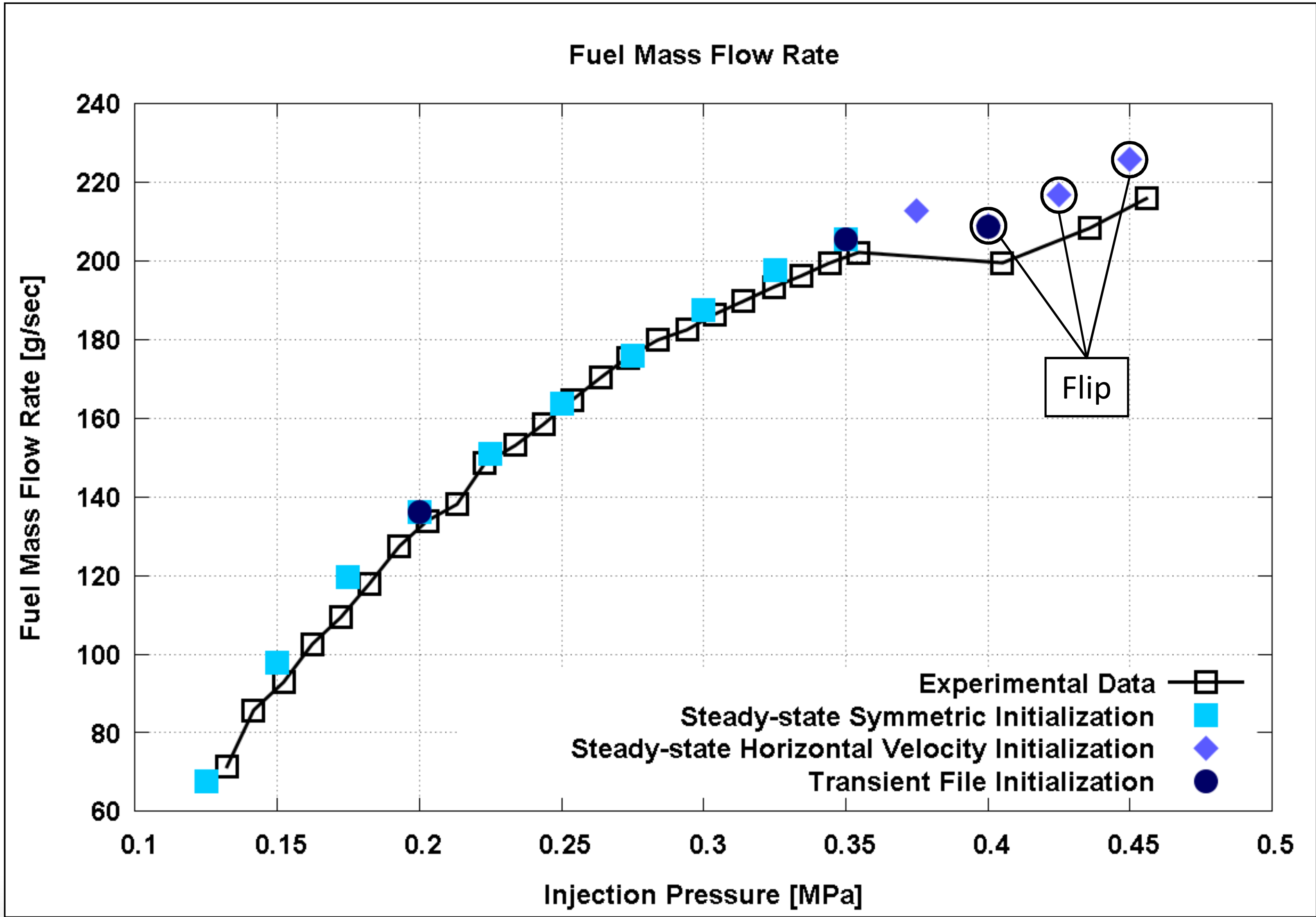



**Outlet pressure**

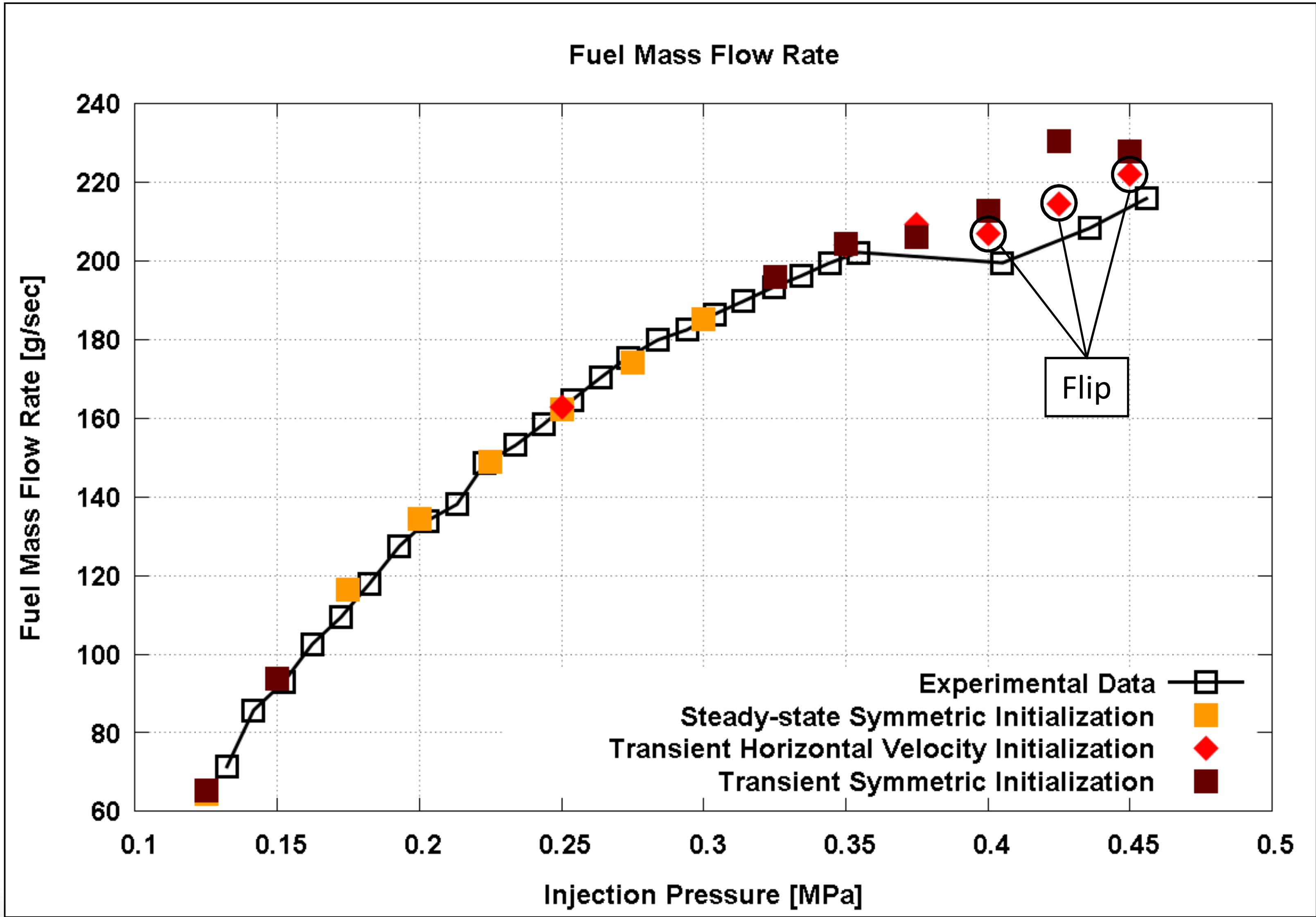



**Diesel fuel volume fraction**

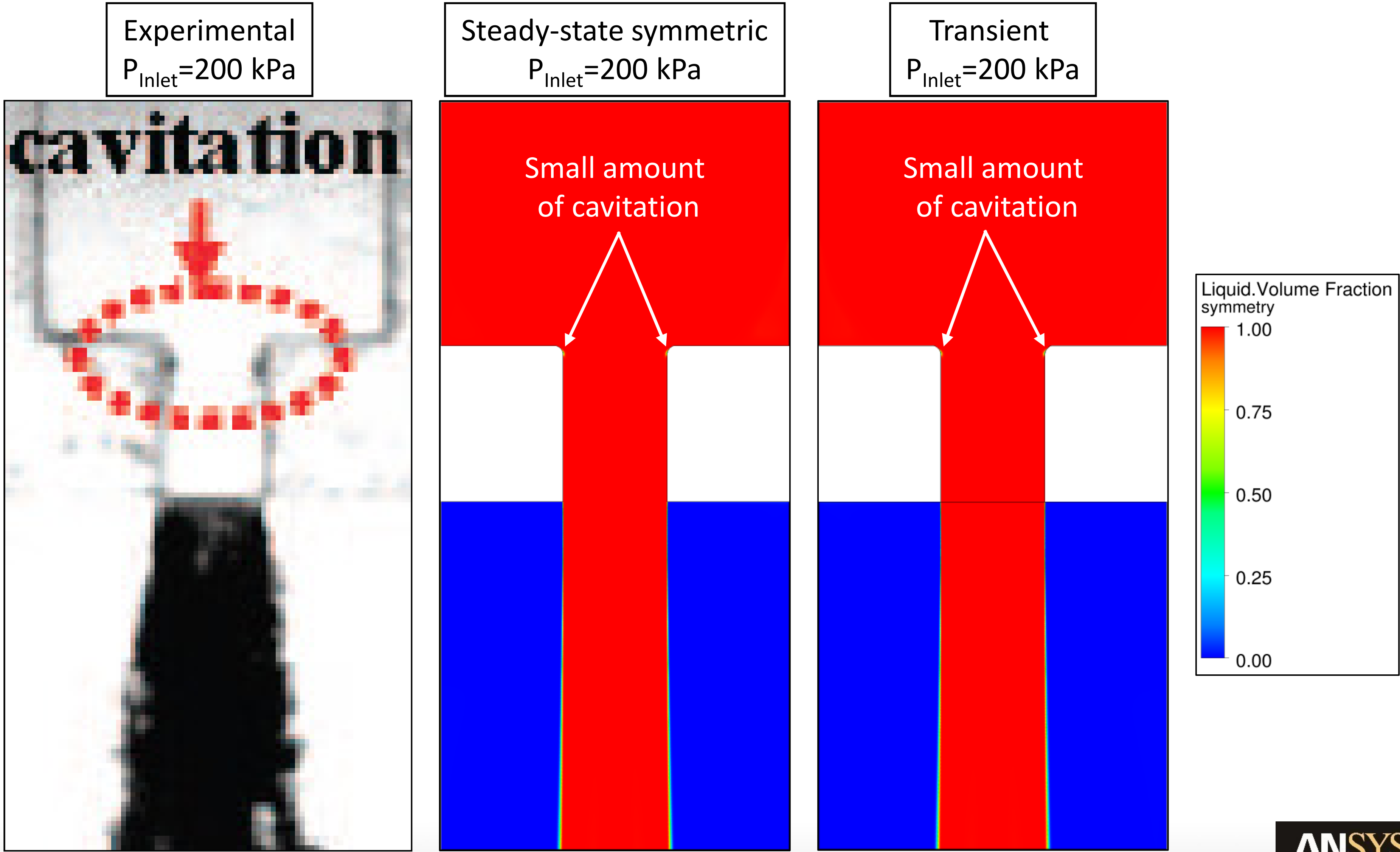
# ANSYS CFX Results – Fuel Mass Flow Rate



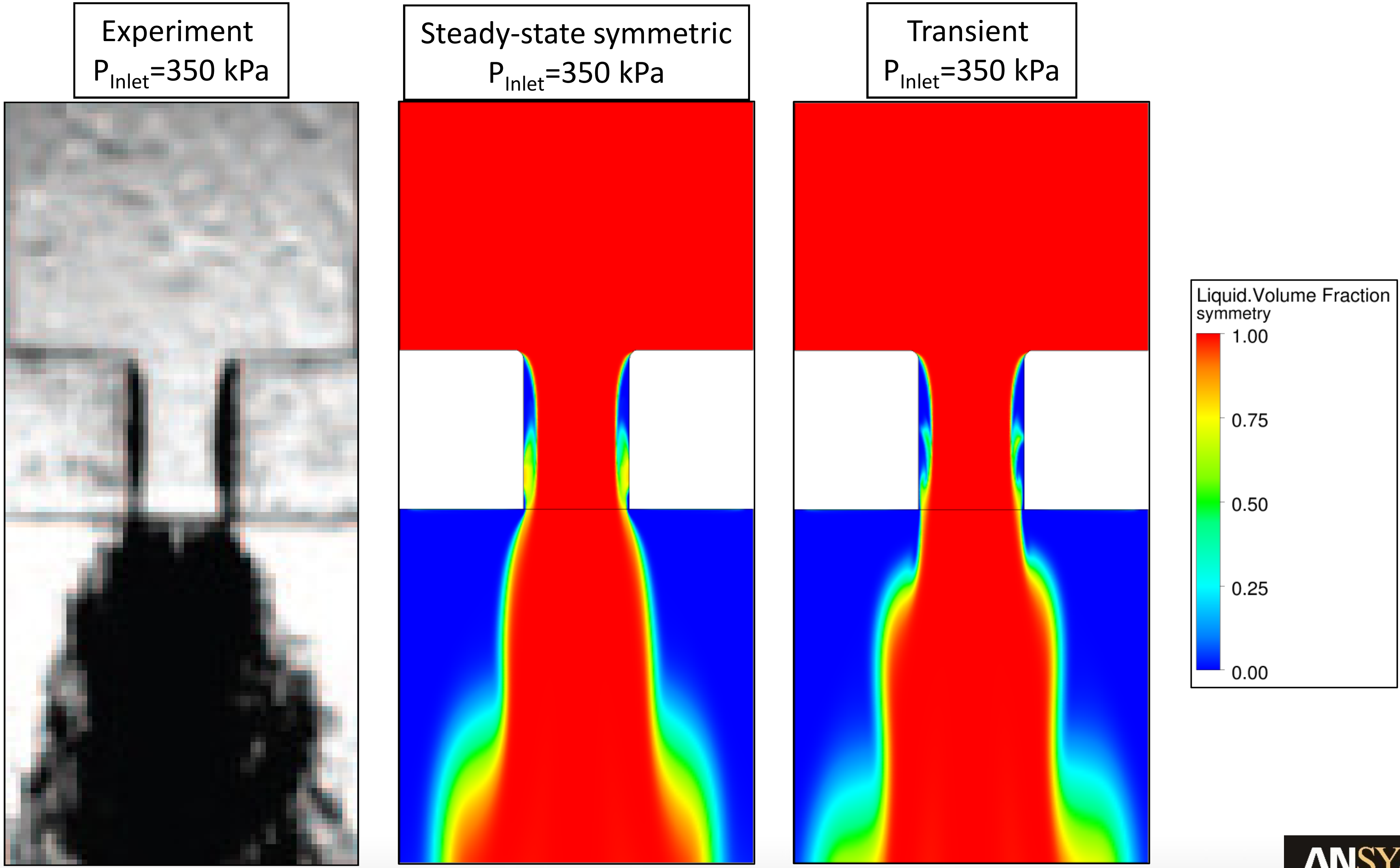
# ANSYS Fluent Results – Fuel Mass Flow Rate



# Results – Flow Pattern Visualization

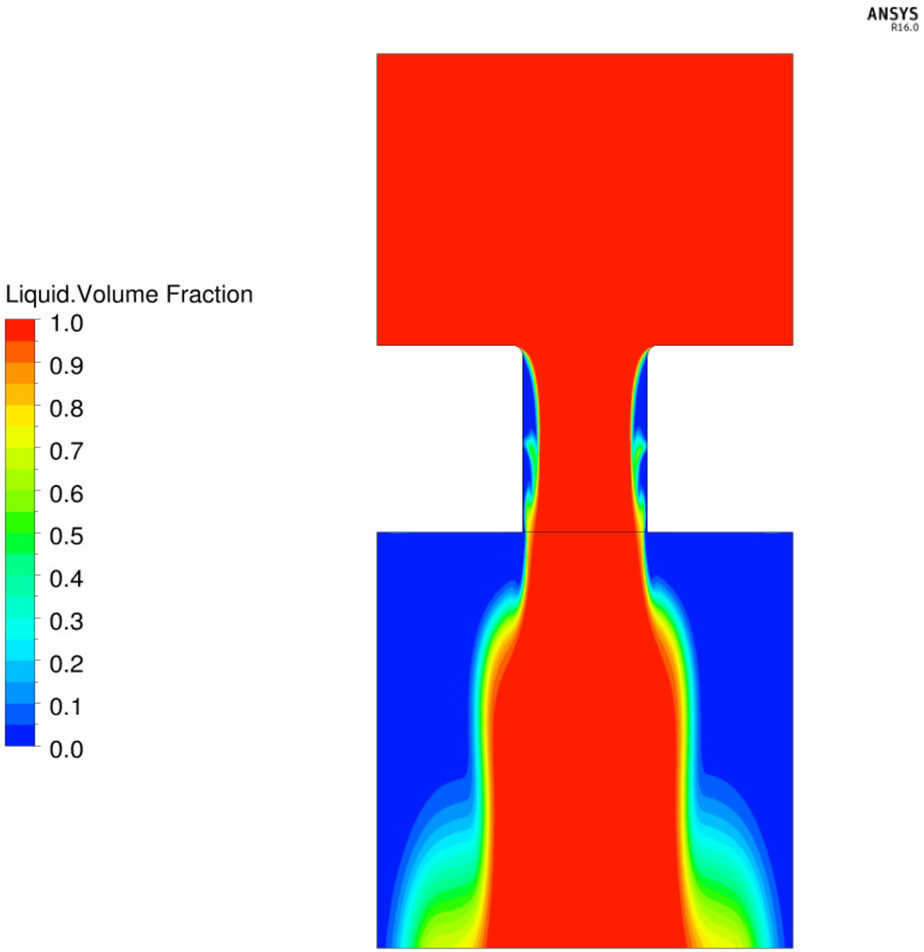


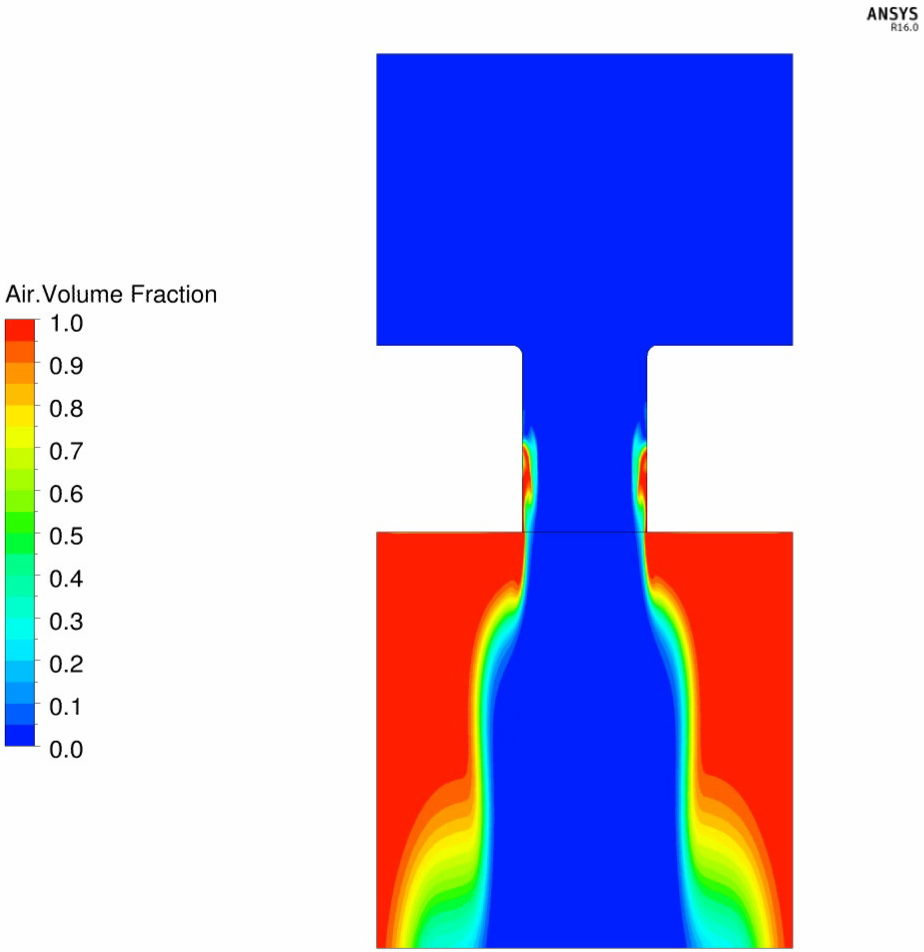
# Results – Flow Pattern Visualization



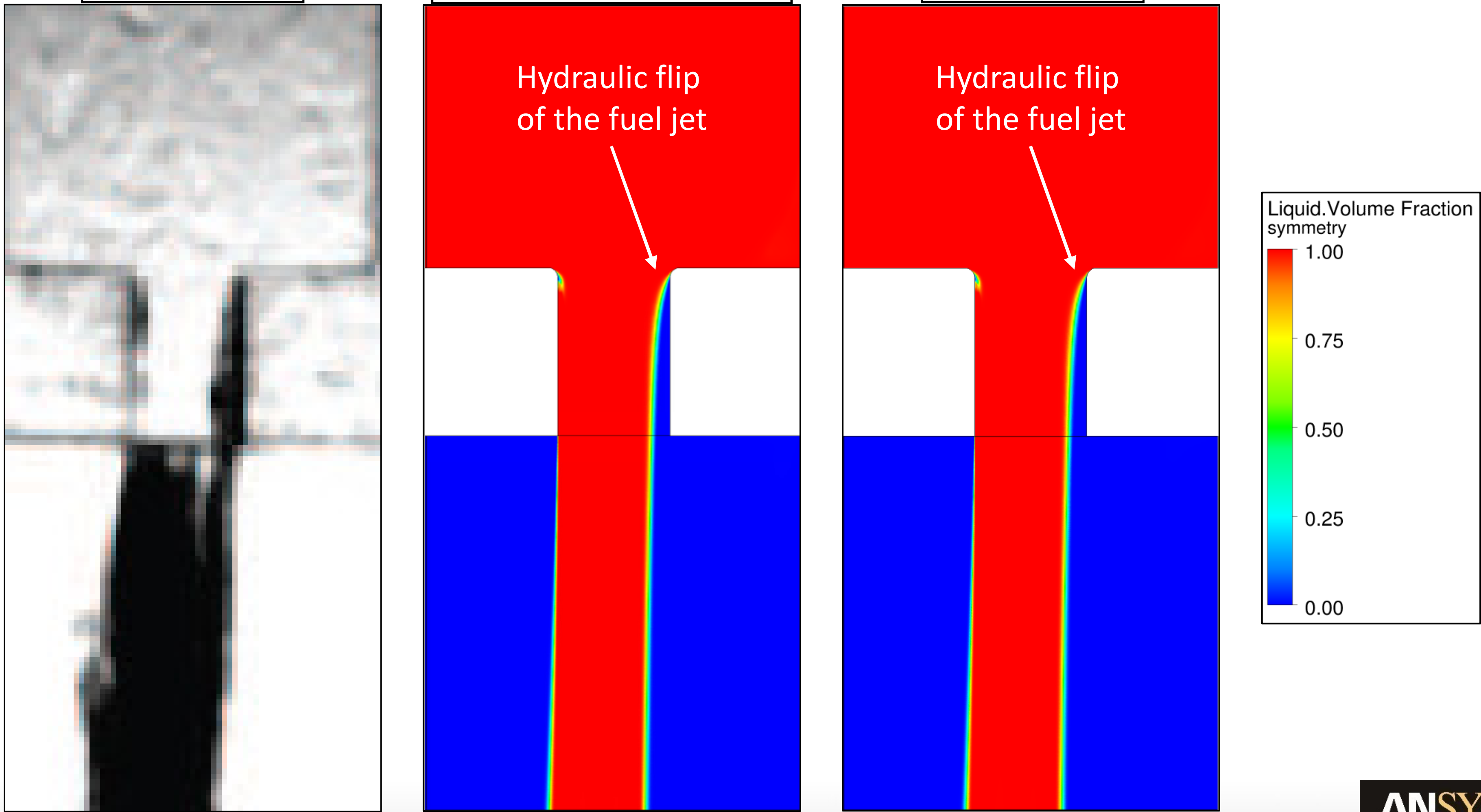
# Results – Pulsating Flow, $P_{Inlet}$ = 350 kPa
# Initialization from Stable Symmetric Jet



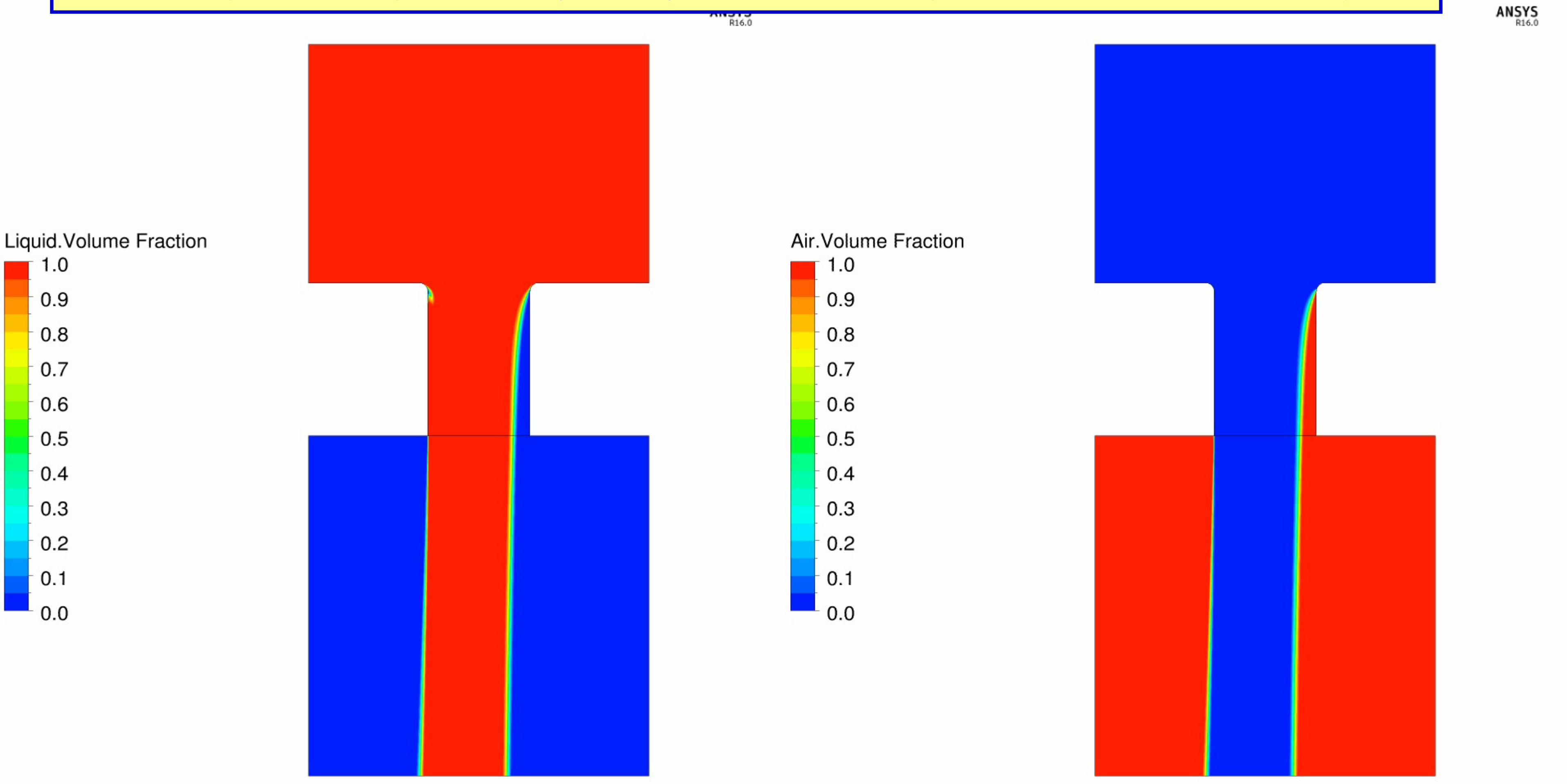
# Results – Flow Pattern Visualization

Experiment $P_{Inlet}$=400 kPa | Steady-state horizontal velocity, $P_{Inlet}$=400 kPa | Transient $P_{Inlet}$=400 kPa



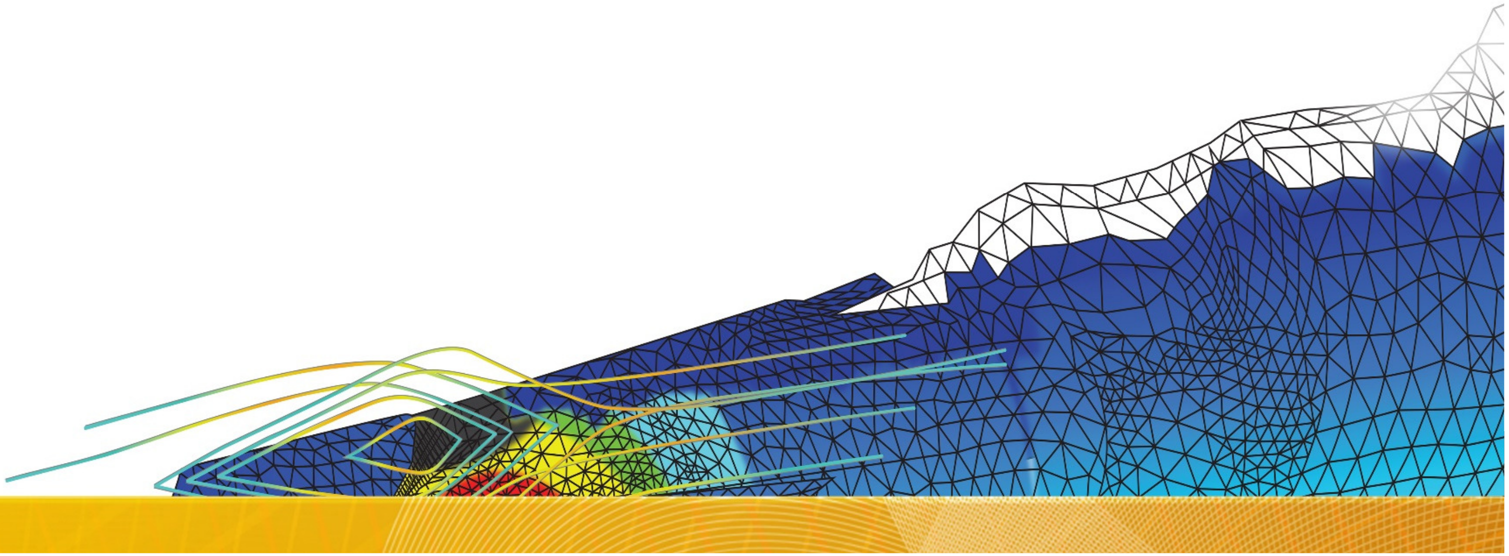
# Results – Video from Hydraulic Flip Simulation $P_{Inlet}$ = 400 kPa

**The hydraulic flip of the injected jet becomes very stable and almost steady**

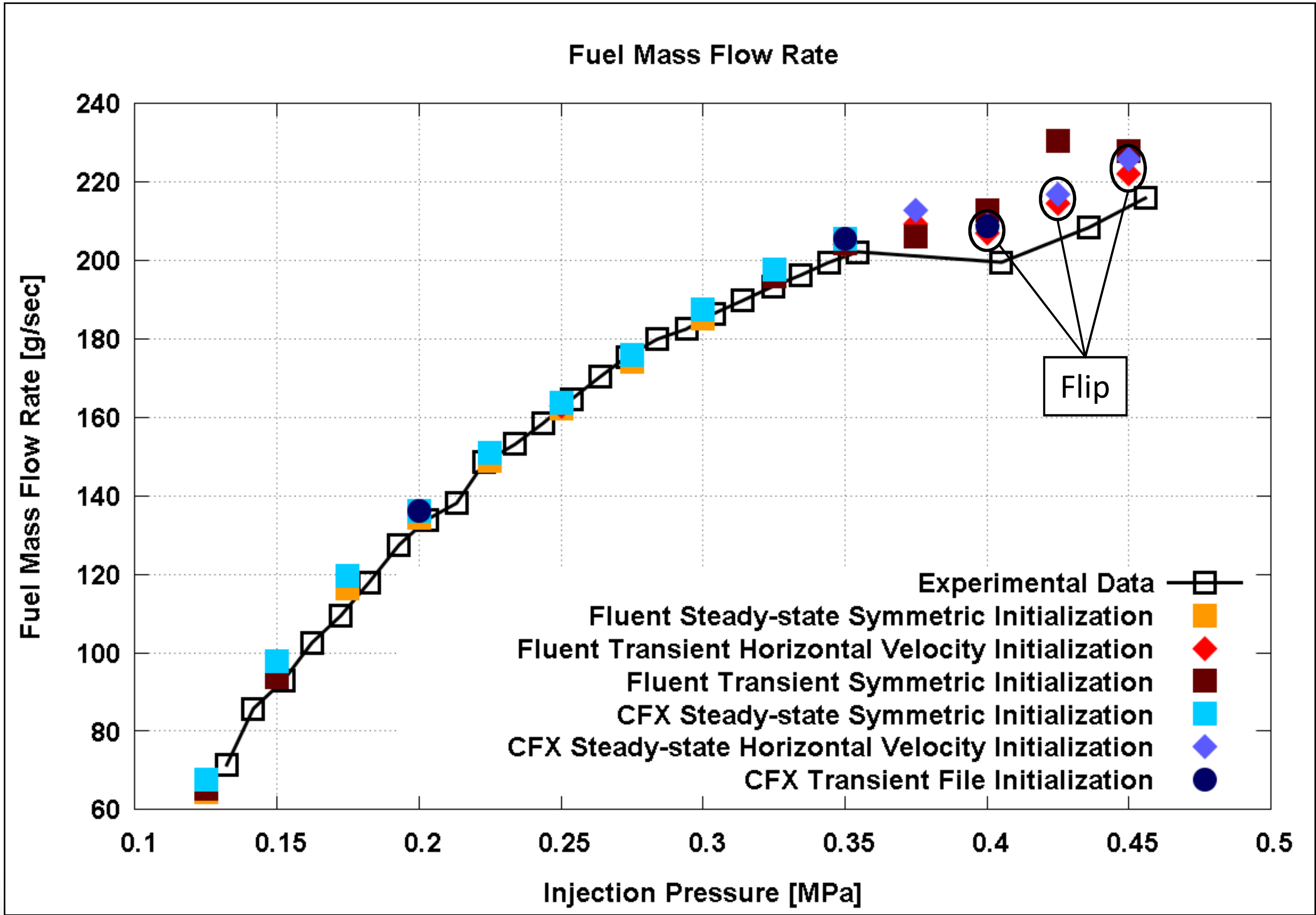


Realize Your Product Promise®

# ANSYS CFD Solver Comparison

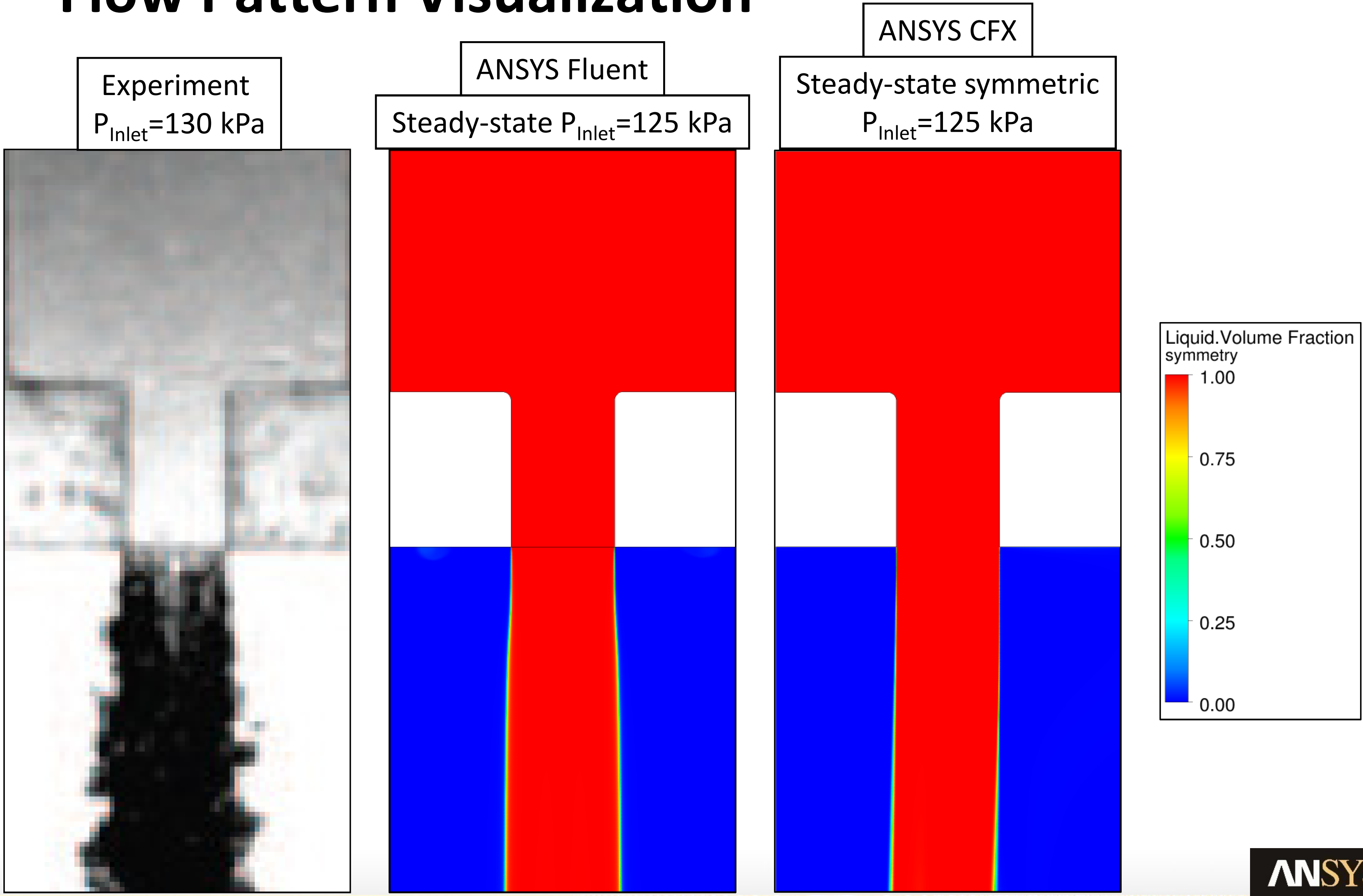


# ANSYS CFD Solver Comparison
# – Fuel Mass Flow Rate –



# ANSYS CFD Solver Comparison – Flow Pattern Visualization –



# ANSYS CFD Solver Comparison – Flow Pattern Visualization –

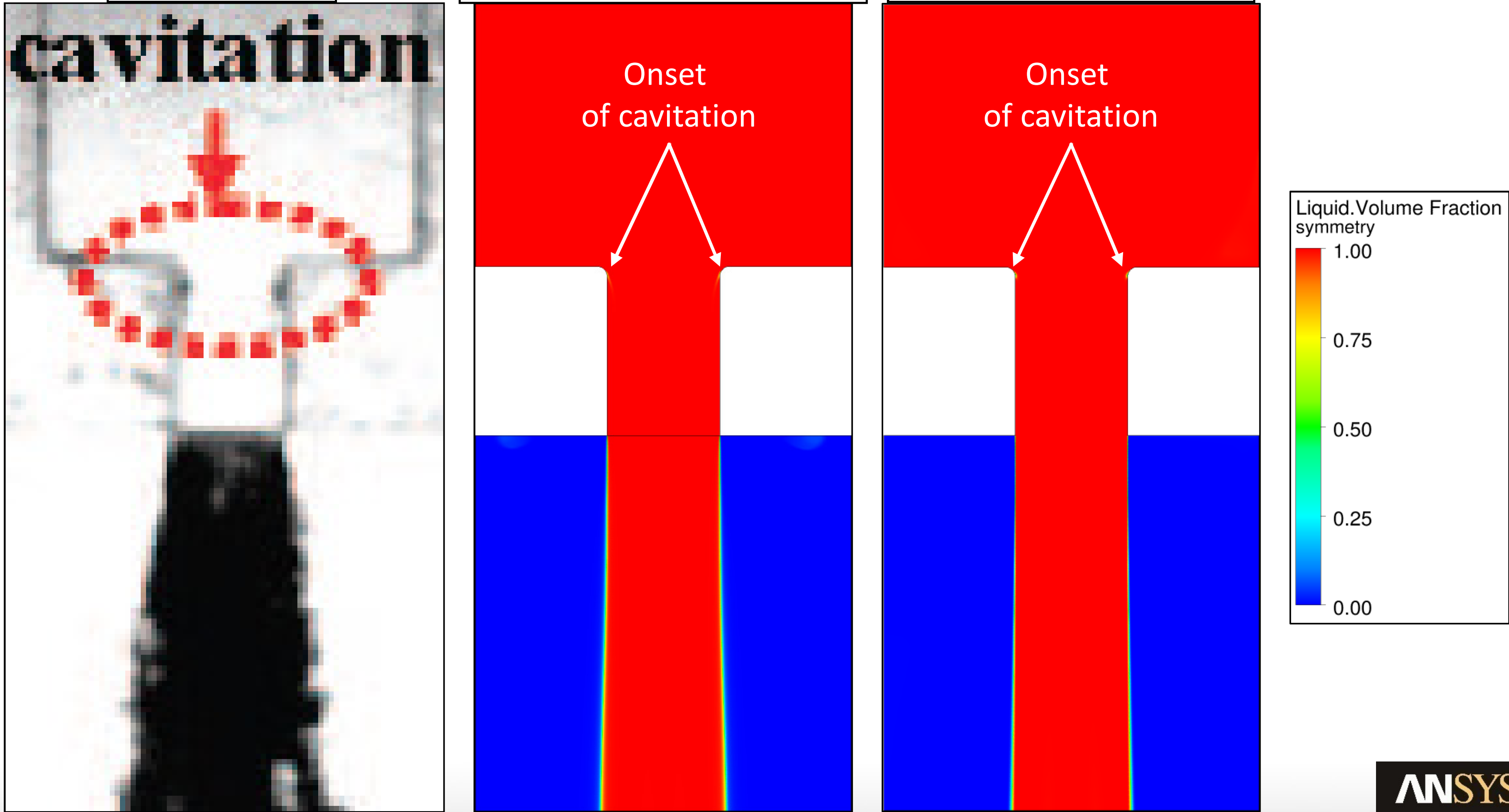
ANSYS CFX

ANSYS Fluent

Experimental $P_{Inlet}$=200 kPa

Steady-state $P_{Inlet}$=200 kPa

Steady-state symmetric $P_{Inlet}$=200 kPa

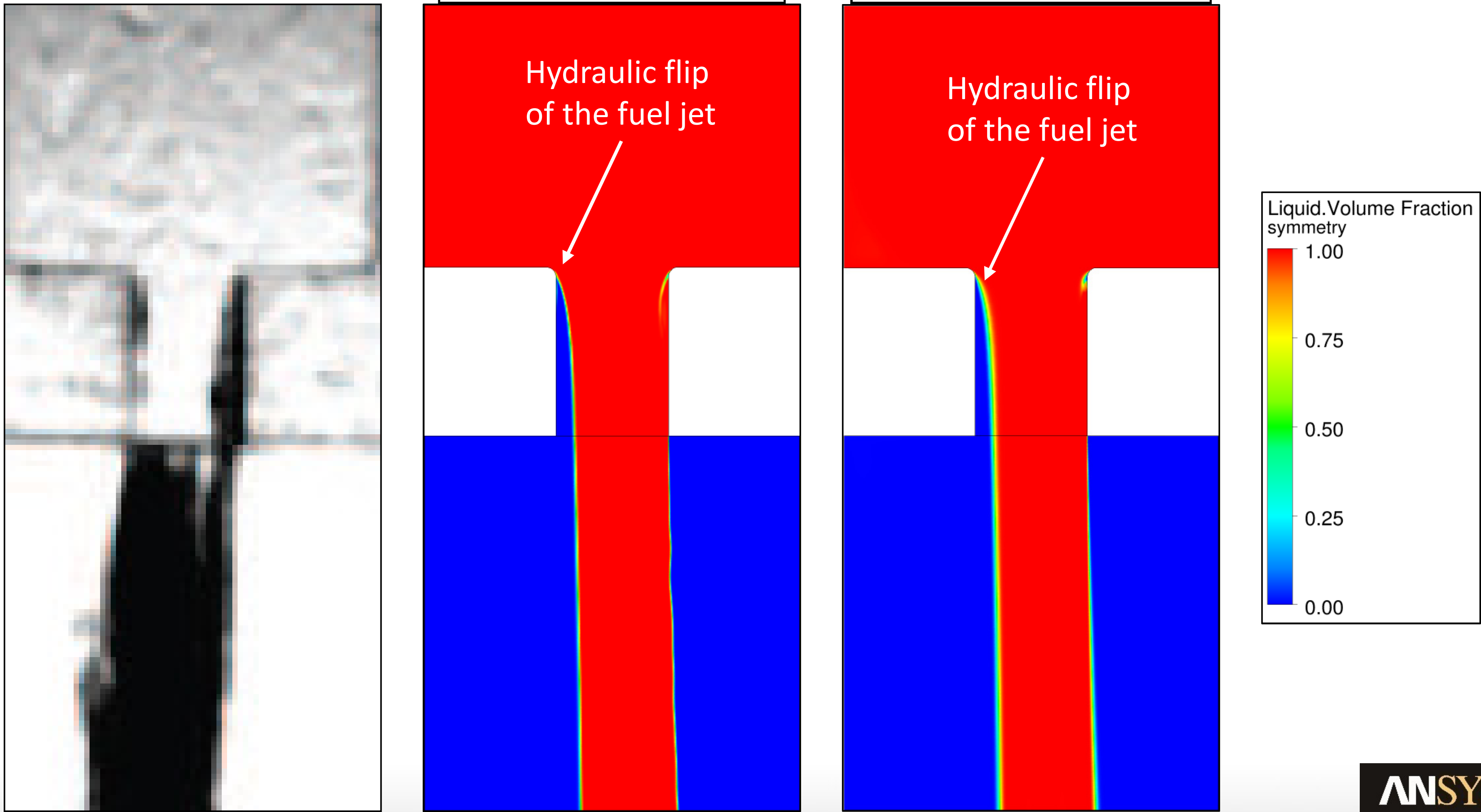


# ANSYS CFD Solver Comparison – Hydraulic Flip

Experiment
$P_{Inlet}$=400 kPa

ANSYS Fluent
Transient horizontal velocity, $P_{Inlet}$=400 kPa

ANSYS CFX
Steady-state horizontal velocity, $P_{Inlet}$=400 kPa



# Final Conclusions

- A CFD methodology was derived for both ANSYS CFD solvers ANSYS CFX and ANSYS Fluent to reliably and accurately predict the phenomenon of the hydraulic flip in fuel injectors.

- The liquid fuel mass flow rate in dependence of varying $P_{Inlet}$ is very well predicted for low and medium inlet pressure. The results for ANSYS CFX and ANSYS Fluent are almost identical and in very good agreement to experimental data.

- The mass flow rate limitation due to the occurrence of the hydraulic flip for high inlet pressure is predicted in good agreement to data. Results of ANSYS Fluent are slightly closer to the experiment.

- The flow regime change for varying $P_{Inlet}$ as predicted by CFD is in good agreement with experimental flow pattern visualization. The resolution of the spray breakup in the air filled chamber was not the focus of this investigation and would require a LES-like simulation approach.

# Thank you!