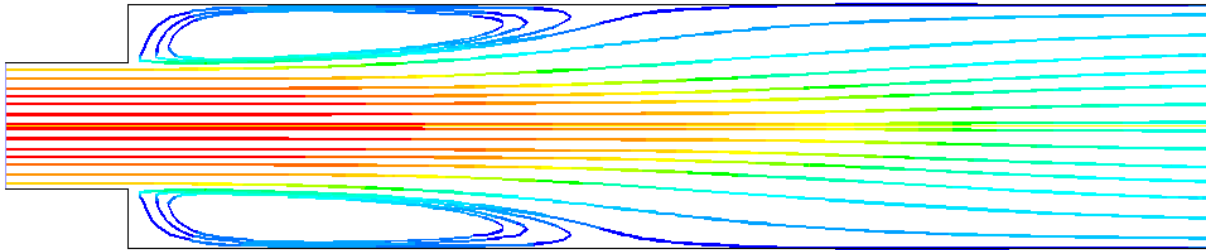# Exercise 3: Turbulent flow through a pipe expansion



*Purpose*

To perform a simulation of turbulent flow through the pipe expansion using a k-$\varepsilon$ turbulence model. Some insight can be gained on influence of the inlet boundary conditions on the turbulent flow characteristics.

*Summary*

The computational domain consists of two pipes. The diameter of the first pipe (inlet) is $D_{inlet}$=0.0788m and the diameter of the and second pipe (outlet) is D=0.1524m. The Reynolds number (based on the diameter D and the averaged velocity at the outlet) is $Re_D$=200 000. The Reynolds number is large so the flow has to be treated as turbulent. The present simulations will be performed using the k-$\varepsilon$ turbulence model with the wall function approach. The flow is 2D axisymmetric so only half of the domain shown above will be considered in simulations. The medium is water.

The velocity inlet boundary condition has to be specified at the entrance to the shorter pipe together with the inlet conditions for the turbulent quantities. The inlet mean velocity profile has been obtained from precursor simulation of the fully developed pipe flow and this profile should be used in the present study  (without any modifications) at the inlet boundary. Three different types of the inlet profiles will be used for the turbulent quantities:

   i) First, the k and $\varepsilon$ profiles obtained from precursor simulations of the fully developed pipe flow will be used (the profiles are included in the text file *inlet_profile_keps.prof*)
   ii) Next, the Default settings by Fluent (k=1 and $\varepsilon$=1) will be employed
   iii) Finally, the estimated values of the turbulent quantities: turbulent intensity Tu [%] and the turbulent  length scale $l_t$ [m] will be used

There is no necessity to generate the mesh. One has to open the *.cas file in Fluent. The discussion of the results has to be provided in the report (max 5 pages).

*Fluent program*

*I. Simulations using pre-computed inlet mean velocity, k and $\varepsilon$ profiles*

1. Unzip the file into your working directory, if it was not done yet.
2. Run the Fluent program (stand-alone version). **StartMenu/Programy/Ansys19/FluidDynamics/Fluent**. Set a double precision solver, 2D, serial.
3. Read the *.cas file (pipe_expansion_coarse.cas).
4. Verify the mesh and size of the domain **Mesh/Check** and the number of cells, **Mesh/Info/Size.**
5. The flow is 2D axisymmetric. Check the settings in **Setup/General.**
6. Select a proper turbulence model **Setup/Models/Viscous** and the wall treatment in under **Near-Wall Treatment.** Use **Standard Wall Function** approach
7. Verify the fluid properties.
8. Specification of the inlet boundary conditions. Read the inlet profiles of the mean velocity, turbulent kinetic energy, k, and dissipation of the turbulent kinetic energy $\varepsilon$ using **File/Read/Profile** (read the attached file 'inlet_profile_keps.prof'). Specify the inlet boundary conditions, **Setup/Boundary Conditions, Inlet, Edit**. Activate the profiles of the velocity magnitude, turbulent kinetic energy and dissipation. Use "line-" profiles instead of Const.
9. Set the discretization scheme for the convective terms to the first order upwind, **Solution/Methods**.
10. Set the convergence limit **Solution/Monitors/Residual** for all equations to 1e-5.
11. Initialize the flow using the inlet boundary and run the simulation. Verify the inlet profiles in **Display/Plots.** Control the convergence history. The simulations have to be continued until 'solution convergence' is obtained. Write the convergence history to the file, **File/Save Picture**.
12. After finishing the simulations with the first order scheme switch to the second order upwind scheme. Continue the iteration process (do not initialize the solution). Write the *.cas (mesh + settings) and *.dat (solution) files, **Write/Case and Data**.
13. Verify the Reynolds number. (Attention: In present case, the Reynolds number is based on the mean velocity at the outlet and the diameter of the second pipe D=0.1524m.) The area-averaged velocity at the outlet can be computed using **Results/Reports/Surface Integrals**, **Report Type, Area-Weighted Average**. If the Reynolds number is incorrect verify a specification of the boundary conditions, size of the computational domain or fluid properties (water).
14. Verify the contour plots of pressure, velocity, turbulent kinetic energy and dissipation, velocity vectors, pathlines in **Graphics and Animations.** Make the figures (report).
15. Verify the area-weighted average of the mean velocity ($U_{av}$) and the maximum axial velocity ($U_{axial,max}$) at the inlet to the computational domain using **Results/Reports/Surface Integrals Report Type, Area-Weighted Average or Vertex Maximum.** Write down both values. These values will be needed for normalization of the mean axial velocity and the turbulent kinetic energy (comparison with experimental data). Normalize the axial-velocity and turbulent

kinetic energy using **User Defined/Field Functions/Custom.** Select an appropriate quantity from the list **Field Functions**, introduce the division sign (from the screen), specify a name of a new quantity and click **Define**. More specifically:

a) Axial velocity has to be normalized (divided) by the maximum velocity at the inlet plane, $U_{axial}/U_{axial,max}$

b) Turbulent kinetic energy has to be normalized (divided) by squared mean velocity at the inlet plane, $k/(U_{av})^2$

Note that $U_{axial}$ and k are a fields of the velocity magnitude and the turbulent kinetic energy, and $U_{axial,max}$ and $U_{av}$ are single numbers.

16. Examine the mesh quality at the walls 'Wall' and 'Wall_d'. Wall_d corresponds to the second pipe (longer pipe). The mesh quality at the wall has to be examined by means of the dimensionless distance $y^+$ **Results/Plots/XY Plot, Turbulence, Wall YPlus**

    *The dimensionless distance $y^+$ is computed as $u_\tau y/\nu$, where $u_\tau=(\tau_w/\rho)^{1/2}$ is the friction velocity, y is the distance between the wall-neighboring cell centre and the physical wall, $\nu$ is the kinematic viscosity of the fluid, $\tau_w$ is the wall shear stress, $\rho$ is the density.*

    *The wall function approach allows for simulations of the turbulent boundary layers on relatively coarse meshes. It means that $y^+$ can be in the range $y^+=30-300$. The logarithmic velocity profile is employed for $y^+=30-300$. If $y^+$ is lower than 10 the linear velocity profile is used. If $y^+>>500$ the mesh has to be refined close to walls. One can always use **Adapt/Boundary** option to refine the mesh near to the wall, if needed. We will not need this option now.*

17. Save the cas and dat files **File/Write/Case&Data**

18. Compare the numerical results with experimental data (included in the folder *exper-data*) along the vertical line x=-0.0381 and along the horizontal line r=0.07271. Fig. 1 shows both vertical and horizontal lines. The sections x=-0.0381 and r=0.07271 are already defined in the *.cas file so there is no necessity to generate them again. Use **Results/Plots/XYPlot, Load File** to load the experimental data (description of the experimental data is given below, section 'Description of the experimental data').

    a) Compare the numerical results with experiment along the vertical line x=-0.0381 (both the normalized axial velocity and normalized turbulent kinetic energy) and write the graphics **File/Save Picture**. The vertical axis should be y, the horizontal axis should be the normalized velocity or normalized k

       *Note that the peak value of k is somewhat underpredicted. But the global result is satisfactory (the mean velocity is correct). This is due to the fact that the wall function technique appropriately corrects the mean velocity gradient at the wall (in the first cell on the wall). Without the wall function technique both mean velocity and the turbulent kinetic energy profiles will be wrong close to wall on such coarse mesh.*

    b) Compare the numerical results with experiment (normalized axial velocity) along the horizontal line r=0.07271. Write the axial-velocity profile along r=0.07271 to the text file **Results/Plots/XYplot, Write to File** for its later comparison with the other results. The vertical axis should be the normalized velocity the horizontal axis should be x.

*The measured axial velocity becomes constant at large axial distance. This is due to the fact that the real measurements were not performed there and the velocity has been estimated.*

## II. Simulations using pre-computed inlet mean velocity profile and the default Fluent conditions for k and $\varepsilon$

1. Continue the work with currently open project.
2. Change the boundary conditions for the turbulent kinetic energy k and for dissipation rate $\varepsilon$ from line- to Default Fluent conditions Const=1. **Setup/Boundary Condition**
3. Initialize the solution and run the simulation.
4. Write the solution using **File/Write/Case&Data**
5. Compare the numerical results with experimental data and with previous result along the horizontal line r=0.07271. Are the current numerical results different from experimental data from previously obtained results? Deactivate the experimental data and the previous numerical result on left panel in **Results/Plots/XYplot**. Write the normalized x-velocity profile along r=0.07271 to the text file **Results/Plots/XYplot, Write to File.**

## III. Simulations using pre-computed inlet mean velocity profile and the estimated values of the turbulent quantities

1. Continue the work with currently open project.
2. Change the inlet conditions for turbulence, **Setup/Boundary Condition, Inlet, Edit**, **Specification Method** from k and $\varepsilon$ to turbulent intensity Tu [%] and to the hydraulic diameter, $D_{inlet}$[m].

   *Relation between the turbulent kinetic energy, k, the turbulent intensity, Tu, and mean velocity $U_{av}$ is as follows:  $k=1.5(Tu\ U_{av})^2$*

   *For the fully developed pipe flow the turbulent intensity is approximately equal to Tu=4-5%.*

   *The turbulent length scale $l_t$ can be estimated as few percent of the inlet diameter (say up to 10%).  In Fluent it is about 7% so $l_t \cong 0.07 D_{inlet}$.*

   *The relation between the turbulent length scale $l_t$, k and $\varepsilon$ is:  $\varepsilon = \dfrac{k^{3/2}}{l_t}$ so one can also express the turbulent intensity Tu and the turbulent length scale $l_t$ by means of k and $\varepsilon$.*

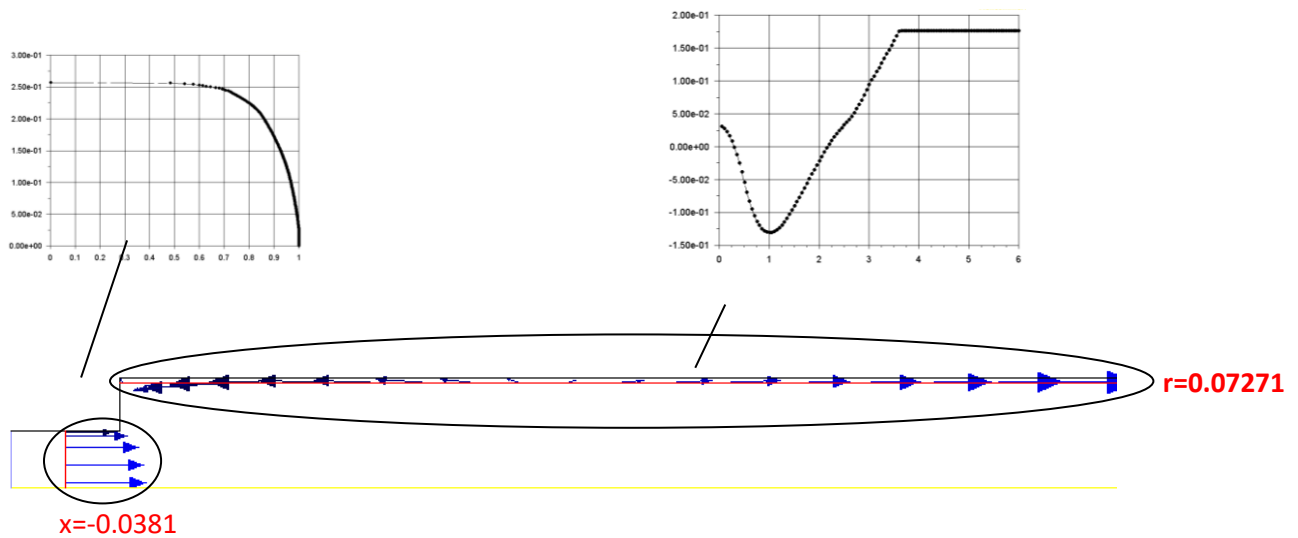   Define simply: Tu=4% and $D_{inlet}$ =  0.0788m.

3. Initialize the solution and run the simulation.
4. Write the solution using **File/Write/Case&Data**
5. Compare the numerical results with experimental data and with previous results along the horizontal line r=0.07271.Make a final plot comparing all the numerical results with

experiment. It there any influence of the inlet conditions on the results? Is it a good practice to use the Default conditions for the turbulent quantities ?

*Description of the experimental data included in the folder 'exper-data' and its graphical presentation*

**exper-axial-vel_x-0_0381.xy** – axial-velocity profile (normalized by the maximum velocity at the inlet) as a function of radius y at x=-0.0381

**exper-axial-vel_r0.07271.xy**– axial-velocity profile (normalized by the maximum velocity at the inlet) as a function of axial distance x at r=0.07271



**exper-k-over-u2_x-0_0381.xy**– the turbulent kinetic energy profile (normalized by the area-averaged velocity squared) as a function of radius y at x=-0.0381 (x/D=-0.25.)
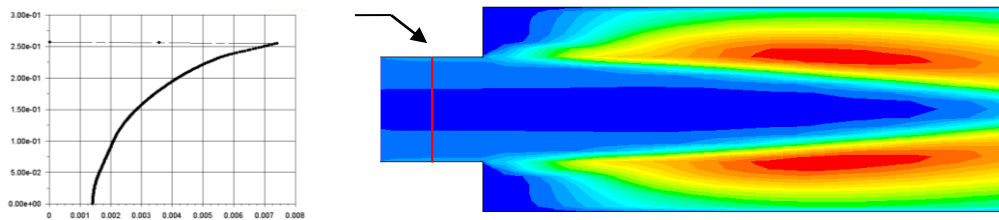


Fig. 1 The vertical and horizontal line segments at x=-0.0381 and r=0.07271. One has to compare the numerical results with experiment along these lines.