

Mercury Flow through a Long Curved Pipe

Wenhai Li & Foluso Ladeinde

Department of Mechanical Engineering

Stony Brook University

Summary

The flow of mercury in a long, curved pipe is simulated in this task, via the computational fluid dynamics (CFD) approach. The Navier-Stokes equations governing mass conservation and the conservation of momentum are solved in a curvilinear coordinate system using a model that includes the effects of the viscosity of mercury, turbulence in the pipe, pressure force, and pipe friction. The equations are solved on a Linux cluster and results for velocity and pressure distributions are provided. Azimuthal modes are significant, leading to a non-symmetric velocity and pressure distribution.

Problem Description

The problem is that of mercury flow in a curved pipe, with specific interest on the velocity profile at the exit of the pipe. This information is provided in Figure 13 in this document. It is pointed out that, for comparison purposes, the results for a straight pipe will be symmetrical, without any azimuthal variations in the distributions of the velocity and pressure. The physical model is provided in Figure 1, and is consistent with the experimental setup at BNL for the muon collider accelerator studies.

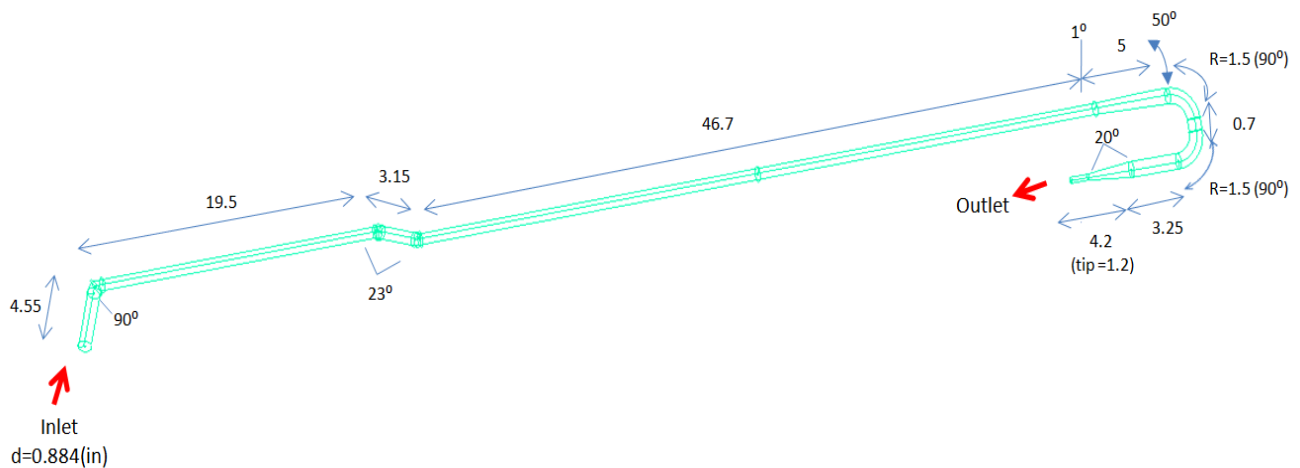


Figure 1 Physical layout of the pipe flow problem

The fluid (mercury) is assumed to have the following properties at 25 at 25°C:

| | |
|--|------------------------|
| Density (kg/m^3) | 13456 |
| Viscosity ($kg/(m \cdot s)$) | 1.526×10^{-3} |
| Dynamic Viscosity (m^2/s) | 1.134×10^{-7} |
| Thermal Conductivity ($W/(m \cdot K)$) | 8.69 |
| Specific Heat ($J/(kg \cdot K)$) | 0.139×10^{-3} |
| Prandtl Number | 0.025 |

The variable values are used:

| | |
|----------------------------|-------|
| Velocity (m/s) | 3.4 |
| Static Pressure (bar) | 18.5 |
| Dynamic Pressure (bar) | 0.7 |
| Diameter (in) | 0.884 |

Mesh

The grid that is used to solve the computational fluid dynamics (CFD) problem using AEROFLO commercial code developed by TTC Technologies, Inc. is shown in Figure 2 below.

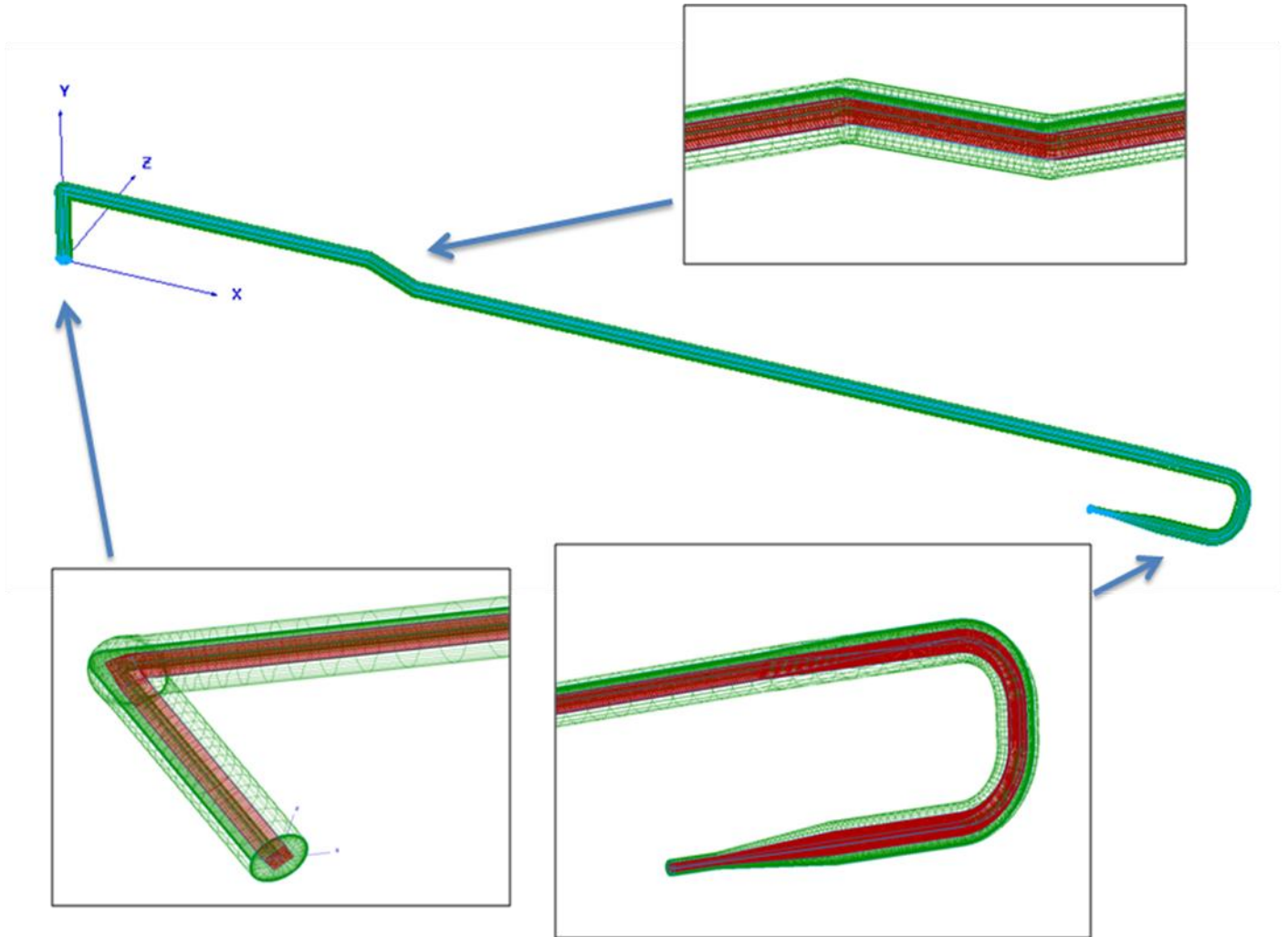


Figure 2 Computational grid used for the CFD solution

The grid consists of two blocks, is in PLOT3D format, with the following properties:

| Mesh File | Block # | Grid Size | Description |
|--------------------|---------|---------------|--|
| <i>pipe_01.grd</i> | Block 1 | 301 × 36 × 25 | Pipe mesh in concentric cylinder form |
| <i>pipe_02.grd</i> | Block 2 | 301 × 20 × 20 | Overset mesh to cover the centerline of pipe |

The lengths in the problem are non-dimensionalized using the pipe diameter at the inlet as the scale. Because we have a non-Cartesian system, a generalized curvilinear coordinate system is used for the numerical solution procedure. As opposed to the Cartesian (x, y, z) coordinate system, we use the (ξ, η, ζ) system in generalized coordinate system. For the “Block 1”, ξ is flow direction; η is radial direction; ζ is azimuthal direction.

Nondimensionalization at Inlet

$$Re = \frac{\rho_i u_i D}{\mu} = \frac{13456 \left(\frac{\text{kg}}{\text{m}^3}\right) * 3.4 \left(\frac{\text{m}}{\text{s}}\right) * (0.884 * 0.0254)(\text{m})}{1.526 * 10^{-3} \left(\frac{\text{kg}}{\text{m} \cdot \text{s}}\right)} = 6.73172 * 10^5$$

Ma = 0.3 (Incompressible flow)

$$\rho^* = \frac{\rho}{\rho_i} = 1$$

$$u^* = \frac{u}{u_i} = 0$$

$$v^* = \frac{v}{v_i} = 1$$

$$w^* = \frac{w}{w_i} = 0$$

$$p^* = \frac{p_i}{\rho_i u_i^2} = \frac{18.5(\text{bar})}{2 * 0.7(\text{bar})} = 13.214$$

It is pointed out that, because we are using a compressible code, the Mach number for the flow has to be specified, for which the value of 0.3 is used. It is well known that the results for this case are essentially those of incompressible flows, and those presented in this work should be valid.

CFD Simulation Parameters

The parameters used for the CFD exercise are shown in the table below.

| Overall Flow Conditions | |
|--------------------------------------|-----------------------------------|
| Mach No. | 0.3 |
| Reynolds No. | 6,731,72 |
| Viscosity | Turbulence (Spalart-Allmaras) |
| Numerical Schemes | |
| Spatial Scheme | MUSCL |
| Time Scheme | BW2, $\Delta t = 0.001$ |
| Boundary Conditions (Block 1) | |
| I=1: Inlet | $\rho=1; u=0; v=1; w=0; p=13.214$ |
| I=301: Outlet | Neumann for ρ, u, v, w, p |
| J=1: | Overset with Block 2 |

| | |
|--------------------------------------|-----------------------------------|
| J=36: | Solid wall |
| K=1, K=25: | Periodic |
| Boundary Conditions (Block 1) | |
| I=1: Inlet | $\rho=1; u=0; v=1; w=0; p=13.214$ |
| I=301: Outlet | Neumann for ρ, u, v, w, p |
| J=1, J=20, K=1, K=20: | Overset with Block 1 |
| Initial Conditions | |
| $\rho=1; u=1; v=0; p=13.214$ | |

Results:

(a) Pipe Cross-Section Velocity Magnitude Profiles (Non-Dimensionalized Value)

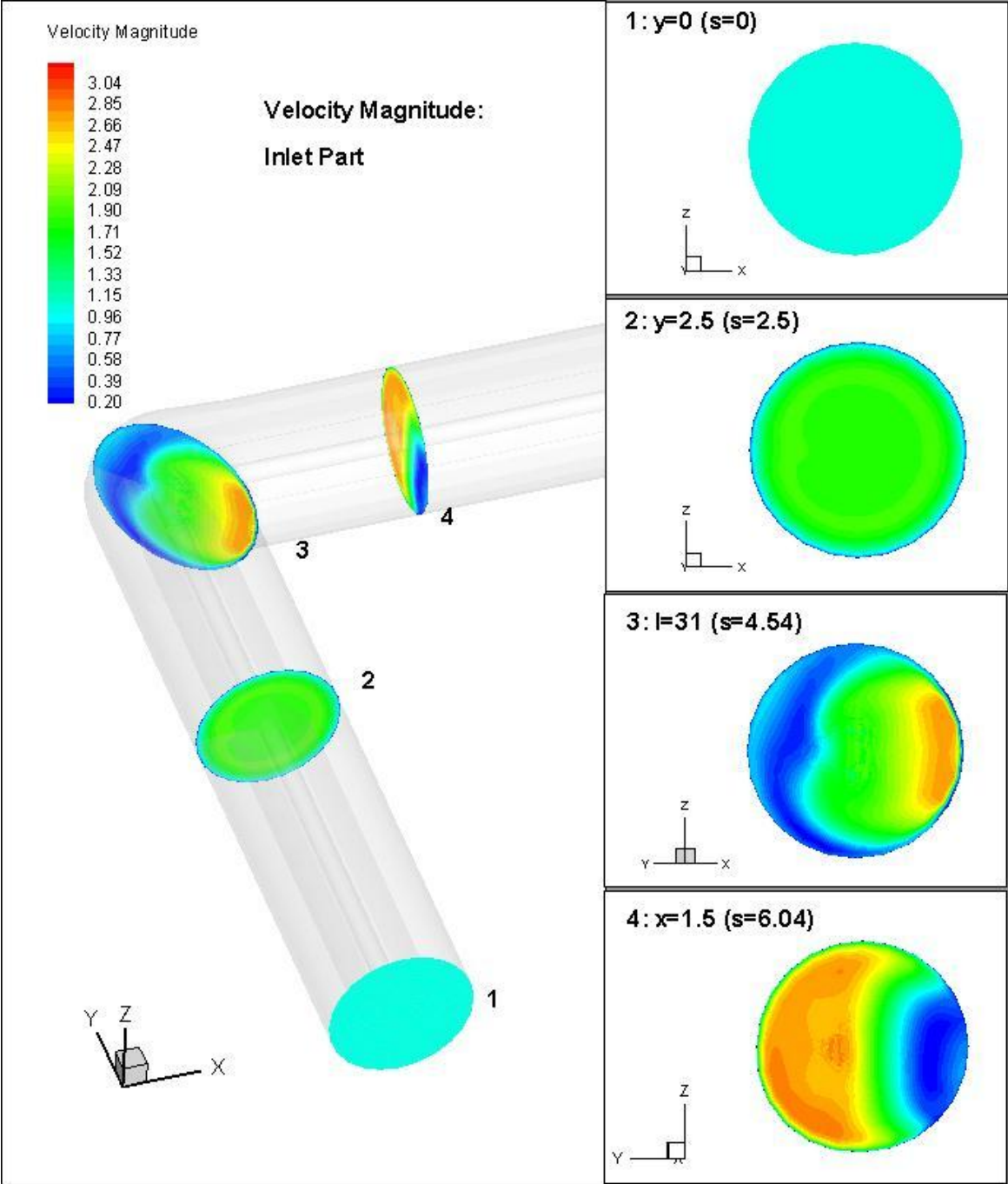


Figure 3 Velocity distribution at various cross sections along the pipe (vicinity of inlet)

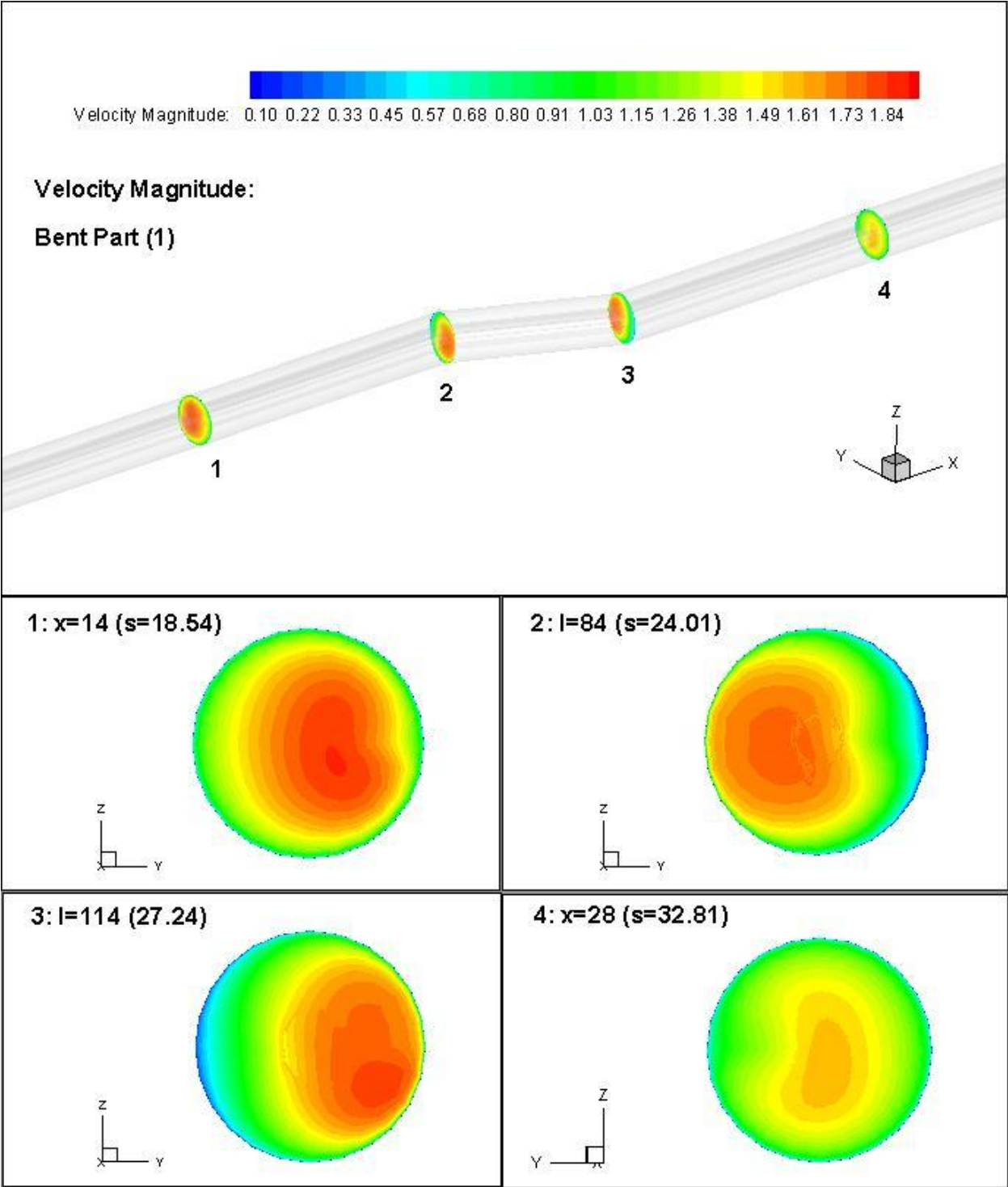


Figure 4 Velocity distribution at various cross sections along the pipe (at the first bend)

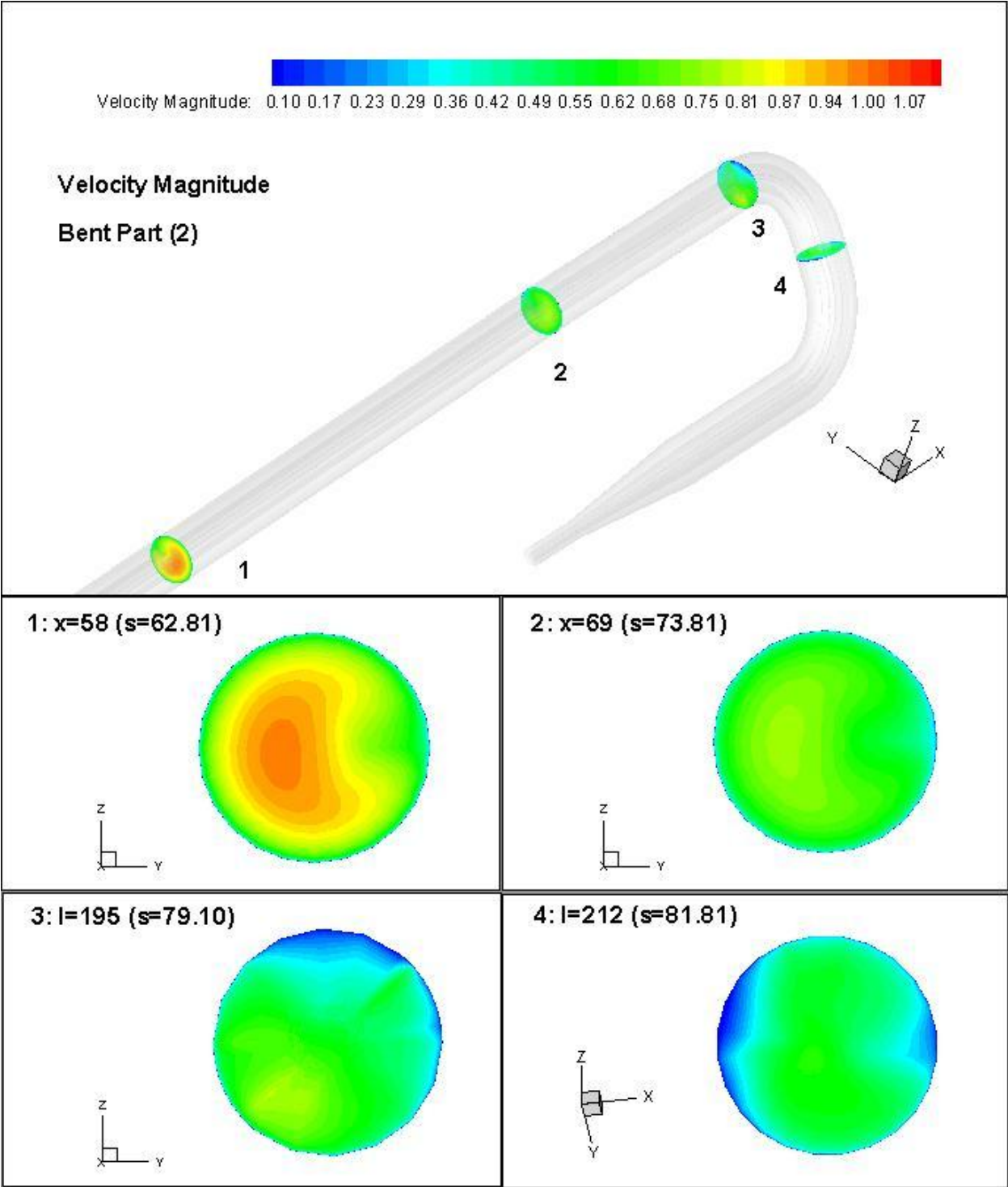


Figure 5 Velocity distribution at various cross sections along the pipe (at the second bend)

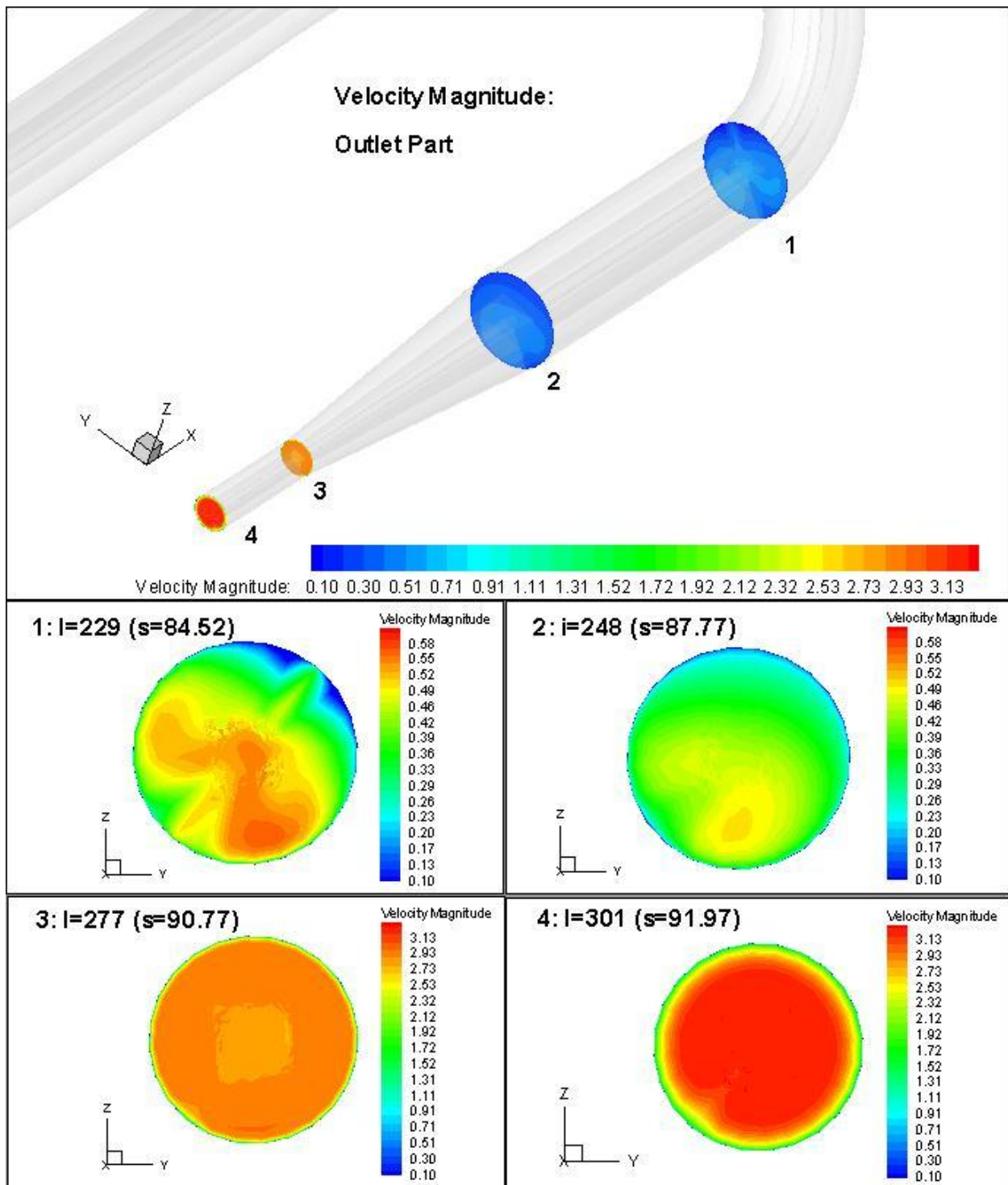


Figure 6 Velocity distribution at various cross sections along the pipe (outlet region)

(b) Pipe Cross-Section Pressure Profiles (Non-Dimensionalized Value)

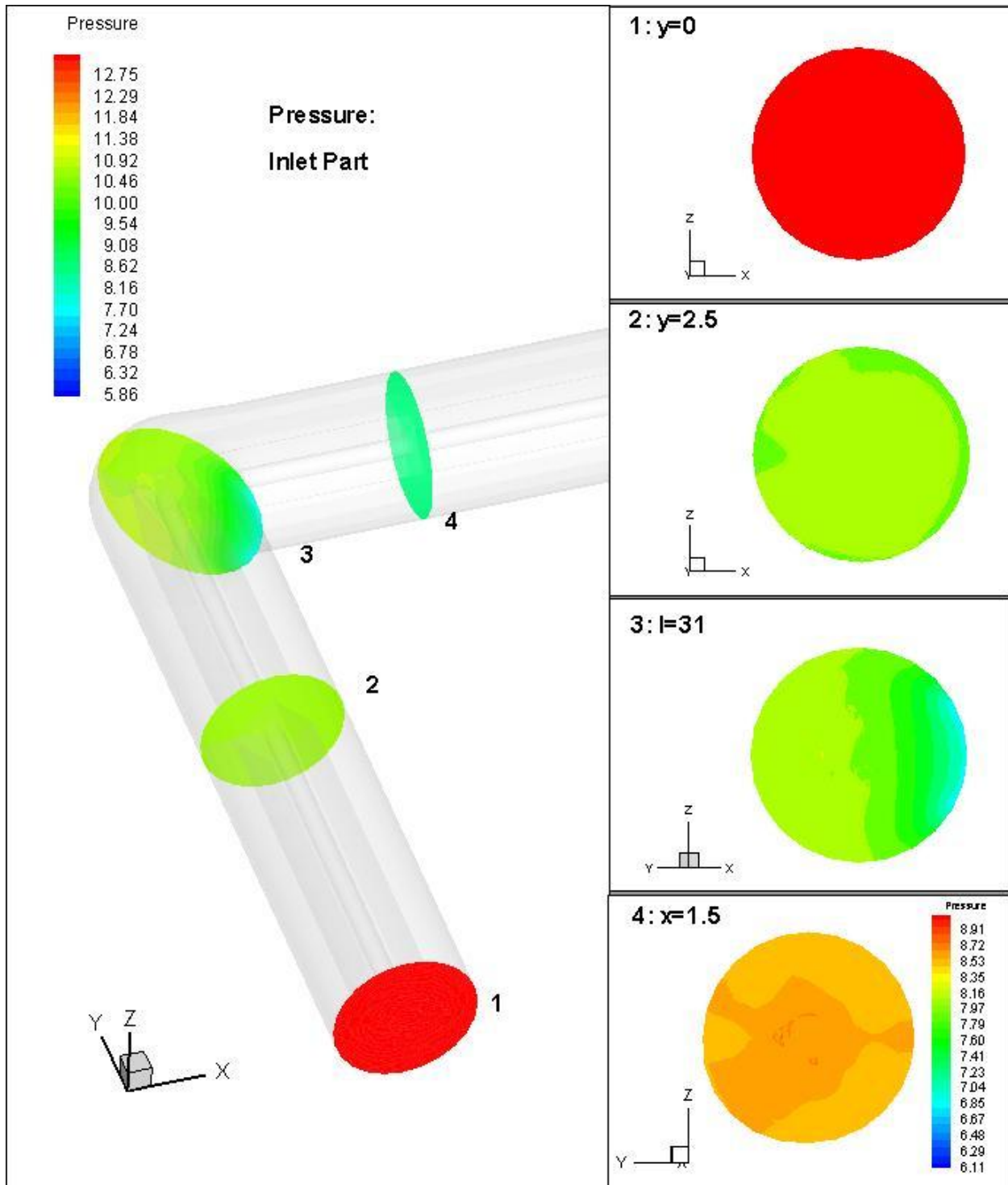


Figure 7 Pressure distribution at various cross sections along the pipe (vicinity of inlet)

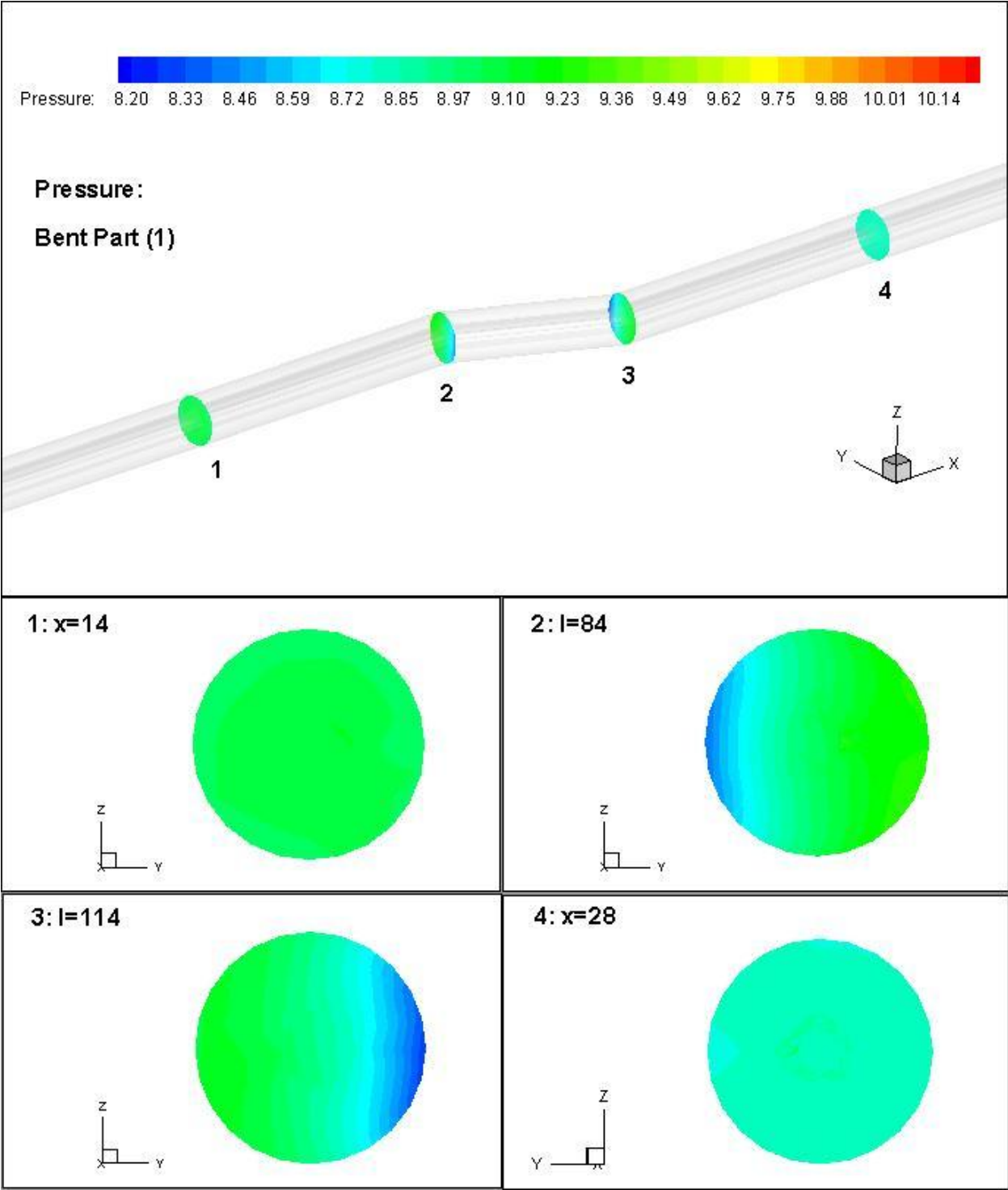


Figure 8 Pressure distribution at various cross sections along the pipe (vicinity of the first bend)

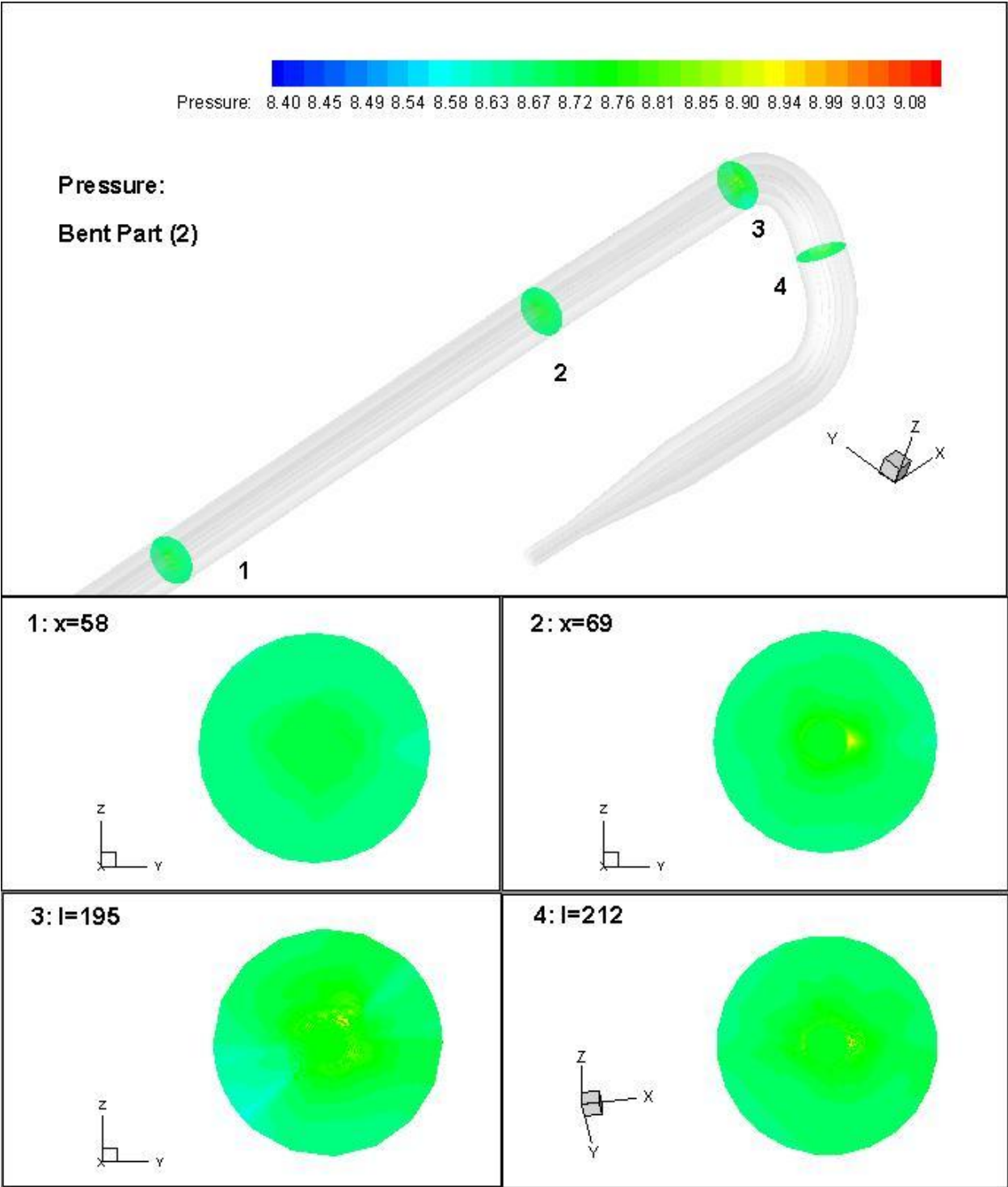


Figure 9 Pressure distribution at various cross sections along the pipe (vicinity of the second bend)

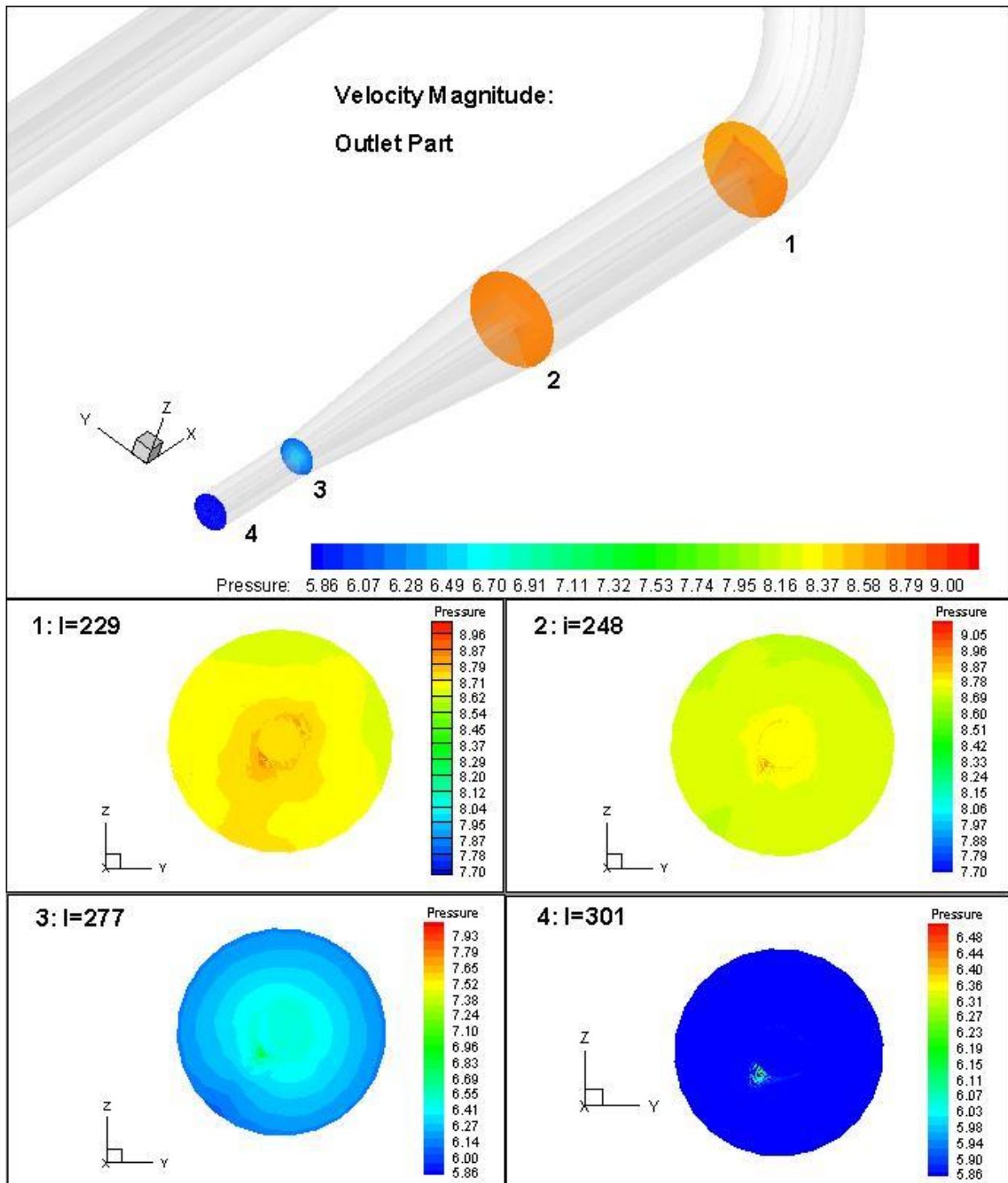


Figure 10 Pressure distribution at various cross sections along the pipe (at the flow exit region)

(c) Outlet Profiles

Average Velocity: -10.5866 m/s

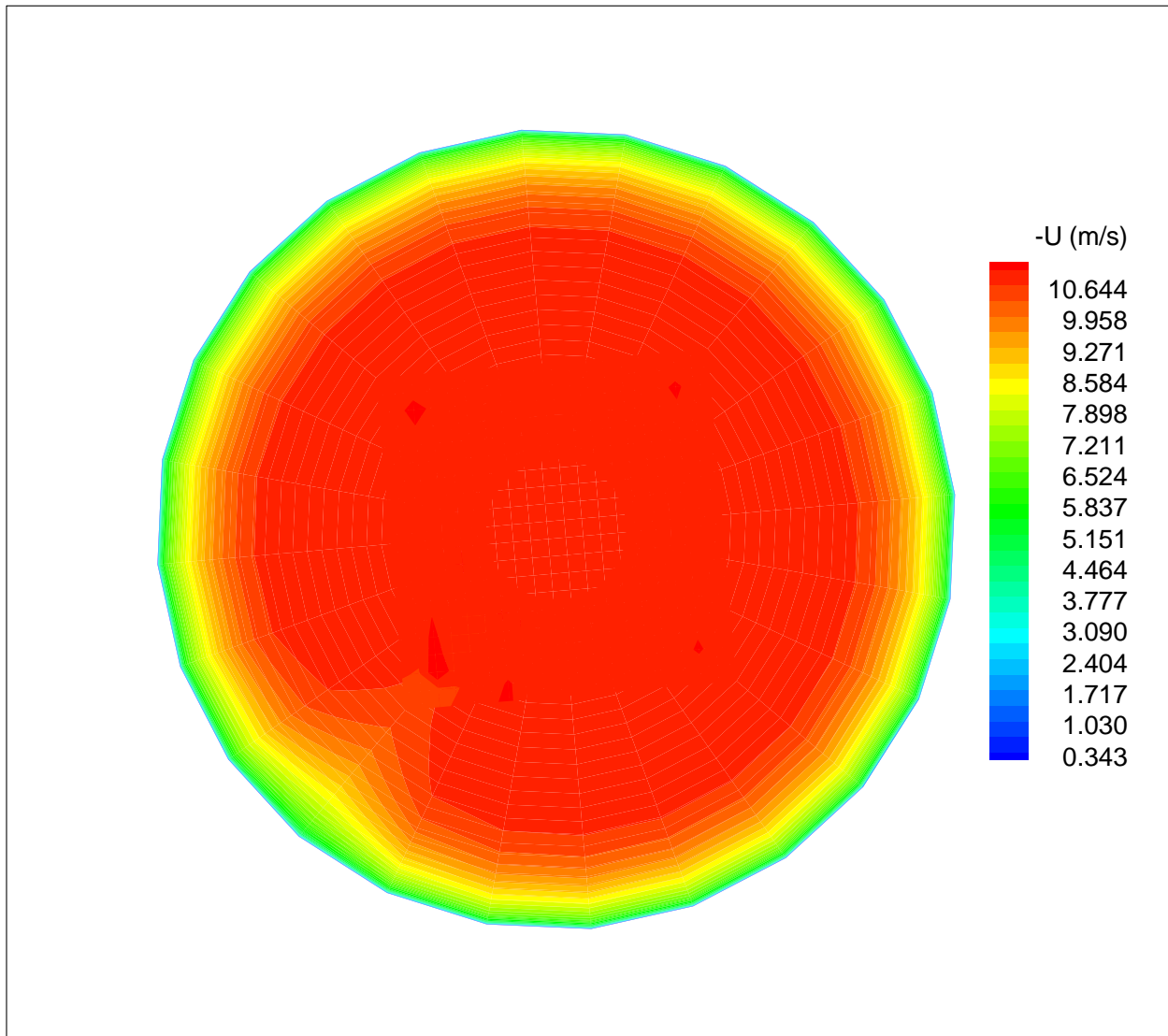


Figure 11 Velocity distribution at pipe exit. Note that the negative of velocity is plotted for clarity. The average exit velocity is approximately 10.5866 m/s.

Average Pressure: 8.061946 bar

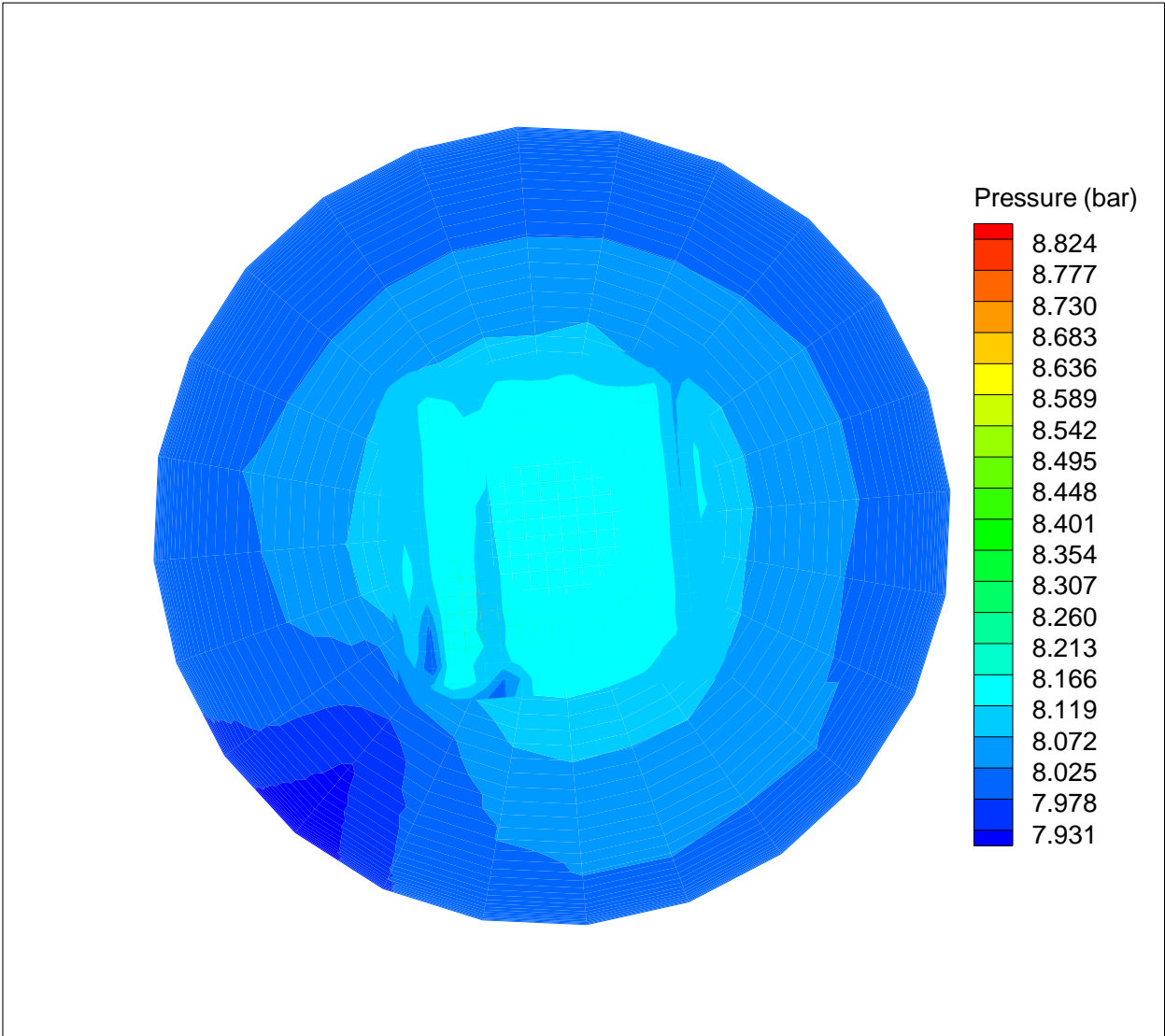


Figure 12 Pressure distribution at pipe exit. The average exit velocity is approximately 8.06 bar.

(d) Asymmetry Profiles

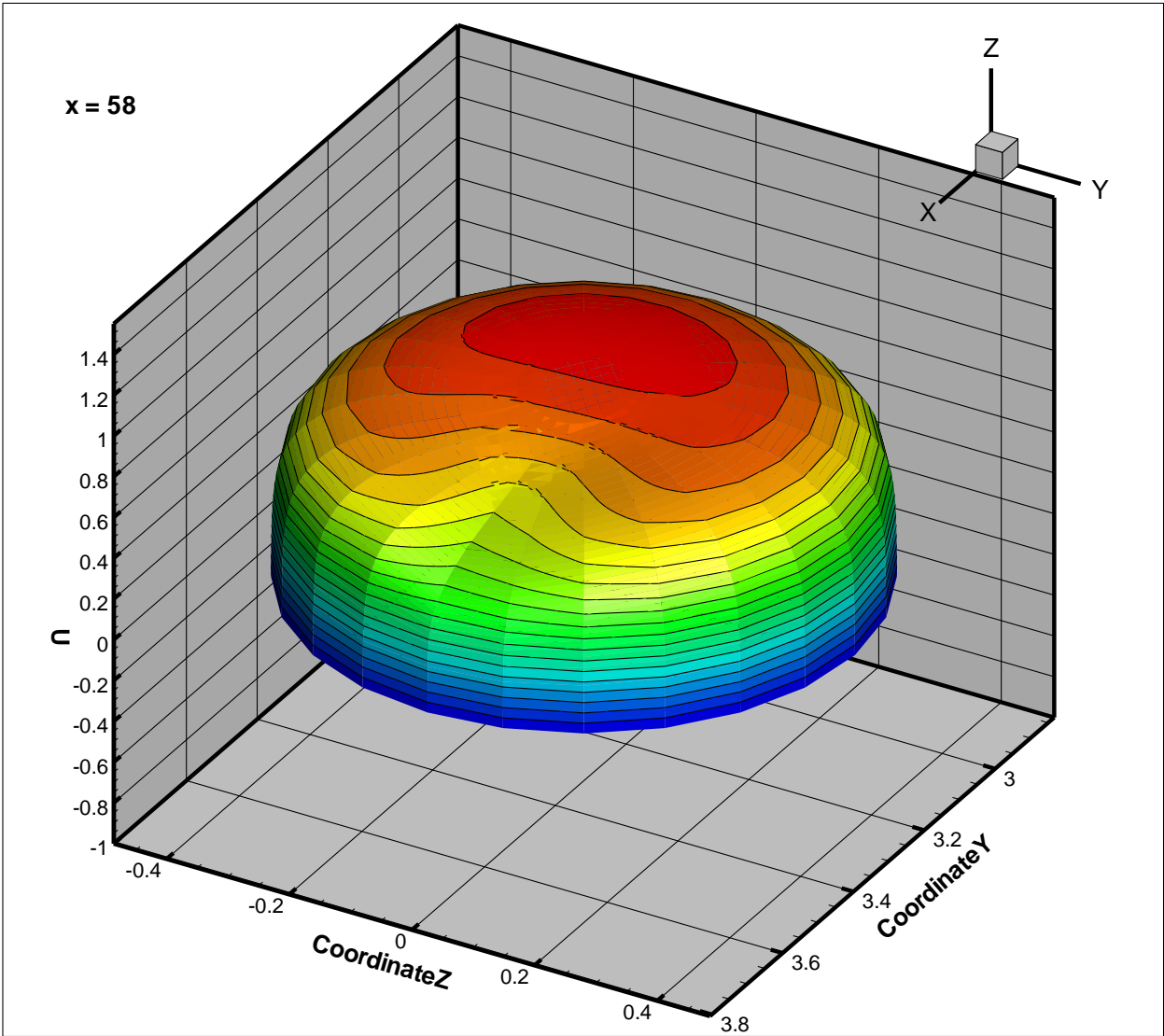


Figure 13 Velocity distribution at pipe exit, in a 3D plot

Detail Profiles of Bent Part

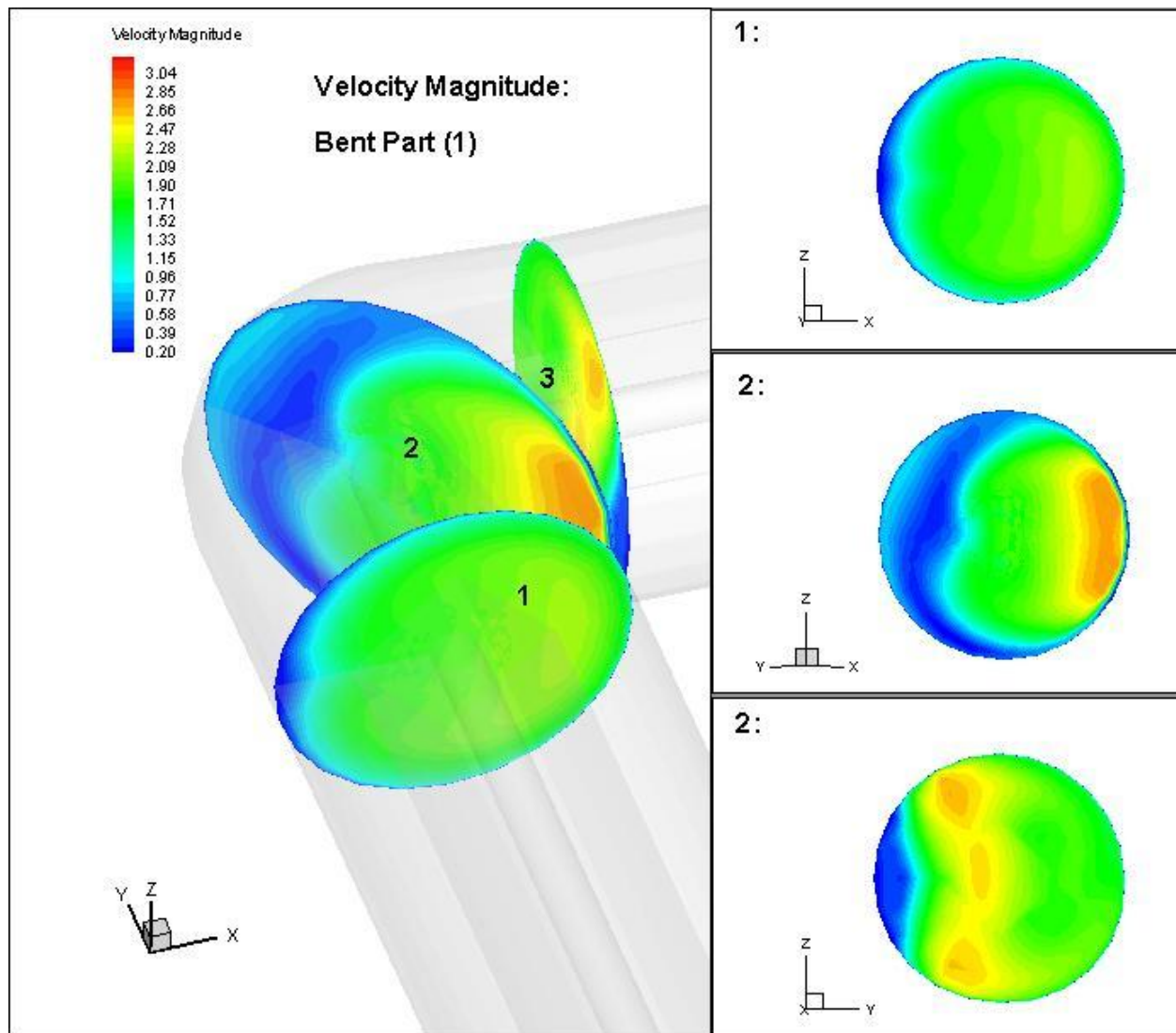


Figure 14 Magnitude of velocity in the vicinity of the first bend

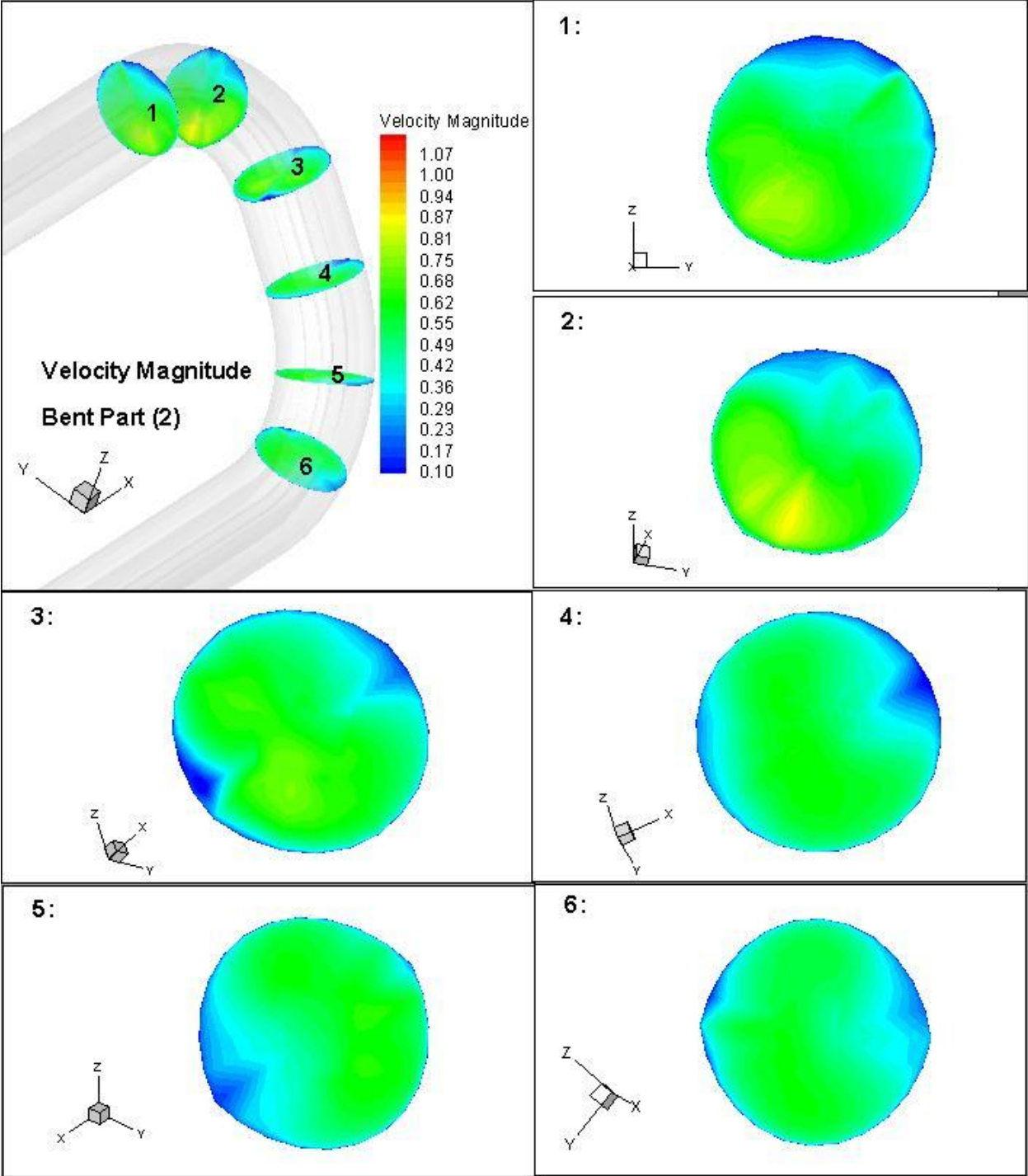


Figure 15 Magnitude of velocity in the vicinity of the second bend