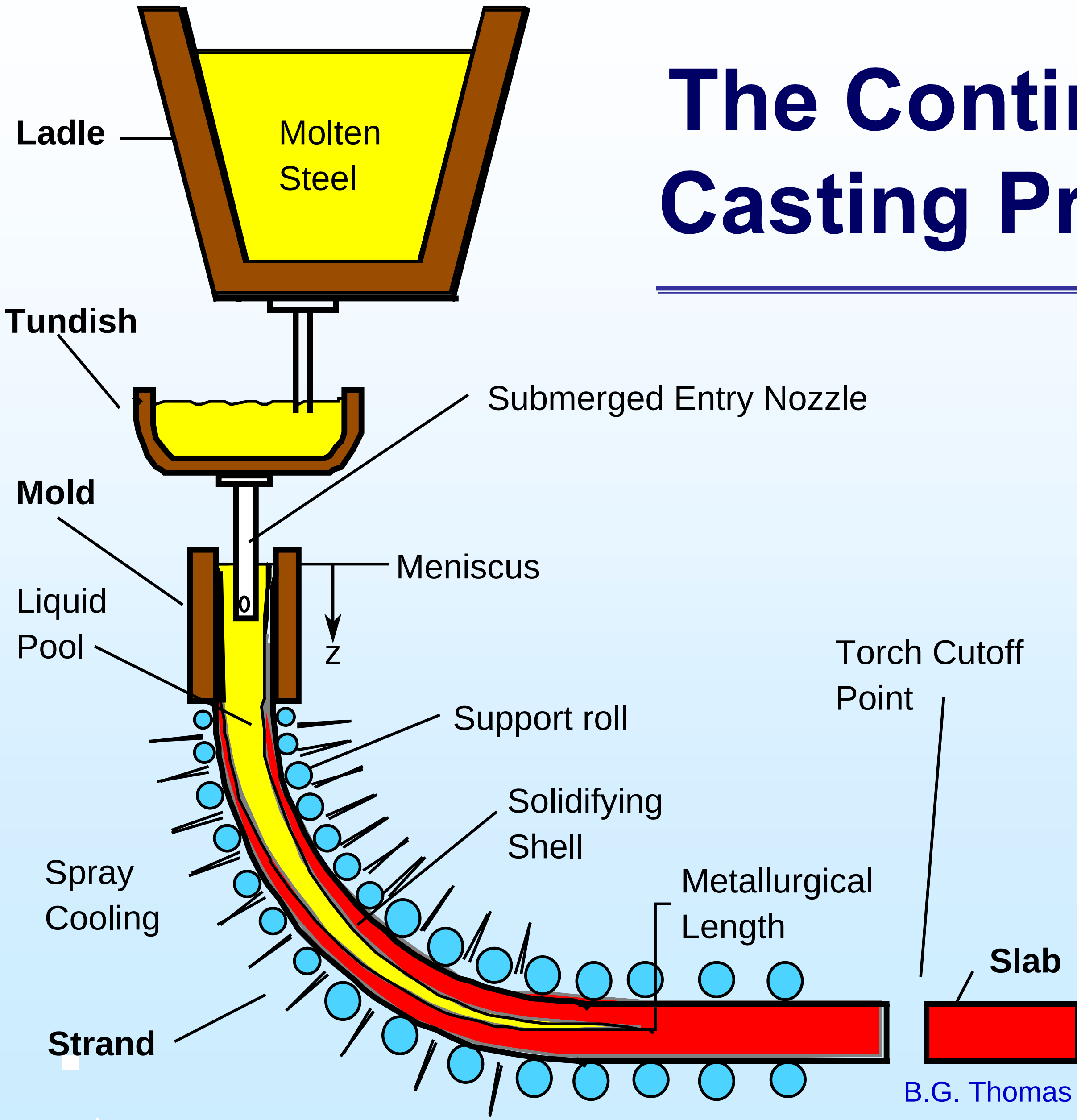
Large Eddy Simulation of Turbulent Flow in Continuous Casting of Steel

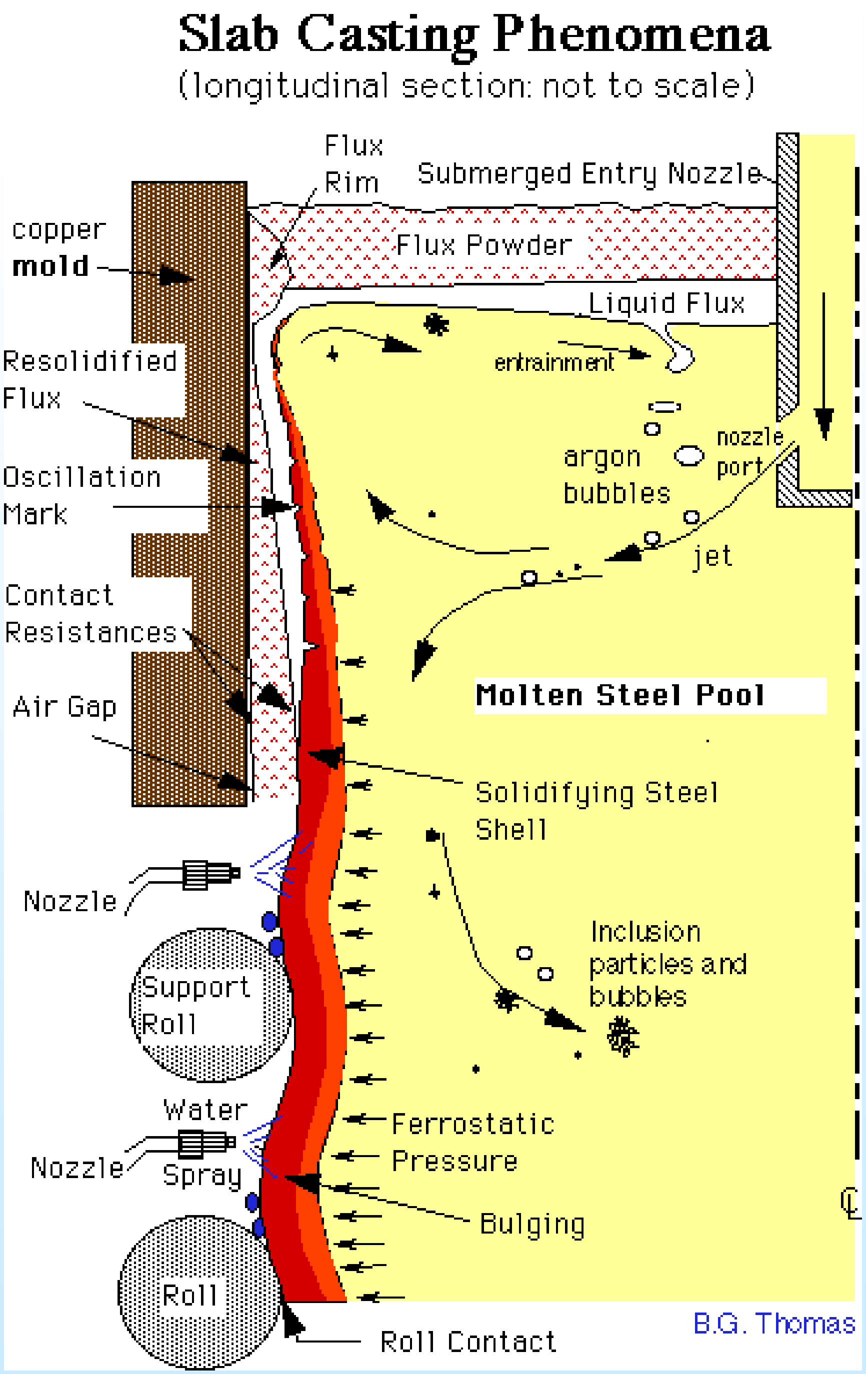
S. Sivaramakrishnan, B.G.Thomas & S.P.Vanka
(University of Illinois at Urbana Champaign)

Acknowledgements

- National Science Foundation for their research grant (NSF - Grant # DMI - 98-00274)
- Continuous Casting Consortium (CCC) at the University of Illinois at Urbana Champaign (UIUC) for their support of this research
- The National Center for Supercomputing Applications (NCSA) at the UIUC for computing time
- Dr. Mohammed Assar, Dr Pierre Dauby and the technicians at LTV Steel for their help with the PIV measurements

The Continuous Casting Process





The Continuous Casting Mold

Characteristics of Fluid Flow in the Mold

- High Reynolds number (Re_{inlet} ~ 50,000), swirling multiphase high temperature (T = 1500^{o}C) jet at inlet
- Jet phases include liquid steel, argon gas bubbles and alumina particles
- Impingement of semi-confined jet with high heat transfer rates
- Recirculation region above and below the jet
- Flow field highly turbulent, three dimensional and transient
- Solidification of molten steel at and below the impingement face
- Free surface covered with flux powder

Need for Transient Mold Flow Modeling

- **Eliminate defects caused by transient flow phenomena, which affect strand quality**

  - Entrainment of liquid flux by shear of the liquid layer
  - Entrapment of argon gas and alumina particles
  - Surface defects

- **Improve sensors used to monitor fluid flow in the mold**

- **Input for subsequent modeling**

  - Heat transfer
  - Solidification
  - Thermal stress analysis

- **Improve future caster design**

Tools to Study Transient Fluid Flow

- Water Modeling
  - Flow Visualization
  - Particle Image Velocimetry (PIV) Measurements

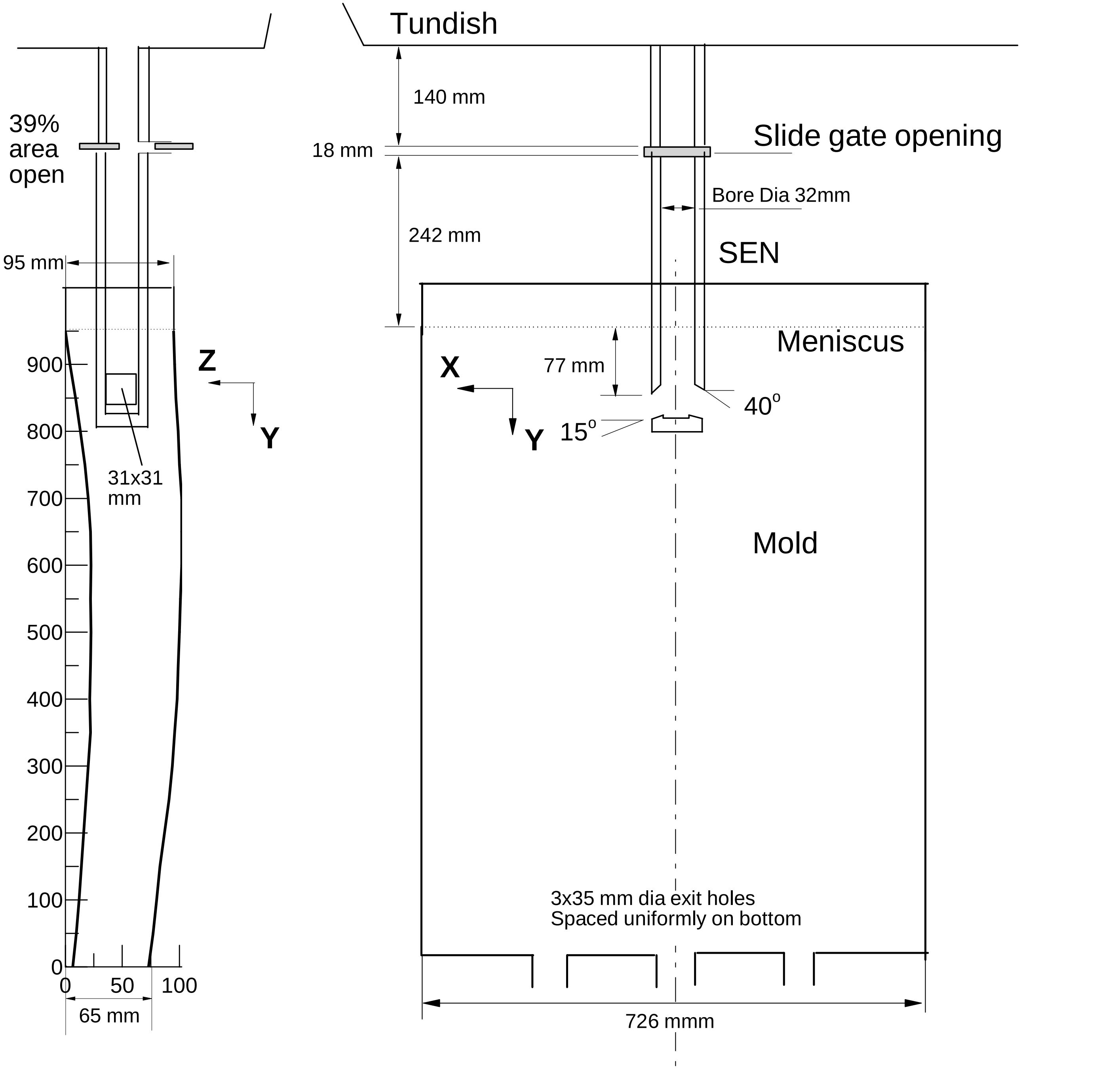
- Numerical Simulation
  - K-ε Models - Unsteady
  - Large Eddy Simulation (LES) – Transient

- Actual Caster
  - eg. Electromagnetic Sensors

Water Modeling

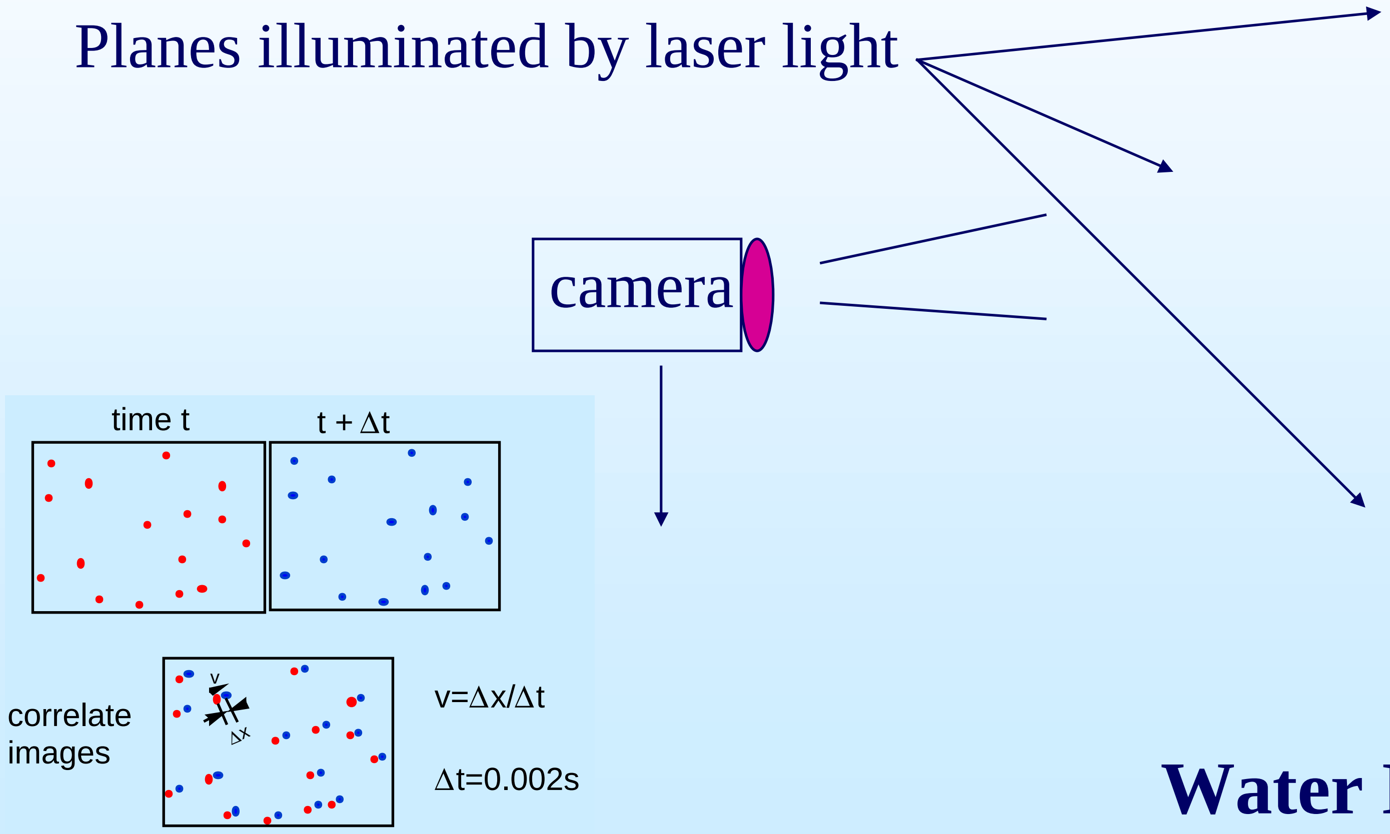
- Kinematic viscosity of molten steel is approximately equal to that of water
- Fluid flow in the caster can be studied using a scaled, transparent, Plexiglas water model
- Take into account steel shell using appropriate taper
- Use Particle Image Velocimetry (PIV) to quantify the fluid flow in the water model
- As a first study perform Large Eddy Simulations (LES) of the fluid flow in the water model

Water Model



Tundish
39% area open
95 mm
140 mm
18 mm
242 mm
Slide gate opening
Bore Dia 32mm
SEN
Meniscus
Z
Y
X
Y
77 mm
40°
15°
31x31 mm
900
800
700
600
500
400
300
200
100
0
0
50
100
65 mm
Mold
3x35 mm dia exit holes
Spaced uniformly on bottom
726 mmm

Particle Image Velocimetry



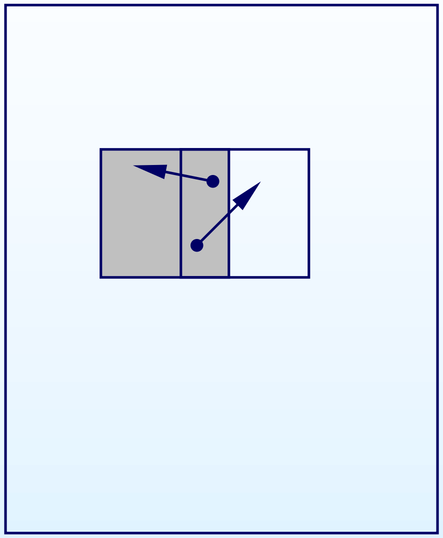
Water Model

PIV - Principle

- Tracer particles (negligible mass and momentum coupling) are injected into the flow
- Two closely spaced (time) snapshots are taken using a CCD camera
- Particle locations in two snapshots are correlated to obtain distance moved by particle
- Flow velocity at particle location = Distance/Time

PIV Methodology

- **Resolution**
  - CCD camera - DANTEC - Double Image 700 with 768x480 pixels
  - Number of pixels per interrogation area 16x16 – 64x64
  - Average of 32x32 used
  - 25% overlap used to capture particles near interrogation area edges
  - Maximum vectors per measurement area – 31x19



- **Time interval between snapsnots** varies from 0.2 – 1s
- **Time interval between correlated frames -** 100 μs
- **Seeding particle** – Aluminum – (30μm)
- **Correlation technique -** Autocorrelation

Modifications - Boundary conditions

- **Constant thickness assumed from top to bottom**
  - Thickness variation is not large enough to justify using more complicated CFD algorithm

- **Free water surface simplified to rigid free slip boundary**
  - Water level variations - spatially and temporally are small

- **Only half the domain is modeled**
  - Not enough evidence was available to assume large scale lower roll motions
  - A factor of two benefit is obtained in computational size

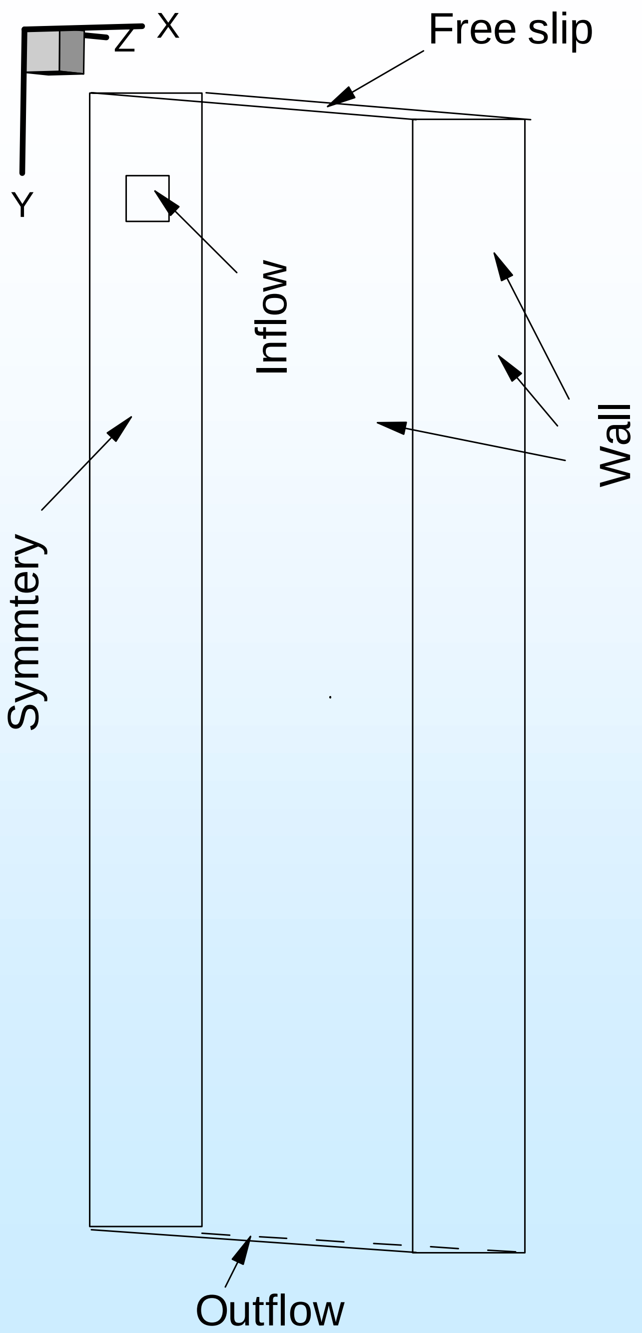
- **Inflow swirl is replaced by fully-developed turbulent flow from square duct**
  - Modeling swirl would require using an iterative solver (AMG) as opposed to a direct (FFT) solver which would make at least a factor of five difference in computational time
  - Helps check hypothesis of flow parameters being sensitive to inlet conditions
  - Square duct (LES) helps validate flow code

Modifications – Phenomena

- **Single phase modeling**
- Water Model can be run easily in single phase mode to validate model for single phase
- Single phase solution can be used as a easy start point for multiphase modeling

- **Solidification neglected**
- Present only deep down in the mold and close to the narrow face and can be neglected when modeling fluid flow
- Will be of significance when modeling heat transfer

- **Heat Transfer neglected**
- Secondary calculation if Boussinesq approximation is invoked
- Like multiphase can be solved for when fluid flow phenomenon have been captured



Computational Flow Simulation Domain

Large Eddy Simulation Model

Continuity $$\frac{\partial v_j}{\partial x_j} = 0$$

Momentum $$\frac{\partial}{\partial t}\rho v_i + \frac{\partial}{\partial x_j}\rho v_j v_i = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\mu_{eff}\left(\frac{\partial v_j}{\partial x_i} + \frac{\partial v_i}{\partial x_j}\right)$$

Smagorinsky Model

$$-\tau_{ij}\overline{S_{ij}} = \varepsilon = \nu_T \overline{S_{ij}}\,\overline{S_{ij}}\,;\nu_T \sim \ell q_{SGS}$$

$$q_{SGS}{}^2 \sim \ell^2 \overline{S_{ij}}\,\overline{S_{ij}}$$

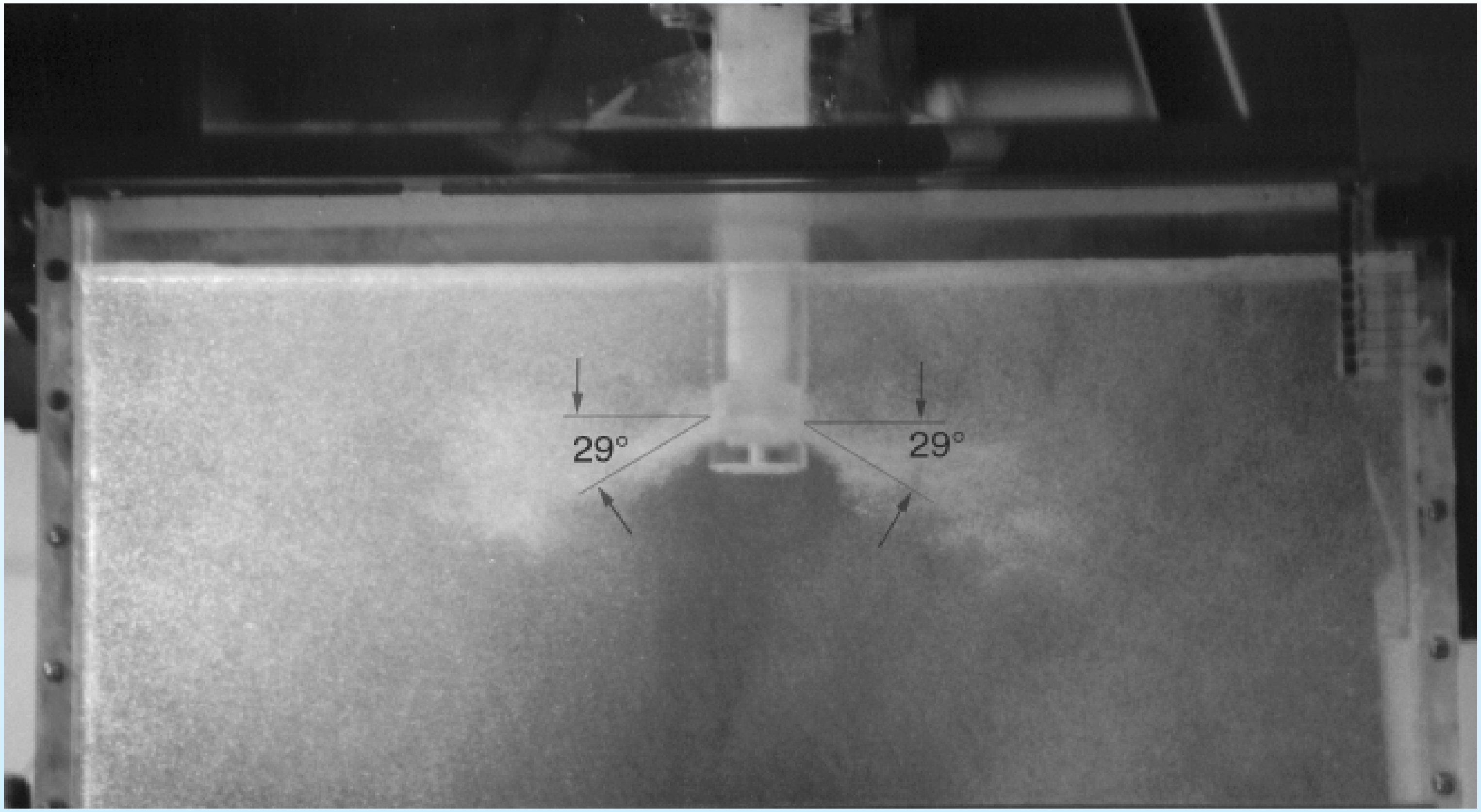
$$\nu_T = (C_S \Delta^2)\left|\overline{S}\right|$$

$$\left|\overline{S}\right| = \left(2\overline{S_{ij}}\,\overline{S_{ij}}\right)^{1/2}$$

Large Eddy Simulation Methodology

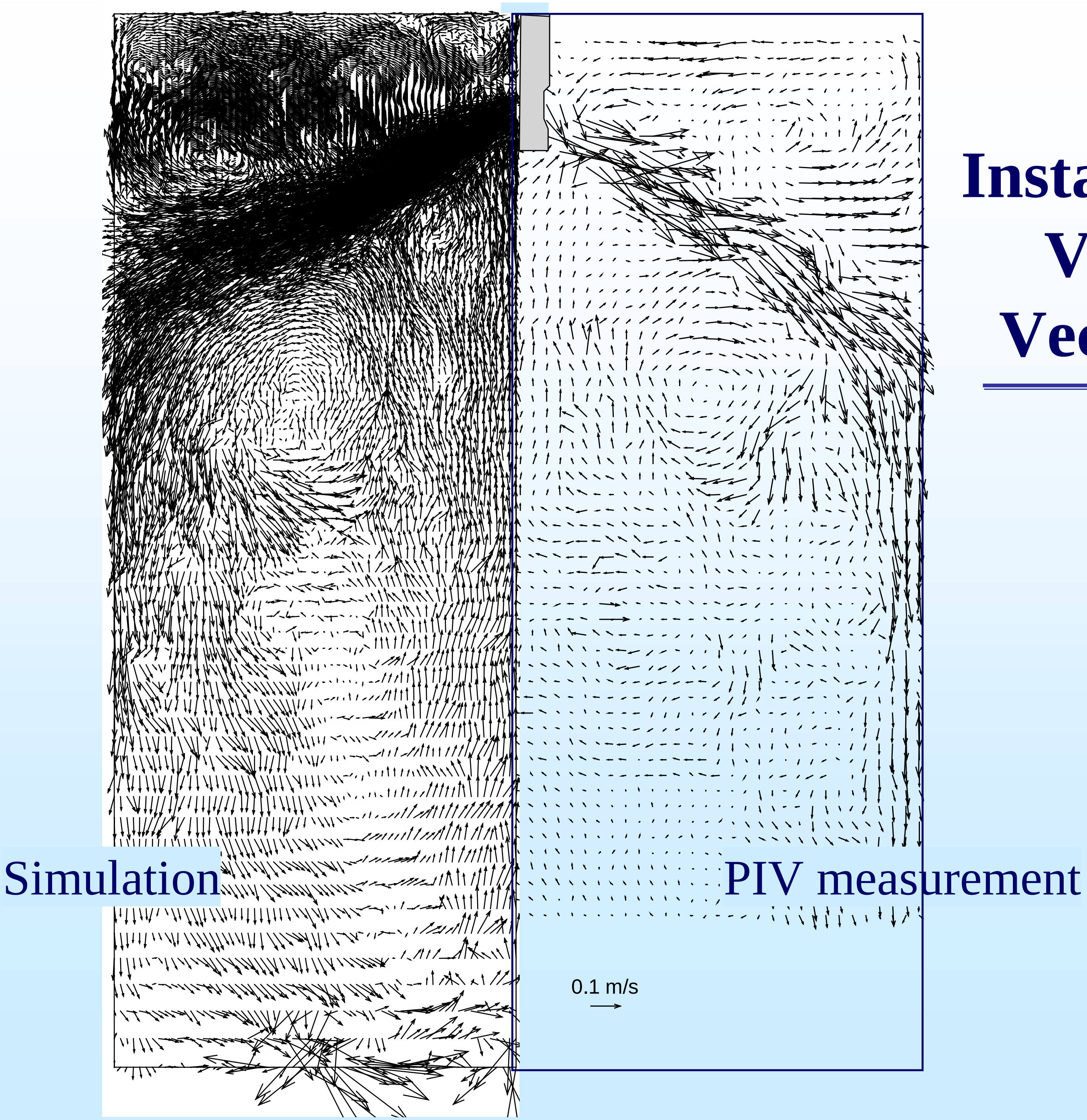
- **The equations are discretized using a fractional step procedure on a staggered grid**
- **A Second order accurate scheme in time and space is used**
- **The implicit diffusion terms** are solved for using Fast Fourier Transform (FFT) and Alternate Line Inversion (ALI)
- **The Pressure-Poisson equation** is solved using direct FFT
- **For parallelization 1-D domain decomposition with MPI (Message Passing Interface) is used**
- **Rectangular computational grid of 1.5 million nodes**
- **18 CPU secs per 0.001s time step or**
  13 days CPU time (Origin 2000) for 60s of flow simulation

Flow Pattern and Jet Angle

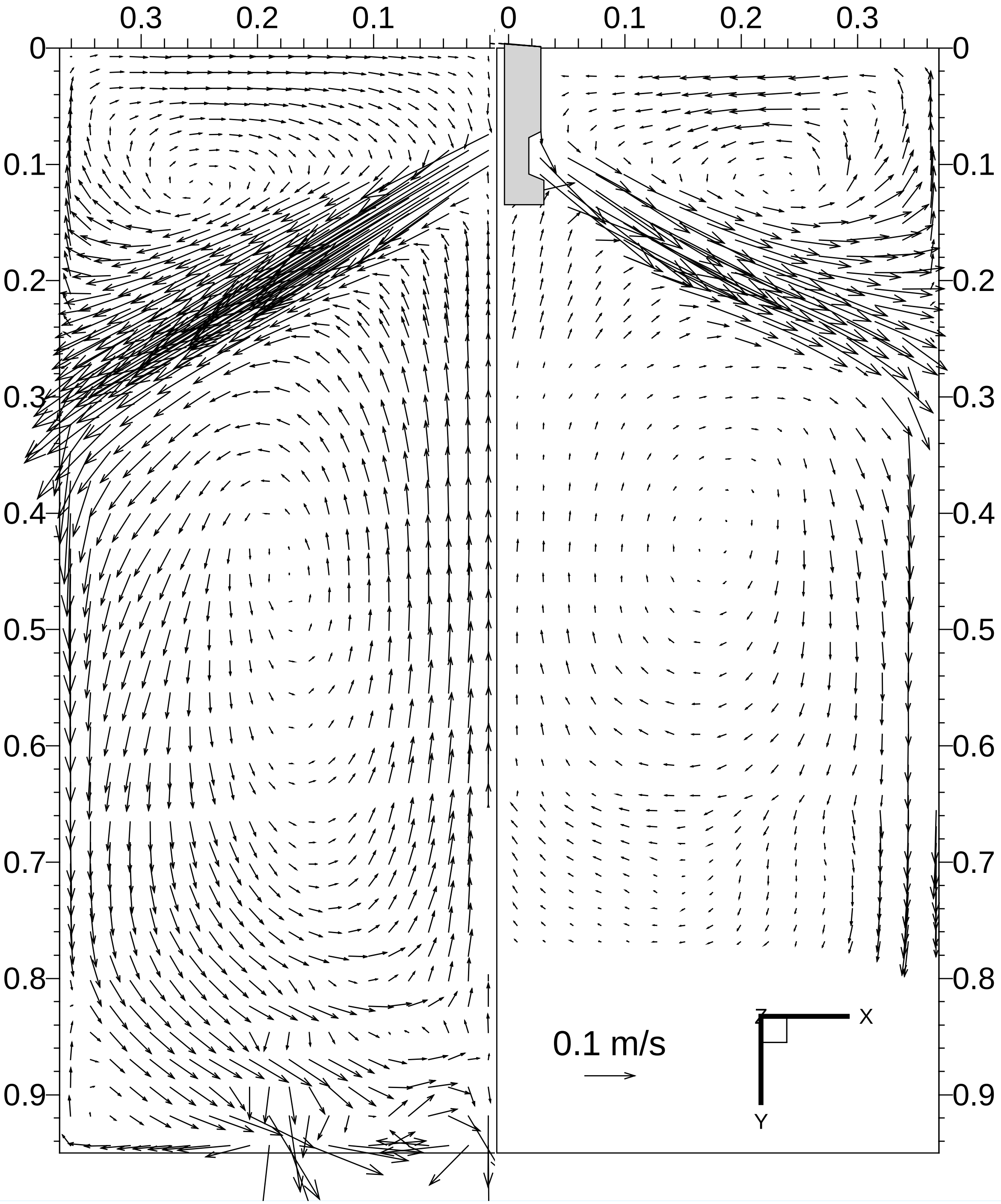


Asymmetry Between Jets

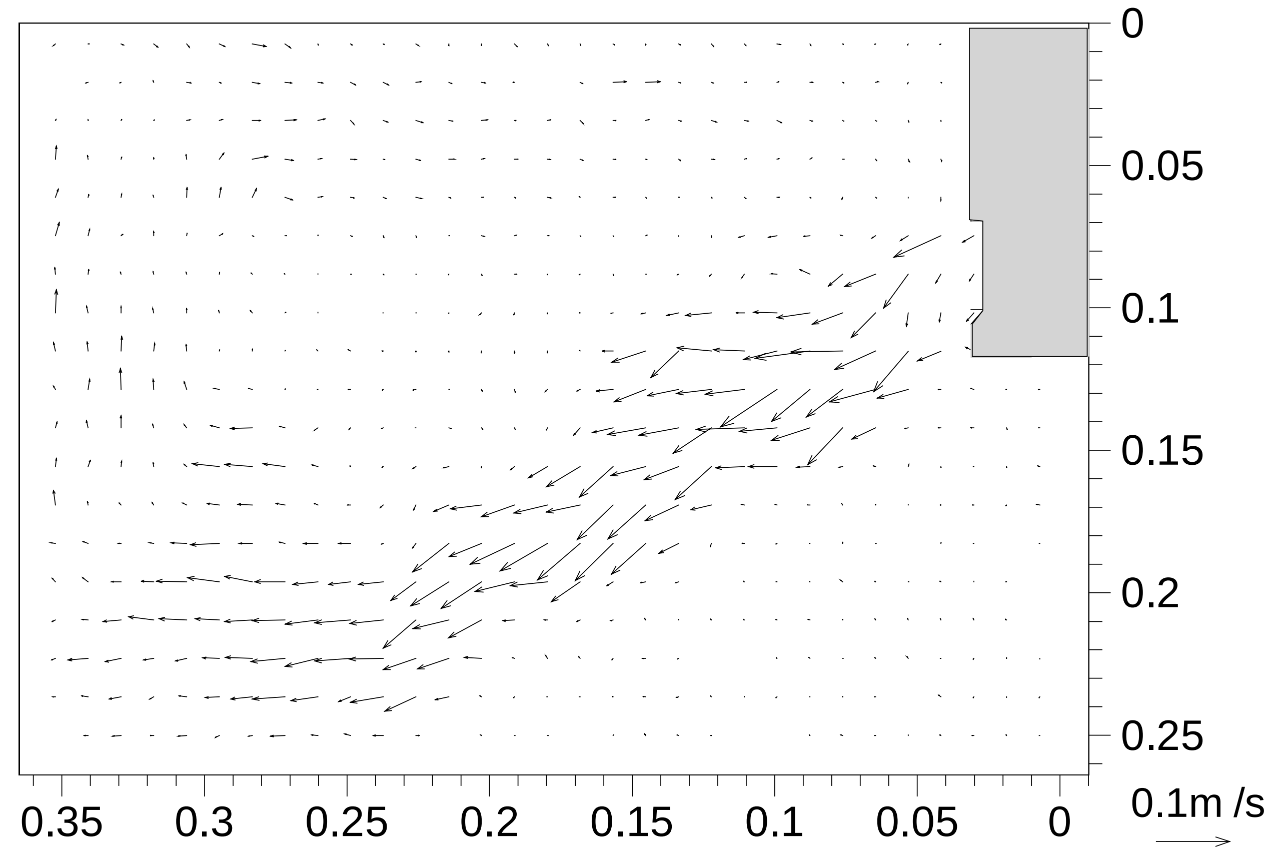
Instantaneous Velocity Vector Plot



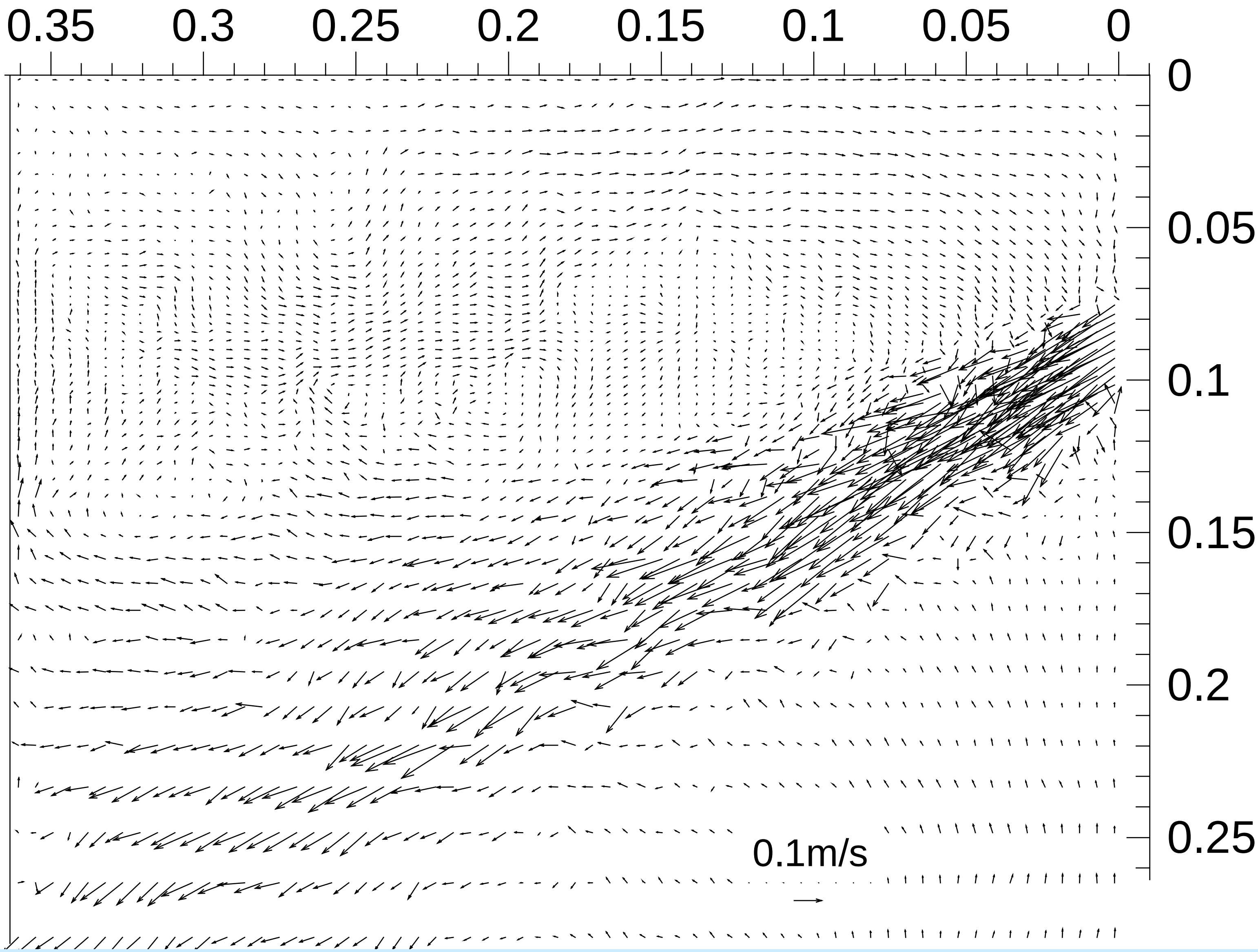
Time Average Comparison



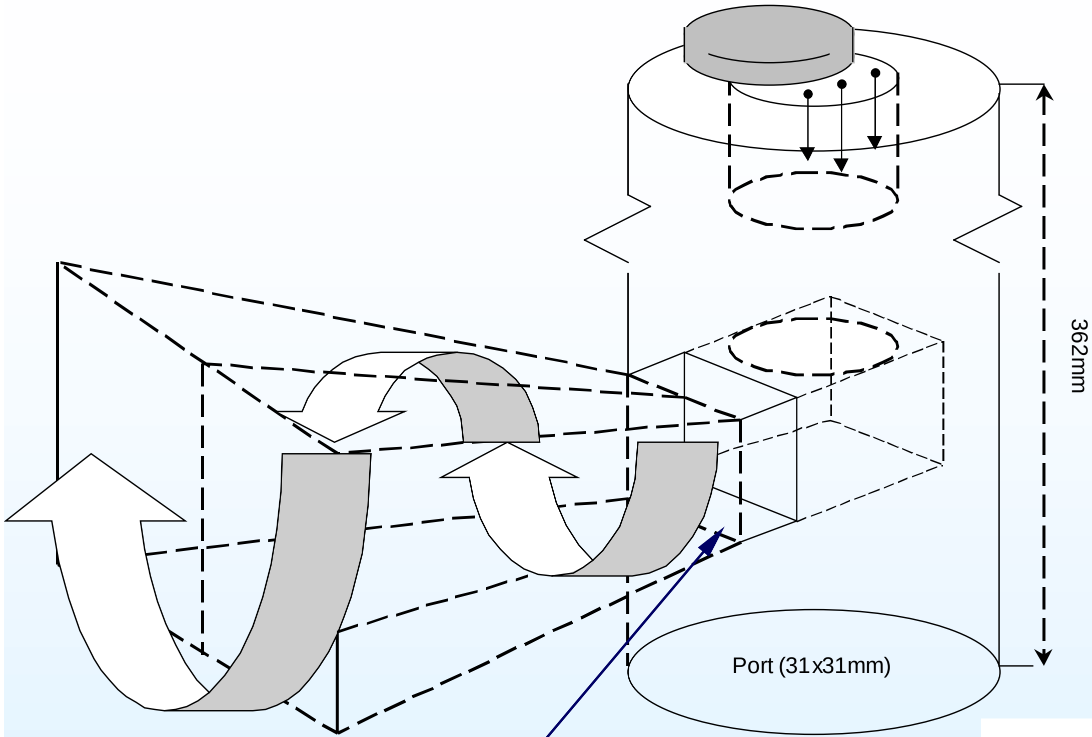
Staircase Effect in Experimental Jet

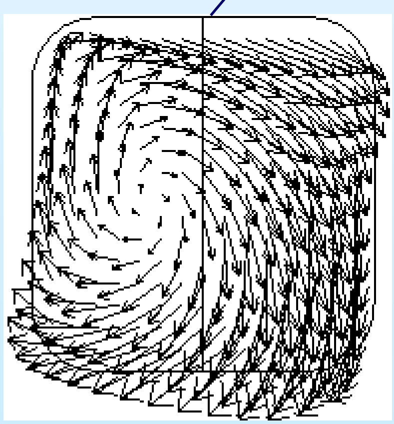


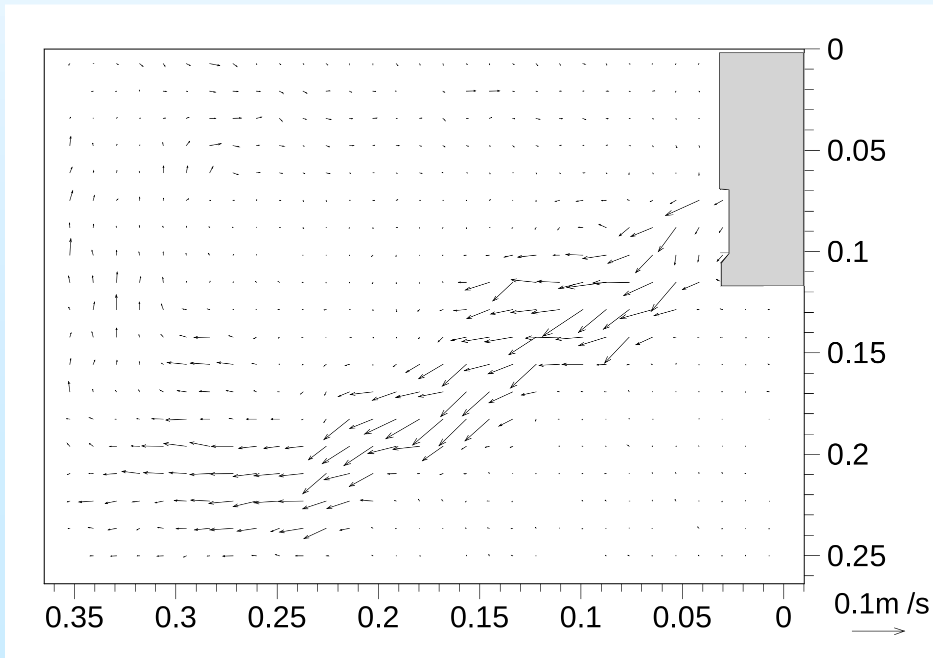
Simulation Jet



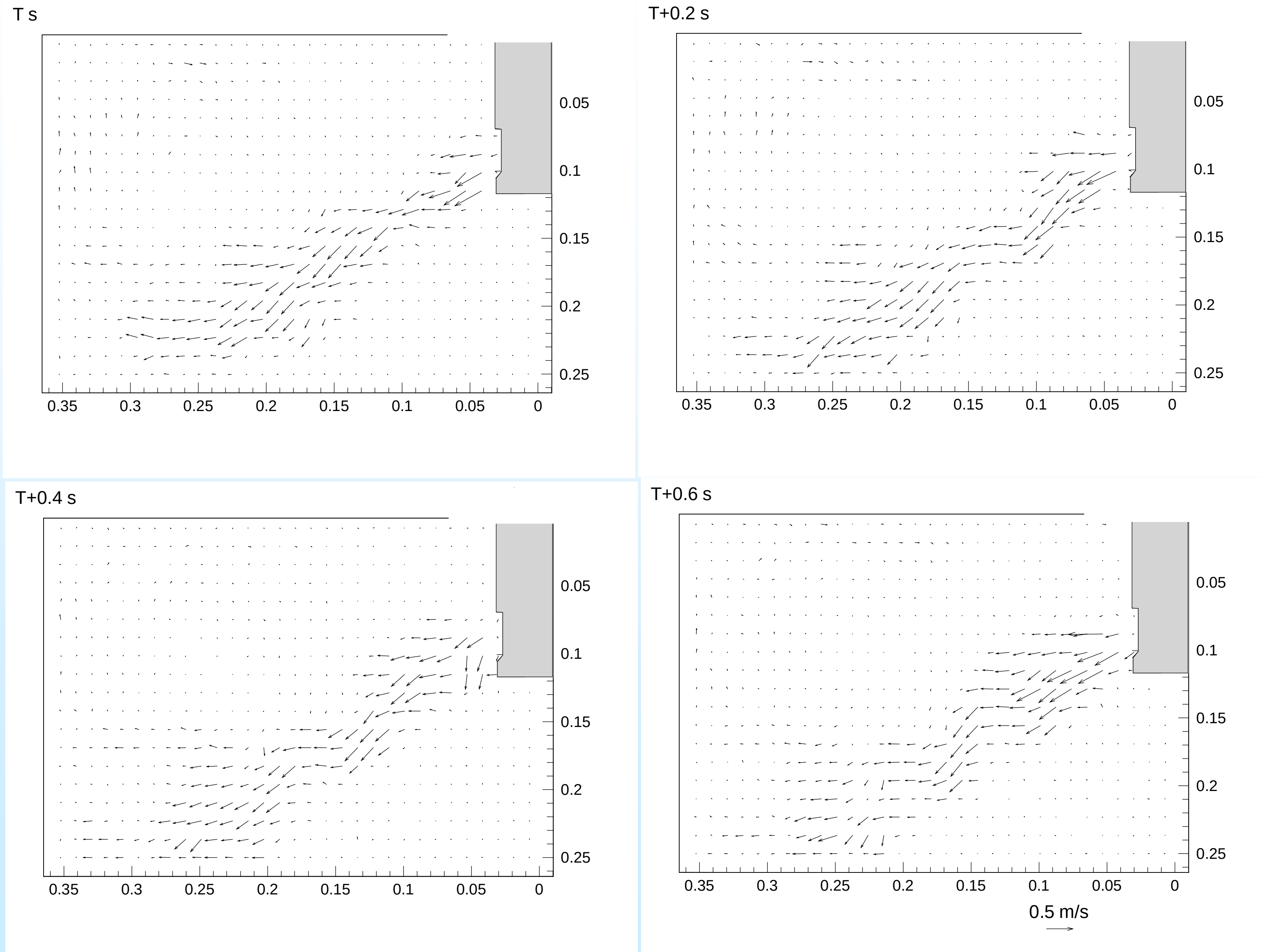
Swirl in Experimental Jet



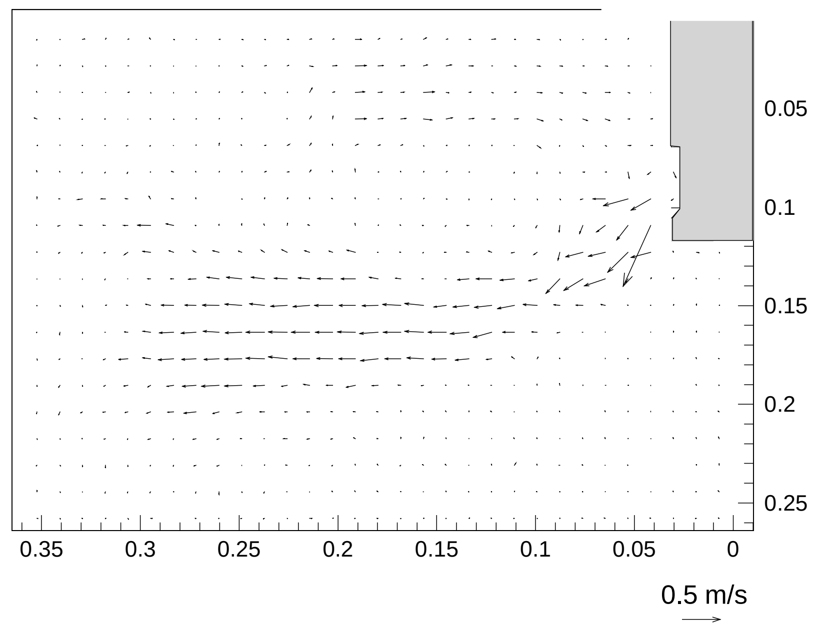




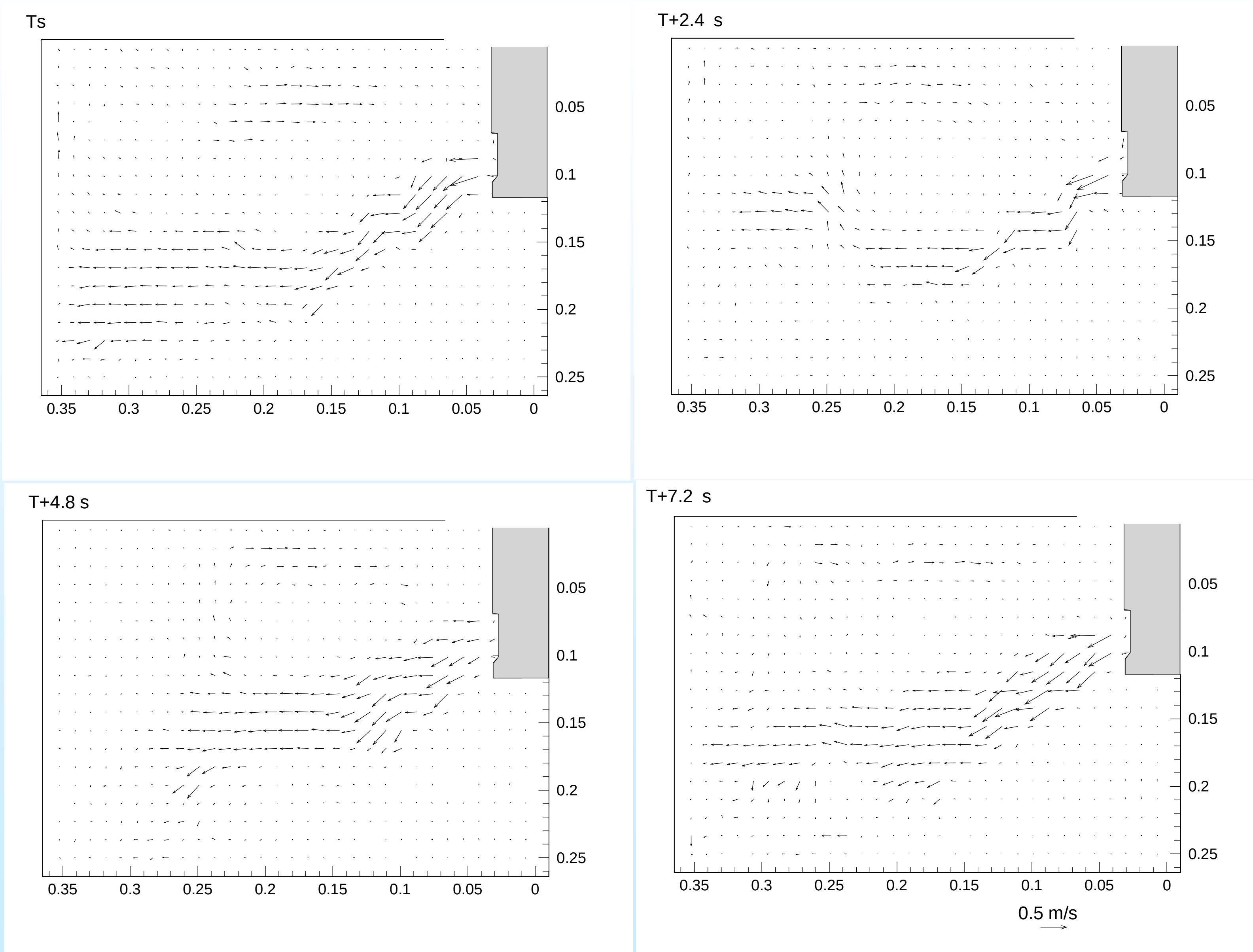
Staircase Effect and Time Scales



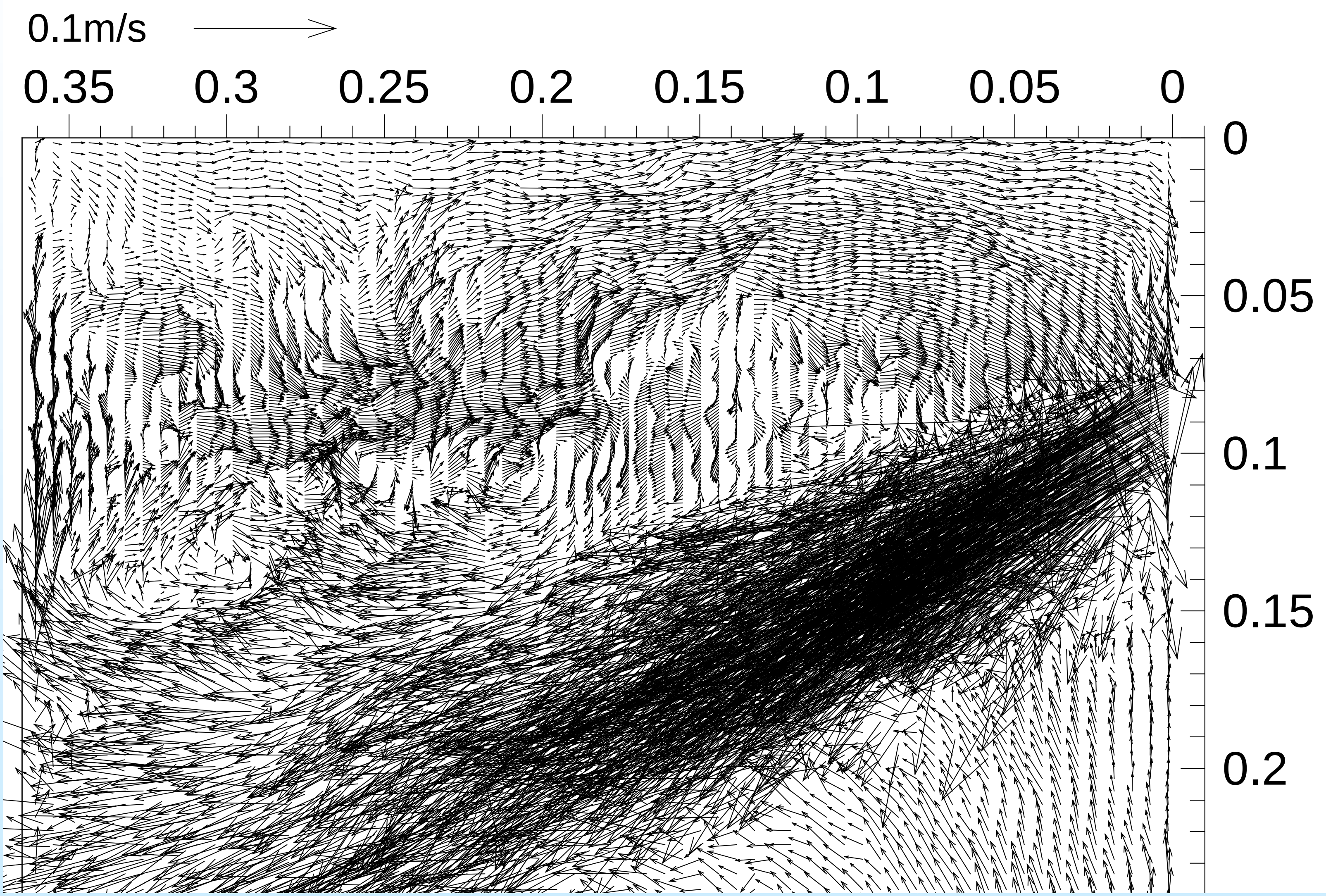
Shallow Penetrating Jet



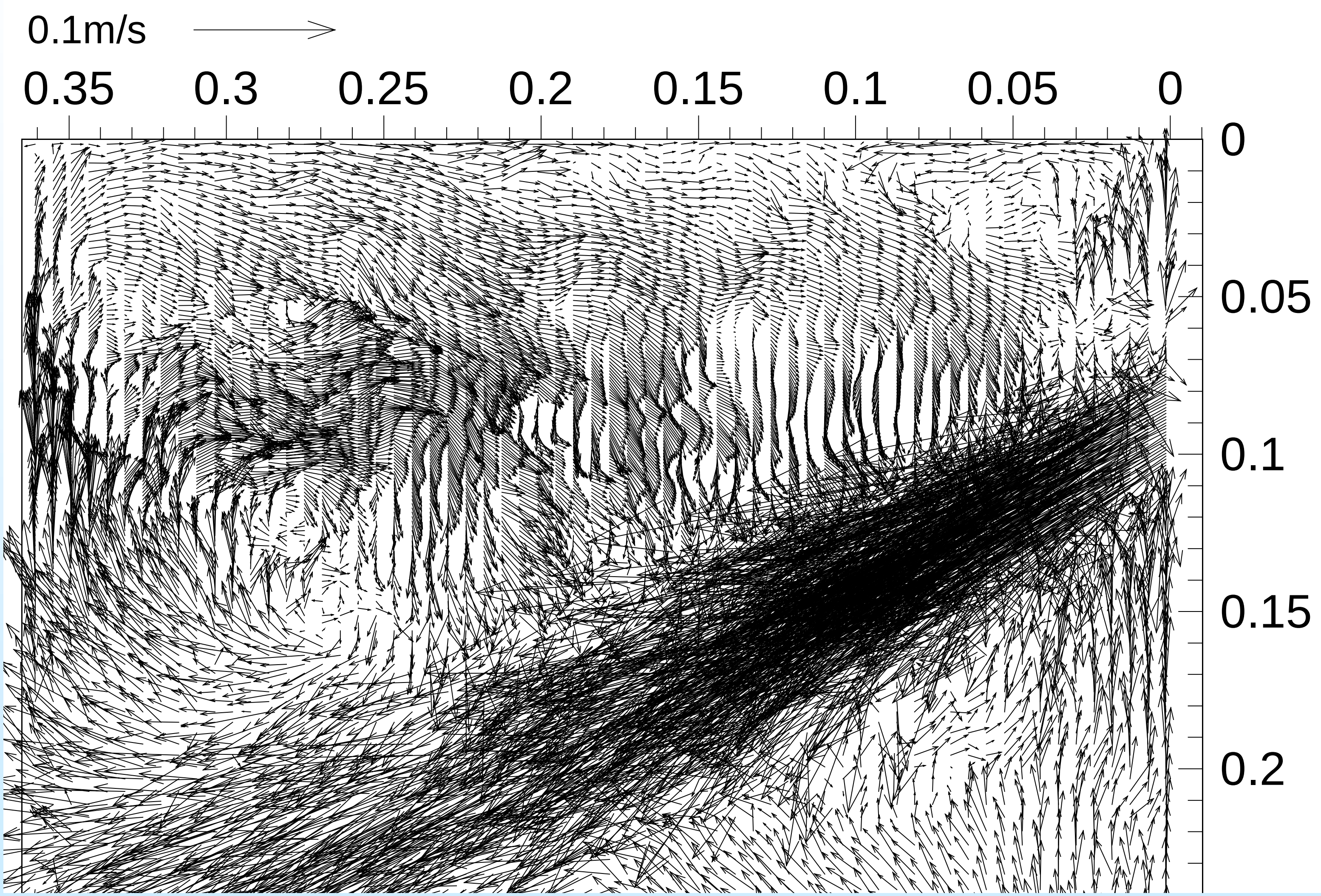
Large Time Scale Motion



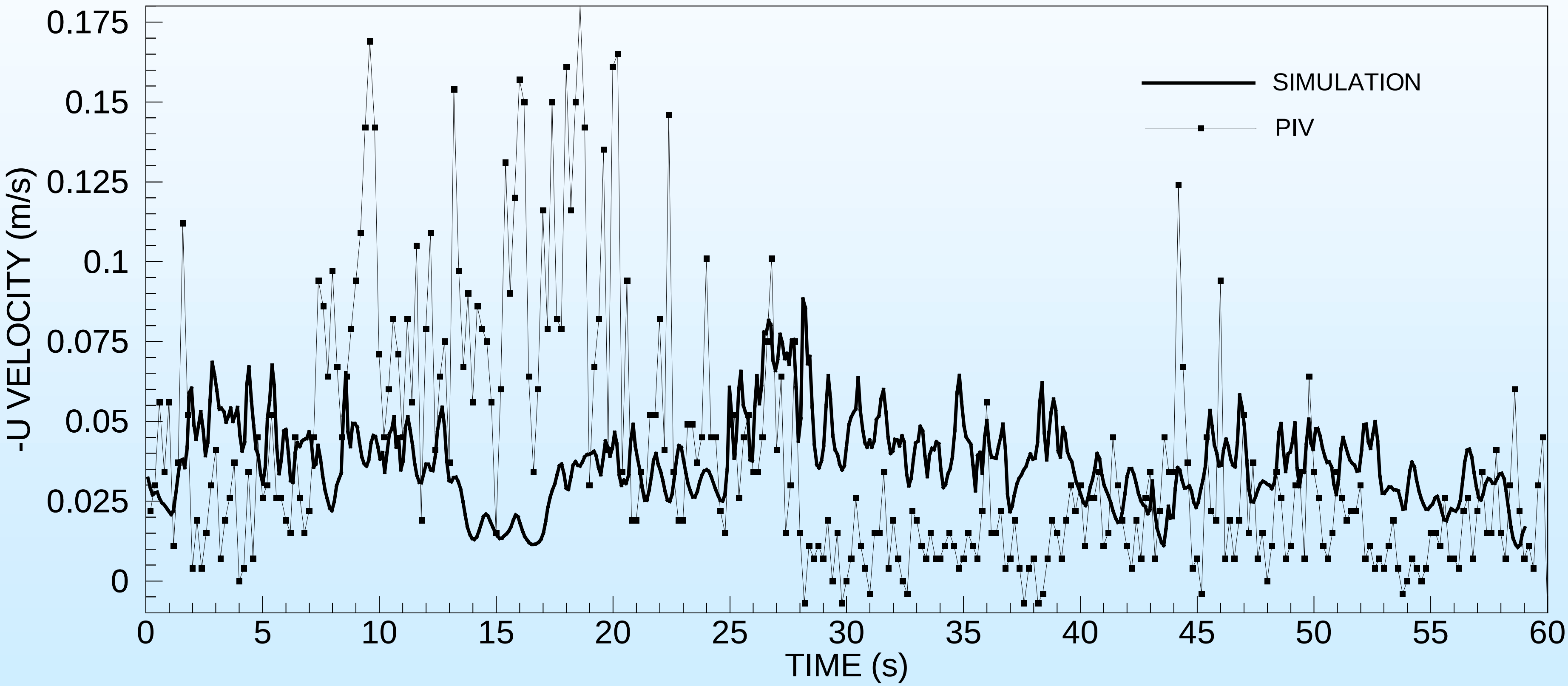
Upper Roll Structure

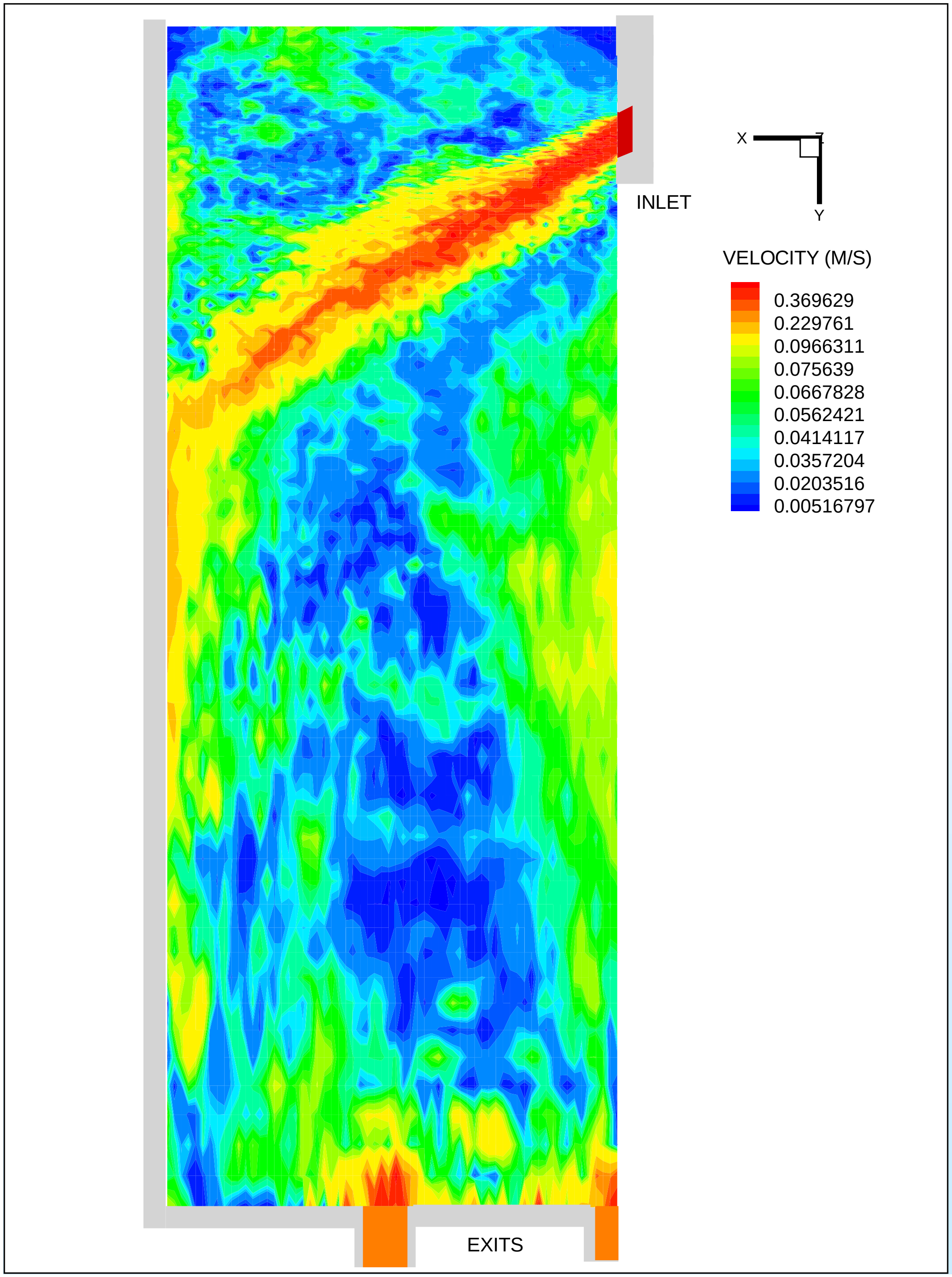


Upper Roll Structure



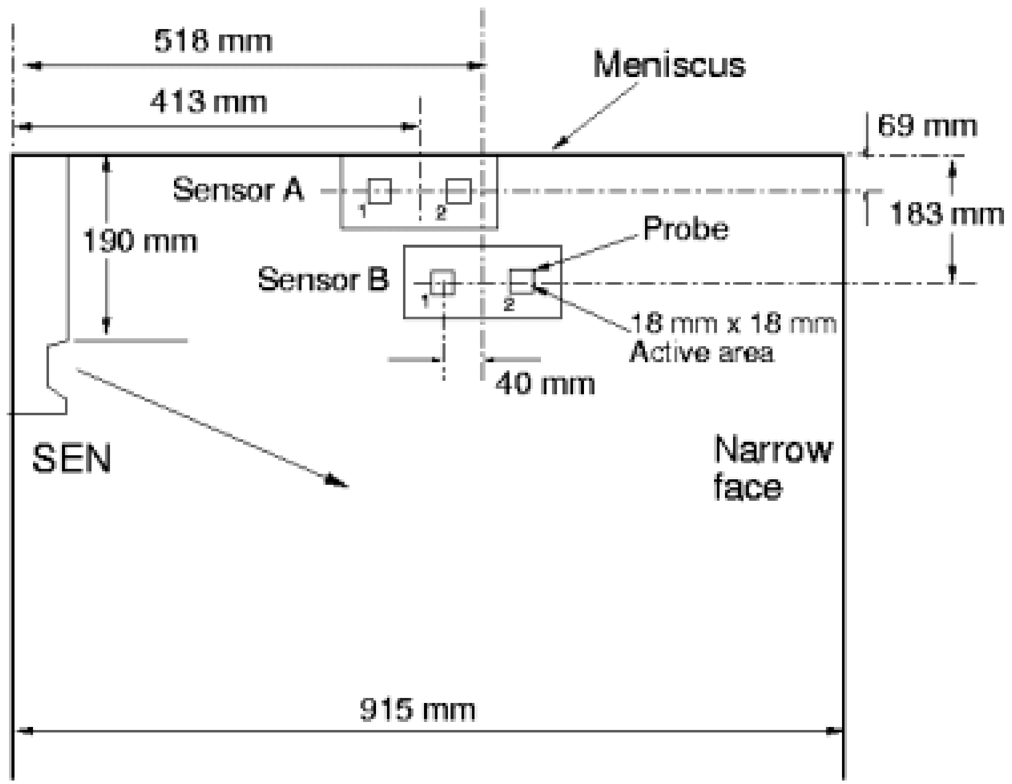
Velocity Time History Near Water Surface

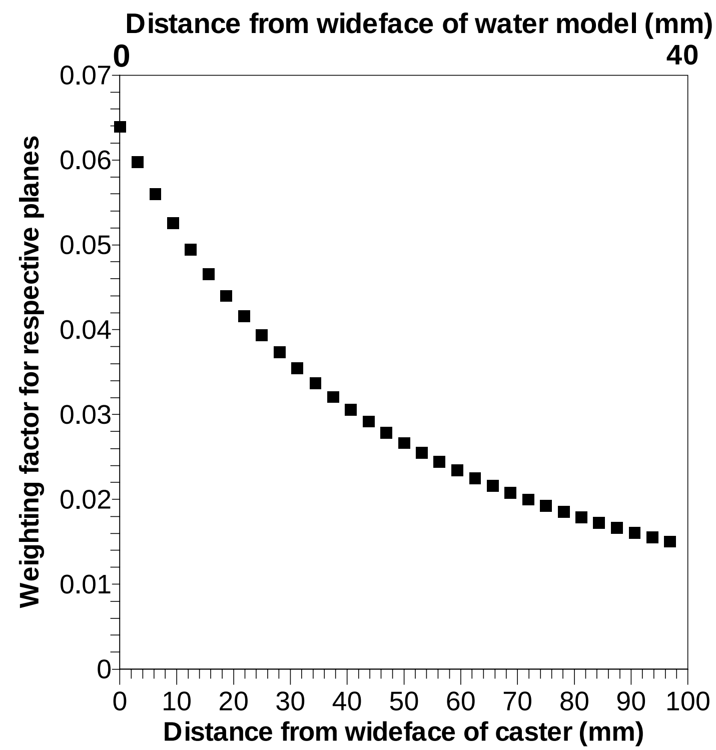


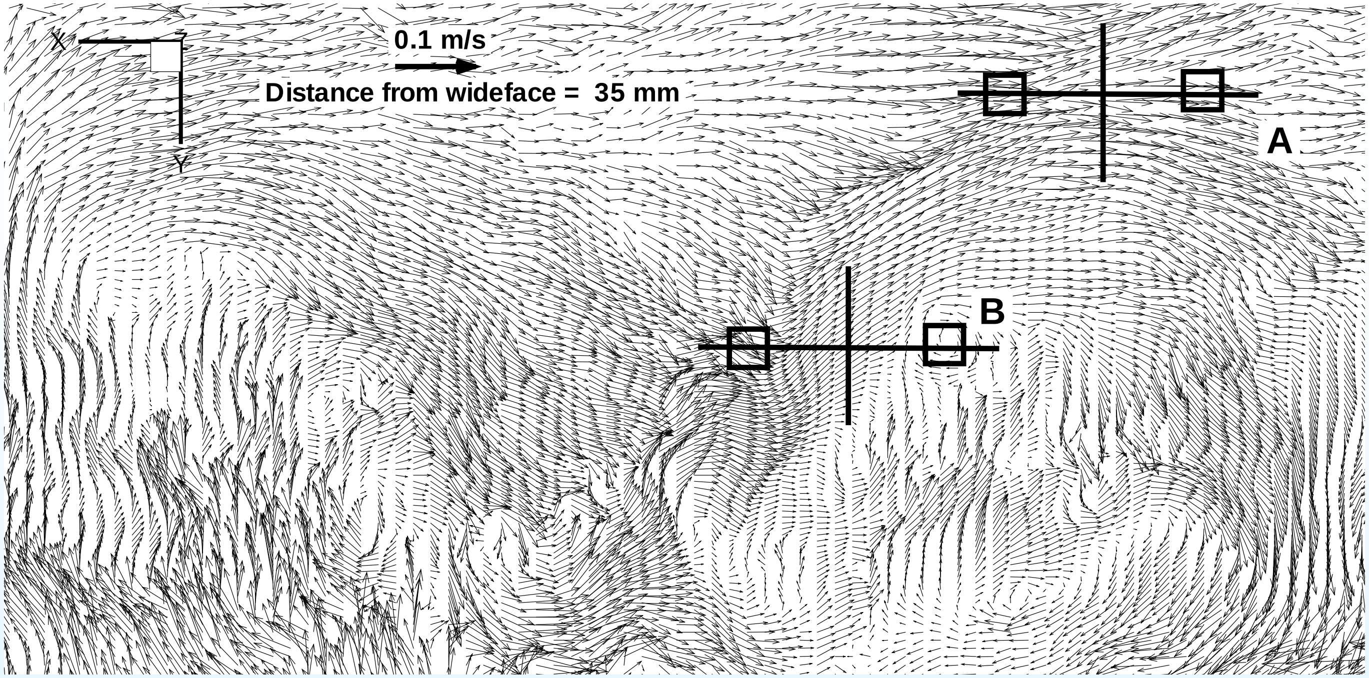


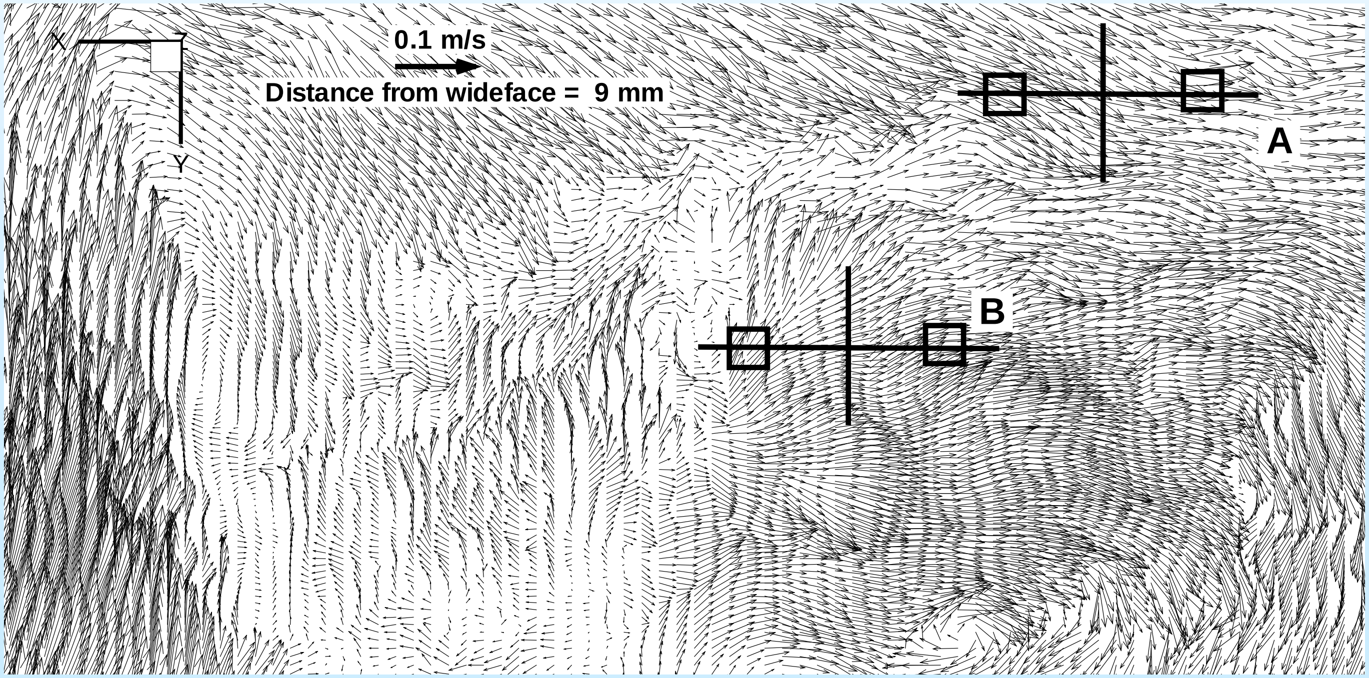
Simulation Jet - Structure and Spread

Sensor (MFC) on Caster

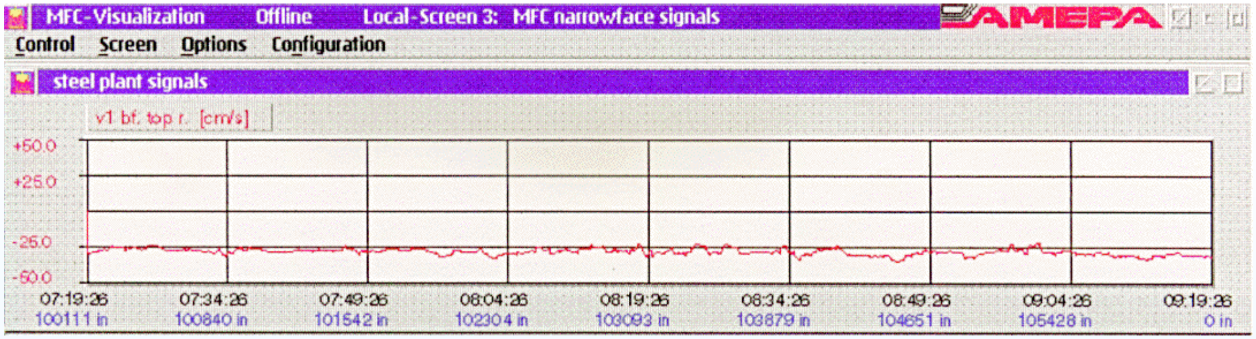


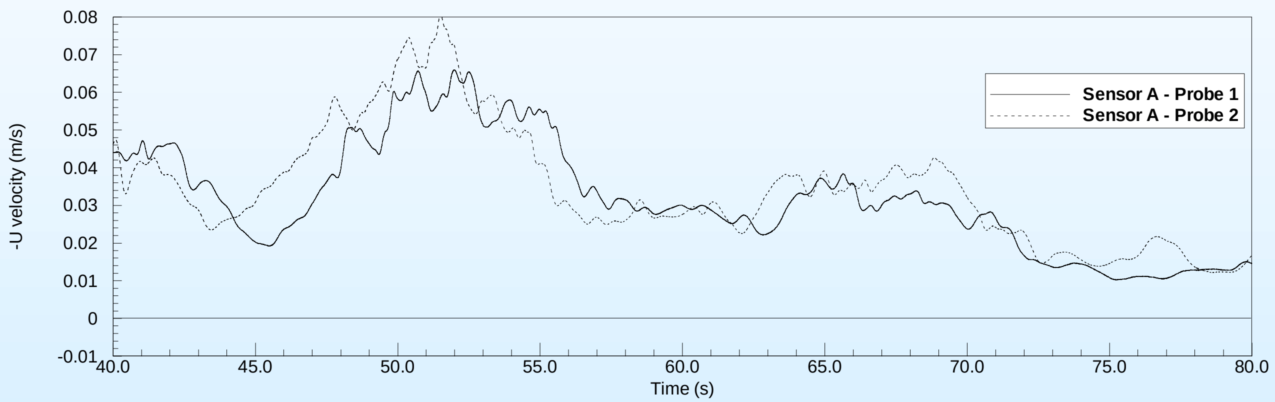


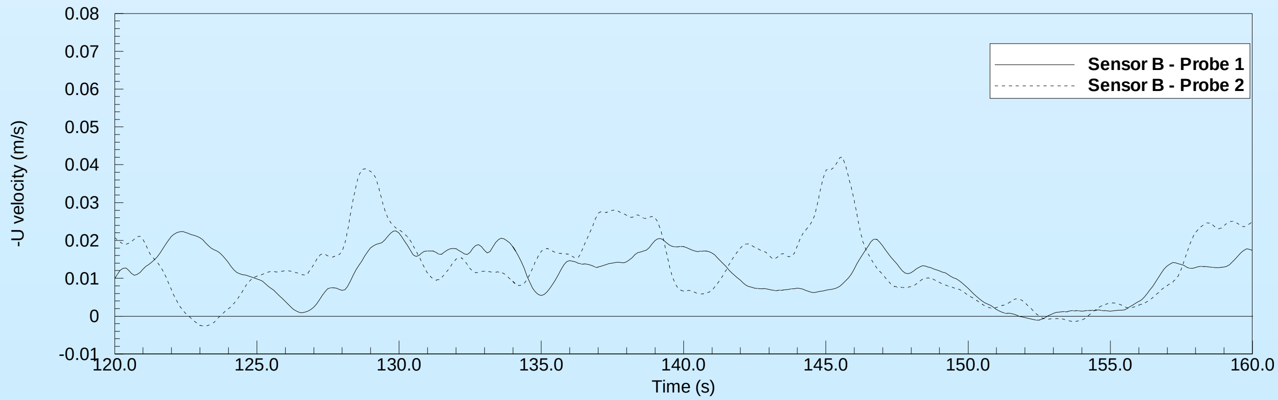




Instantaneous Flow Past Sensor

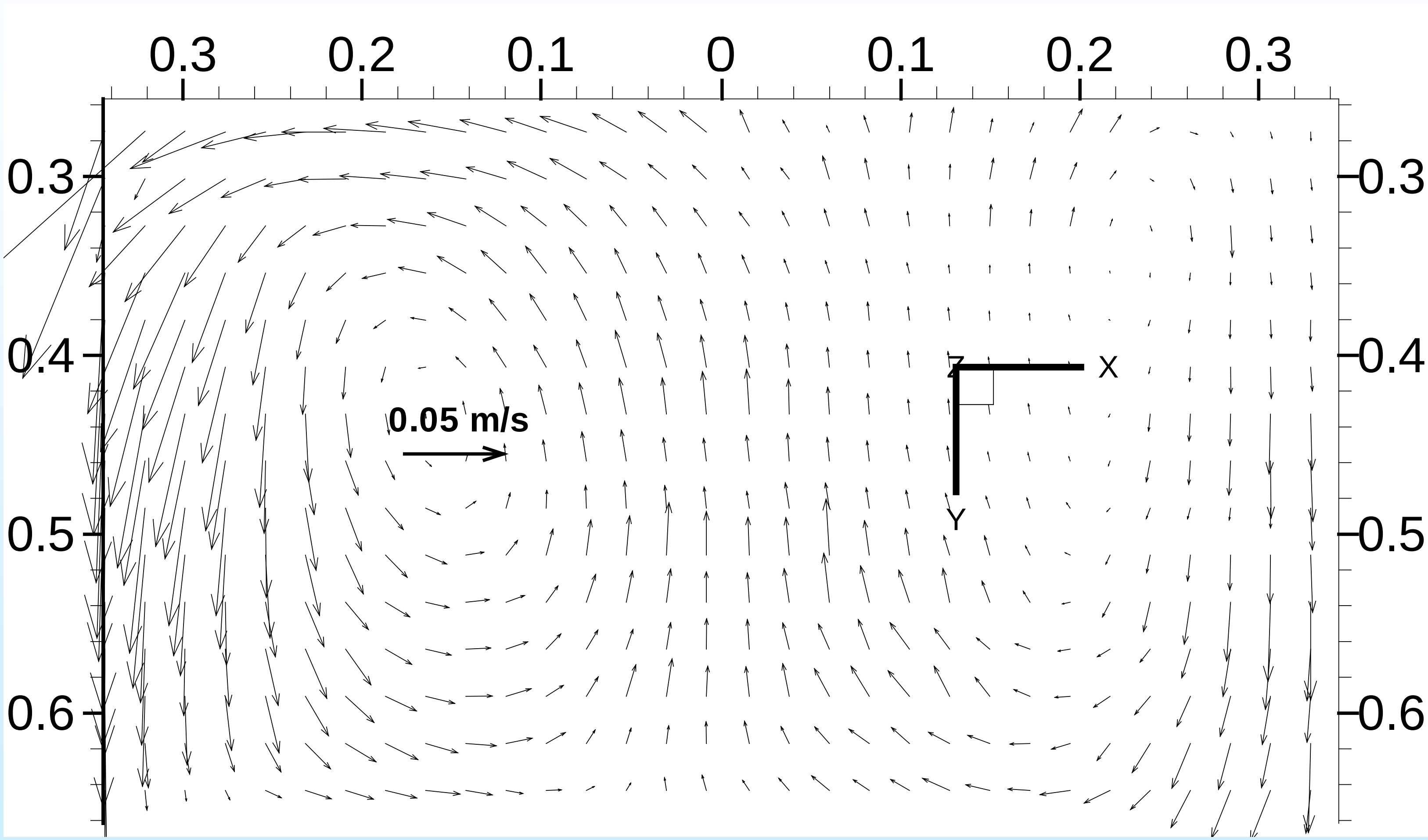




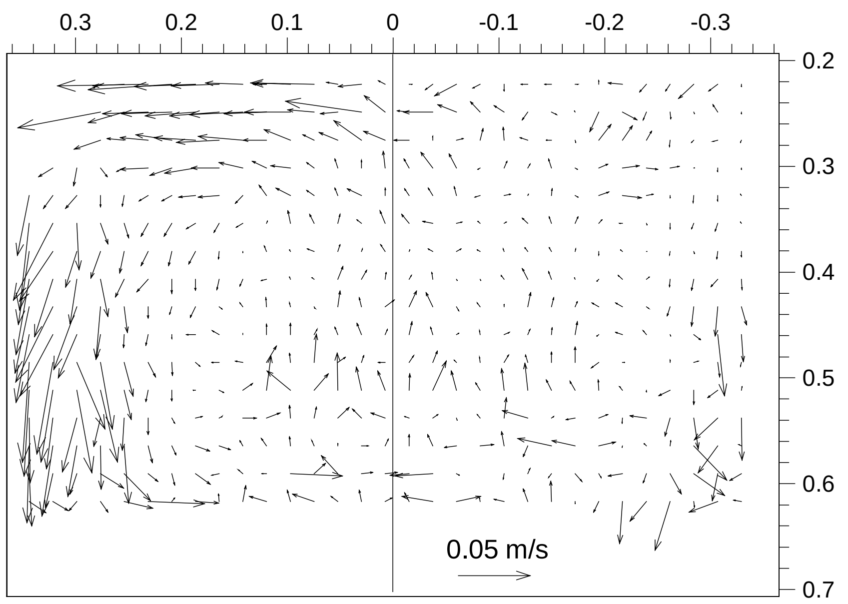


Qualitative Comparison of Sensor Outputs

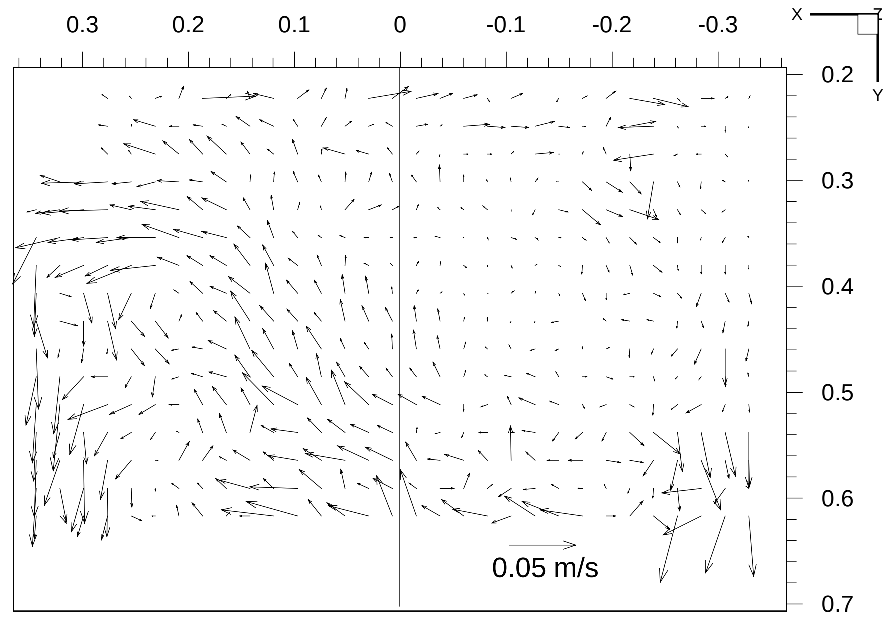
Asymmetry in Lower Rolls



Lower Roll Transients



Rolls Same Size



Right Roll Larger

Results - Summary

- **Jet** vector plots show a **staircase pattern** caused by the inlet swirl (PIV)
- The **jet** has many **different time scales** of motion (PIV)
- Velocity variation close to the water surface has two time scales (PIV and LES)
- **Upper roll alternates** between a single recirculation region and a set of distinct vortices (PIV and LES).
- **MFC sensor** should be placed **close to the water surface** for accurate interpretation
- **Lower rolls** are **significantly asymmetric** (PIV)
- Lower rolls go through a **repeating sequence** of flow structures (PIV)

\- The short-circuit structure is significant to **particle entrapment**

Table of Conditions

No.	Property	Water model	Simulation
1	Length of the model	0.950m	0.956m
2	Thickness of model	Varies from 0.095m at the top to 0.065m	Constant 0.08m
3	Port opening	0.031 x 0.031m	0.031 x 0.031m
4	Top surface	Free surface	Free slip boundary
5	Flow rate through each port	$3.528 \times 10^{-4}\ m^3/s$ (5.6 gal/min)	$3.528 \times 10^{-4}\ m^3/s$ (5.6 gal/min)
6	Average inlet velocity	0.4239m/s	0.4239m/s
7	Average jet inlet angle	30^o	30^o
8	Distance of top of port outlet from top surface (submergence depth)	0.075 m (Varies with time)	0.07207m
9	Outlet	1.5 35mm diameter outlets along each half of the bottom	1.5 35mm square outlets at the bottom
10	Fluid used	Water	Water
11	Kinematic viscosity	$1 \times 10^{-6}\ m^2/s$	$1 \times 10^{-6}\ m^2/s$
12	Gas flow rate (cubic ft / hr)	0.0	0.0