

Q MTF271, Assignment 1: Reynolds averaged Navier-Stokes

Q.1 Part I: Data of Two-dimensional flow

You can do the assignment on your own or in a group of two. It is recommended (but not required) that you use L^AT_EX (an example of how to write in L^AT_EX is available on the course www page). You find L^AT_EX at [here](#). You can also use L^AT_EX on-line.

You'll use data from a coarse DNS. Although some of the data are probably not fully accurate, in this exercise we consider the data to be exact. You can use Matlab, Octave or Python. Both Octave and Python are open-source software. Octave is a Matlab clone. Many large Swedish industries prefer engineers to use Python instead of Matlab due to Matlab's high license fees.

Use Matlab/Octave/Python to read data files of the mean flow (\bar{v}_1 , \bar{v}_2 , \bar{p}) and turbulent quantities. ($\overline{v_1'^2}$, $\overline{v_2'^2}$, $\overline{v_3'^2}$ and $\overline{v'_1 v'_2}$). You will analyze one of the following flows:

Case 1: Flow over a wavy wall (small wave) [[207](#), [207](#)]. $Re = 10\,000$ ($\nu = 1 \cdot 10^{-4}$, $\rho = 1$) based on the bulk velocity in the channel at $x = 0$ and the channel height.

Case 2: Flow over a wavy wall (large wave) [[207](#), [207](#)]. $Re = 10\,000$ ($\nu = 1 \cdot 10^{-4}$, $\rho = 1$) based on the bulk velocity in the channel at $x = 0$ and the channel height.

Case 3: Flow over a hill $Re = 10\,595$ ($\nu = 1/10595$, $\rho = 1$) based on the bulk velocity in the channel at $x = 0$ and the channel height.

Case 4: Flow over two hills $Re = 10\,595$ ($\nu = 1/10595$, $\rho = 1$) based on the bulk velocity in the channel at $x = 0$ and the channel height.

Periodic boundary conditions are imposed in streamwise (x_1) and spanwise (x_3) directions in all flows.

Get the case the case number by register in a group at Canvas. Download the data from

http://www.tfd.chalmers.se/~lada/comp_turb_model. At the www-page you can download a py-file (`pl_vect.py`) or m-file (`pl_vect.m`) which read the data and plots the vector field and the pressure contours. You must also download the function `dphidx_dy.py` or `dphidx_dy.m` which compute the gradients, see Section [Q.2](#). Make sure you put this function in the directory where you execute `pl_vect.py` or `pl_vect.m`.

The report, along with the Matlab/Octave/Python files(s), should be submitted electronically at Canvas; the deadline can be found in the time table.

Report: See the following as a guideline to discuss the results. Write a flowing report, so the reader does not need the assignment. Discuss all plots based on reasonability. If you figure out deviations from the expected outcome discuss them. You should discuss all figures; if you have nothing to write about a figure, don't include it.

Q.1.1 Analysis

Study the flow. In which regions do you expect the turbulence to be important? Let's find out. The two-dimensional time-averaged Navier-Stokes for the x_1 momentum

reads (the density is set to one, i.e. $\rho = 1$)

$$\begin{aligned}\frac{\partial \bar{v}_1 \bar{v}_1}{\partial x_1} + \frac{\partial \bar{v}_1 \bar{v}_2}{\partial x_2} &= -\frac{\partial \bar{p}}{\partial x_1} + \nu \frac{\partial^2 \bar{v}_1}{\partial x_1^2} - \frac{\partial \bar{v}_1'^2}{\partial x_1} + \nu \frac{\partial^2 \bar{v}_1}{\partial x_2^2} - \frac{\partial \bar{v}_1' \bar{v}_2'}{\partial x_2} \\ \frac{\partial \bar{v}_1 \bar{v}_2}{\partial x_1} + \frac{\partial \bar{v}_2 \bar{v}_2}{\partial x_2} &= -\frac{\partial \bar{p}}{\partial x_2} + \nu \frac{\partial^2 \bar{v}_2}{\partial x_2^2} - \frac{\partial \bar{v}_1' \bar{v}_2'}{\partial x_1} + \nu \frac{\partial^2 \bar{v}_2}{\partial x_1^2} - \frac{\partial \bar{v}_2'^2}{\partial x_2}\end{aligned}\quad (\text{Q.1})$$

Recall that all the terms on the right-hand side represent x_1 components of forces per unit volume.

Q.1.2 The momentum equations

The files `pl_vect.py`, `pl_vect.m` loads the data file and plots the profiles of $\bar{v}_1'^2$ at some x stations, the velocity vector field and a contour plot of velocity gradient $\partial \bar{v}_1 / \partial x_2$. Choose two x_1 locations (vertical grid lines), one without recirculation (close to the left boundary, for example) and one grid line where there is a recirculation region.

Assignment 1.1. Plot the stresses along vertical grid lines at these two locations using the Python `plt.plot(x, y)` or Matlab/Octave command `plot(x, y)`. Please make sure that in your report the numbering on the axis and the text in the legend is large enough; you can use the command

```
h1=gca;
set(h1,'fontsize',[20]) %the number '20' gives the fontsize
```

The size of the labels and the title is similarly controlled by

```
xlabel('x/H','fontsize',[20])
ylabel('y/H','fontsize',[20])
title('velocity','fontsize',[20])
```

Assignment 1.2. Compute and plot all terms in Eq. Q.1 (see Fig. 6.6). You will need to compute the derivatives of e.g. \bar{v}_1 and \bar{p} . In `pl_vect.m` and `pl_vect.py` the function `dphidx_dy` is used to compute $\partial \bar{v}_1 / \partial x_1$ and $\partial \bar{v}_1 / \partial x_2$. Use this function to compute all derivatives that you need. To enhance readability you may omit the small terms or use two plots per vertical grid line. Make also a zoom near the walls. For example, for a $x - y$ plot

```
plot(u,y,'linew',2) % linewidth=2
```

you may want to zoom in on $y=[0 0.01]$ and $u=[-0.1 0.4]$; this is achieved by

```
axis([-0.1 0.4 0 0.01])
```

The 'axis' command can be used together with any plot, e.g. with 'surf' and 'quiver'.

Which terms are negligible? Can you explain why they are negligible?

What about the viscous terms: where do they play an important role?

Use the Taylor expansion for the fluctuating terms and show analytically, which terms are non-zero *at* the wall. Compare with the DNS results and explain derivations.

So far we have looked at the \bar{v}_1 -momentum equation. The database corresponds to a two-dimensional flow. Now let's think of the forces as vectors. The gradient of the normal stresses in the $x_1 - x_2$ plane represents the force vector (see Eq. Q.1)

$$\mathbf{F}_N = \left(-\frac{\partial \bar{v}_1'^2}{\partial x_1}, -\frac{\partial \bar{v}_2'^2}{\partial x_2} \right) \quad (\text{Q.2})$$

and the corresponding force vector due to the shear stresses reads (see Eq. Q.1)

$$\mathbf{F}_S = \left(-\frac{\partial \bar{v}_1' \bar{v}_2'}{\partial x_2}, -\frac{\partial \bar{v}_1' \bar{v}_2'}{\partial x_1} \right) \quad (\text{Q.3})$$

Find the first term in Eqs. Q.2 and Q.3 in the \bar{v}_1 momentum equation, Eq. Q.1. Consider the second line in Eq. Q.1 which is the \bar{v}_2 equation and find the other two terms in Eqs. Q.2 and Q.3. Note that \mathbf{F}_N and \mathbf{F}_S are forces per unit volume ($[N/m^3]$).

Assignment 1.3. Plot the vector field \mathbf{F}_N to learn something about its properties. When $\bar{v}_2'^2$ reaches a maximum or a minimum along a grid line normal to the wall, what happens with the vector field \mathbf{F}_N ? Zoom-in on interesting regions.

Assignment 1.4. Plot also vector fields of the shear stress, \mathbf{F}_S (see Eq. Q.3), the pressure gradient and the viscous terms. Zoom up in interesting regions.

Q.1.3 The turbulent kinetic energy equation

The exact transport equation for the turbulent kinetic energy, k , reads

$$\begin{aligned} \frac{\partial}{\partial x_j} (\bar{v}_j k) &= \nu \frac{\partial^2 k}{\partial x_j \partial x_j} + P^k + D_k - \varepsilon \\ P^k &= -\bar{v}_i' \bar{v}_j' \frac{\partial \bar{v}_i}{\partial x_j} \end{aligned} \quad (\text{Q.4})$$

Assignment 1.5. Plot the production term along the two grid lines (see Fig. 8.3). Explain why it is large at some locations and small at others. The production term consists of the sum of four terms, two of which involve the shear stress while the other include the normal stresses. Compare the contributions due the shear stress and the normal stresses.

Assignment 1.6. Plot the dissipation and compare it with the production. Do you have local equilibrium (i.e. $P^k \simeq \varepsilon$) anywhere? Take ε from 2D RANS simulation.

Q.1.4 The Reynolds stress equations

The modeled transport equation for the Reynolds stresses can be written as

$$\begin{aligned}\frac{\partial}{\partial x_k} \left(\bar{v}_k \bar{v}'_i \bar{v}'_j \right) &= \nu \frac{\partial^2 \bar{v}'_i \bar{v}'_j}{\partial x_k \partial x_k} + P_{ij} + \Phi_{ij} + D_{ij} - \varepsilon_{ij} \\ P_{ij} &= -\bar{v}'_i \bar{v}'_k \frac{\partial \bar{v}_j}{\partial x_k} - \bar{v}'_j \bar{v}'_k \frac{\partial \bar{v}_i}{\partial x_k}\end{aligned}\quad (\text{Q.5})$$

The pressure-strain term, Φ_{ij} , use the models in Eqs. 11.91, 11.57, 11.90, 11.95 and 11.96.

1. In the damping function, f (see Eq. 11.92), $|n_{k,w}(x_k - x_{k,w})|$ denotes the distance to the nearest wall. If, for example, a lower wall is the closest wall to node (I, J) , then

$$|n_{k,w}(x_k - x_{k,n})| = \{(x(I, J) - x(I, 1))^2 + (y(I, J) - y(I, 1))^2\}^{1/2} \quad (\text{Q.6})$$

Note that you have to search through all wall nodes to find which wall node gives the smallest value. The damping function, f , involves k and ε . These should be taken from the 2D RANS simulation (they are loaded in `pl_vect.py` and `pl_vect.m`).

2. If we assume, again, that the lower wall is the closest wall to cell (I, J) and that the lower wall is horizontal, then $n_{i,w} = (0, 1)$. To compute $n_{i,w}$ for the general case (see Eqs. 11.95 and 11.96), compute first the vector which is parallel to the wall, $s_{i,w}$, and then compute $n_{i,w}$ from $s_{i,w}$ (see Eq. Q.11)
3. The diffusion terms D_{ij} and D^ε can be modeled using the Generalized Gradient Diffusion Hypothesis GGDH of [208]

$$D_{ij} = \frac{\partial}{\partial x_m} \left(c \bar{u}_k \bar{u}_m \frac{k}{\varepsilon} \frac{\partial \bar{v}'_i \bar{v}'_j}{\partial x_k} \right) \quad (\text{Q.7})$$

This diffusion model can cause numerical problems, and the GGDH is then replaced by a simple eddy viscosity model

$$D_{ij} = \frac{\partial}{\partial x_m} \left(\frac{\nu_t}{\sigma_k} \frac{\partial \bar{v}'_i \bar{v}'_j}{\partial x_m} \right), \quad \nu_t = C_\mu k^2 / \varepsilon \quad (\text{Q.8})$$

The following constants should be used:

$$(c_\mu, c_1, c_2, c_{1w}, c_{2w}, \sigma_k) = (0.09, 1.5, 0.6, 0.5, 0.3, 1)$$

4. The dissipation is modelled as (see Section 11.7.3)

$$\varepsilon_{ij} = \frac{2}{3} \varepsilon \delta_{ij}$$

Assignment 1.7. Choose two stresses. Plot the different terms in the equations for one vertical grid line fairly close to the inlet (not too close!). Looks at the corresponding plot for channel flow, see Fig. 9.1. Use the simple eddy viscosity model for the

turbulent diffusion term. Use k and ε from the 2D RANS simulation. If the figure becomes too crowded, use two plots per vertical grid line or simply omit terms that are negligible. Try to explain why some terms are large and vice versa. Usually, a stress is large in locations where its production (or pressure-strain) term is large. Is that the case for you?

- Assignment 1.8. Compute the stresses using the Boussinesq assumption, i.e. $\overline{v'_i v'_j} = -2\nu_t \bar{s}_{ij} + (2k/3)\delta_{ij}$ where $\nu_t = c_\mu k^2/\varepsilon$. Use k and ε from the 2D RANS simulation. Compare the eddy-viscosity stresses with two of the Reynolds stresses from the database. Make also a zoom-in near walls.

When using the Boussinesq assumption the production of turbulent kinetic energy (use k and ε from the 2D RANS simulation)

$$P^k = 2\nu_t \bar{s}_{ij} \bar{s}_{ij} \quad (\text{Q.9})$$

is always positive. The exact production of turbulent kinetic energy (see Eq. Q.4) is usually positive. It can however become negative.

- Assignment 1.9. Compute the exact production in Eq. Q.4 in the entire domain to investigate if the production is negative anywhere. If so, explain why (for physical explanation, see Item II at p. 99).

The reason why the eddy-viscosity production in Eq. Q.9 must be positive is that neither ν_t nor $\bar{s}_{ij} \bar{s}_{ij}$ can go negative. Another way to explain this fact is that the modeled Reynolds stress, $\overline{v'_i v'_j}$, and the strain rate tensor, \bar{s}_{ij} are parallel. To find out to what degree the exact Reynolds stress and the strain rate are parallel, one can compute the eigenvectors.

- Assignment 1.10. The realizability concept is discussed in Section 13. It is shown that the Boussinesq assumption can give unphysical negative normal Reynolds stresses (if $\partial \bar{v}_1 / \partial x_1$ in Eq. 13.2 is very large then $\overline{v_1'^2}$ goes negative). A remedy is presented in Section 13 in which the turbulent viscosity is limited using the eigenvalues of the strain-rate tensor, \bar{s}_{ij} , see Eq. 13.12. Compute the eigenvalues of \bar{s}_{ij} in the entire domain. Check if the limiter in Eq. 13.12 reduces ν_t along the two vertical lines you previously chose. Find out where in the entire domain the limiter has largest effect. Is this region a stagnation region? (the object of Eq. 13.12 is to limit ν_t in stagnation regions, see Section 12.3).

Q.2 How to compute derivatives on a curvi-linear mesh

In this section we describe how the derivatives on a curvi-linear grid are computed in the provided Python `dphidx_dy.py` or Matlab function `dphidx_dy.m`. On a Cartesian grid it is more convenient to use the built-in Python/Matlab/Octave function `gradient`, but the approach used below works for all meshes, including Cartesian ones.

The data you have been given, x_1 and x_2 and all variables, are stored at the grid points, i.e. at $(x_{1,sw}, x_{2,sw})$, $(x_{1,se}, x_{2,se})$, $(x_{1,nw}, x_{2,nw})$ and $(x_{1,ne}, x_{2,ne})$. When you need a variable, say v_1 , at the center of the cell, compute it as

$$v_{1,P} = \frac{1}{4}(v_{1,sw} + v_{1,se} + v_{1,nw} + v_{1,ne}) \quad (\text{Q.10})$$

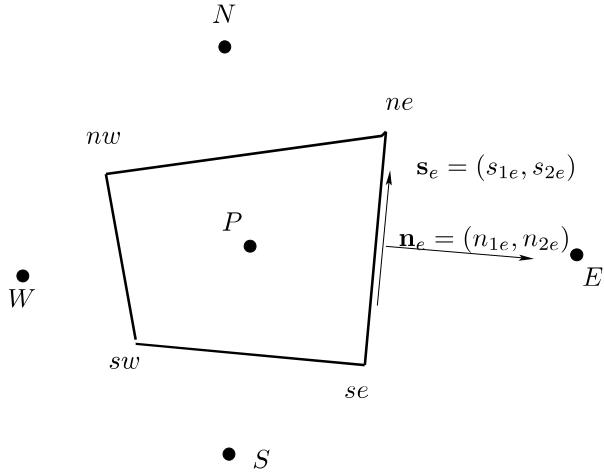


Figure Q.1: Control volume. The velocity v_1 is stored at the corners (ne, nw, ...). Coordinates x_1, x_2 are given at the corners (ne, nw, ...).

Let's compute $\partial v_1 / \partial x_1$. In order to do that we use Gauss' law over a control volume centered at face e (dashed control volume in Fig. Q.1). The divergence theorem for a scalar, ϕ , reads

$$\int_V \frac{\partial \phi}{\partial x_i} dV = \int_A \phi n_i dA$$

To compute $\partial v_1 / \partial x_1$ we set $\phi = v_1$ and $i = 1$ which gives

$$\int_V \frac{\partial v_1}{\partial x_1} dV = \int_A v_1 n_1 dA$$

Assuming that $\partial v_1 / \partial x_1$ is constant in the volume V we obtain

$$\frac{\partial v_1}{\partial x_1} = \frac{1}{V} \int_A v_1 n_1 dA$$

In discrete form we can write (see Fig. Q.1)

$$\left(\frac{\partial v_1}{\partial x_1} \right) = \frac{1}{V} \sum_{i=e,n,w,w} (v_1 n_1 A)_i =$$

$$\frac{1}{V} \{(v_1 A n_1)_e + (v_1 A n_1)_n + (v_1 A n_1)_w + (v_1 A n_1)_s\}$$

Q.2.1 Geometrical quantities

It is useful to first compute the unit vectors \mathbf{s} along the control volume. For the east face, for example, we get

$$s_{1e} = \frac{x_{1,ne} - x_{1,se}}{d_e}$$

$$s_{2e} = \frac{x_{2,ne} - x_{2,se}}{d_e}$$

$$d_e = \sqrt{(x_{1,ne} - x_{1,se})^2 + (x_{2,ne} - x_{2,se})^2}$$

(note that the area of the east face A_e is equal to d_e since $\Delta z = 1$). The relation between the normal vector \mathbf{n} , \mathbf{s} and the unit vector in the z -direction

$$\begin{aligned}\mathbf{s} \cdot \mathbf{n} &= 0 \\ \mathbf{s} \times \hat{\mathbf{z}} &= \mathbf{n},\end{aligned}$$

gives us the normal vector for the east face as

$$\begin{aligned}n_{1e} &= s_{2e} \\ n_{2e} &= -s_{1e}.\end{aligned}\tag{Q.11}$$

Q.3 Part II: StarCCM+

In this task, a commercial CFD software (StarCCM+ 11.02.009) will be used. The task is to do simulation of an asymmetric diffuser. Several turbulence models will be used and the results will be compared with experimental data. Before doing the task, it is recommended to first do one of the tutorials in the StarCCM+. The tutorial is the steady backward facing step tutorial. The tutorials can be found in the StarCCM+ user guide.

Q.3.1 Backward Facing Step Tutorial

This tutorial is a good bridge before doing the asymmetric diffuser case. Here are some steps to access the tutorial:

1. Open a terminal window. In the terminal window, type starccm+
2. To start a new simulation, click File → new simulation
3. Tick the Power-On-Demand box and fill the license box with the POD code.
4. Download the tutorial instruction and data from the course homepage

Assignment 1.11 Continue to work on the backward facing step flow. Choose a new turbulence model (right click on Continua/Fluid and select Select models Untick All y+ Wall treatment and then untick the turbulence model. Now you can choose a new turbulence model. Run a couple different turbulence model and compare the results. A short recirculation region is usually connected to high turbulence in the shear layer bounding the recirculation region.

Q.3.2 Asymmetric diffuser case

In this case 2D Asymmetric diffuser will be studied. The Reynolds number ($Re = 20000$) is based on the bulk velocity and the inlet channel height. The case is built based on these references. It is highly recommended to read these references before doing the case :

1. Ercoftac case 8.2: Flow through an asymmetric plane diffuser
[Description](#).
2. El-Behery & Hamed [209]
3. Buice & Eaton [210]
4. Davidson [162].

Q.3.3 Brief instruction to begin the asymmetric diffuser case

1. Start the StarCCM+
 - Open a terminal window → type StarCCM+
2. Create a new simulation
 - Click the new simulation icon → tick the power on demand box
 - Fill the license box with the POD license
3. Draw the geometry
 - Follow and modify the geometry in the ercoftac case 8.2
 - Create a new 3D-CAD Models
 - Create a new sketch on XY plane
 - Start by drawing the asymmetric diffuser without the radius
 - Right click on each lines and apply vertical / horizontal constraint accordingly
 - Set the diffuser angle to 10 degree
 - Set the length of each lines. Use $H = 1$ m. Length of the inlet = $60H$. Length of the diffuser = $21H$. Length of the outlet = $22H$. Do not need to set the height of the outlet.
 - Create a fillet on the beginning and end of the diffuser line. ($R = 9.7H$)
 - Set your base point (which is the point where the channel start to bend) at $X = 0$ and $Y = 3.7$ m
 - Extrude the sketch 1 m for easier renaming
 - Rename the faces accordingly (inlet, outlet, top, bottom, leftside, rightside)
4. Create a new geometry part. (The same way as in backward facing step tutorial)
5. Assign parts to regions. (The same way as in backward facing step tutorial)
6. Set the boundary condition accordingly. Inlet = velocity inlet; Outlet = pressure outlet; leftside = symmetry plane; rightside = symmetry plane.

7. Create mesh, choose either unstructured (recommended) or directed mesh

- Create unstructured mesh
 - Right click on the operation → new → mesh → Badge for 2D Meshing → Input Parts → Diffuser/Region → Execute
 - Right click on the operation → new → mesh → Automated mesh (2D) → Parts → Diffuser/Region → Polygonal Mesher → Prism layer Mesher → OK
 - Expand Automated mesh (2D) → Right click Default Controls → Edit
 - Target Surface Size
 - * Percentage of Base = 10.0
 - Minimum Surface Size
 - * Percentage of Base = 2.5
 - Number of prism layers
 - * Number of prism layers = 10.0
 - Prism layer stretching
 - * Prism layer stretching = 1.2
 - Prism layer total thickness
 - * Percentage of Base = 25.0
 - Right click on Custom Control → Surface Control
 - Expand Surface Control → expand Controls → Wake Refinement → Check
 - Expand Values → Wake Refinement
 - Isotropic Size
 - * Percentage of Base = 10.0
 - Wake Refinement
 - * Growth Rate = 1.05
 - Right click Automated mesh (2D) → Execute
- Create directed mesh
 - Right click on the operation → new → directed mesh → body1 → OK
 - Expand operations → Right click on the directed mesh → edit
 - Add rightSide as a source surfaces and leftSide as target surfaces
 - Right click on source meshes → new source mesh → patch mesh → body1 → OK
 - Click auto populate feature edges icon
 - Change the mode from patch topology into patch mesh
 - Click the inlet line
 - * Number of divisions = 80
 - * Type = two sided hyperbolic
 - * Spacing start and end = 3e-4 m
 - Click the top line
 - * Number of divisions = 421
 - * Type = constant

- Right click on the mesh distributions → create new volume distribution. Set the number of layers = 2
- Close the directed mesh and execute the directed mesh
- Create 2D Mesh
 - * Go to Menu bar → Mesh → Convert to 2D → Delete 3D regions after conversion → OK

8. Select turbulence model

- Right click on Physics 1 2D → Select model
- In the new window select:
 - (a) Untick the auto-select recommended models
 - (b) Two dimensional
 - (c) Steady
 - (d) Gas
 - (e) Coupled Flow
 - (f) Constant Density
 - (g) Turbulent
 - (h) Reynolds-Averaged Navier-Stokes
 - (i) Choose turbulence model that you want to use.

9. Set the Gas properties

- Expand Physics 1 folder → Gas → Air → Material Properties → Dynamic Viscosity → Constant → Value = $6E - 5Pa \cdot s$

10. Set the boundary condition

- Go to Regions.
- In the properties window, check that mesh continuum is parts meshes. Check that physics continuum is physics 1 2D
- Expand Body 1 2D → Boundaries → Inlet
 - (a) Physics Conditions → Velocity Specification → Method → Components
 - (b) Physics Values → Velocity → value → [1,0,0] m/s

11. Extract more data from the simulation (for example: Turbulent kinetic energy production)

- Expand Solvers folder → go to every subfolder and tick temporary storage retained if you find one

12. Set the stopping criteria

- For Reynolds Stress Model, use the default setting of the stopping criteria
- For the rest of the model use instruction below
- Disabled the maximum iteration criteria
- Create new stopping criteria

- (a) Right click on the Report folder → go to New Report → click Pressure Drop
- (b) In the Pressure Drop 1 properties window. High Pressure = Outlet. Low Pressure = Inlet.
- Right click on the Pressure Drop 1 → click on create monitor and plot from report
- Right click on the Stopping criteria folder → choose create new criterion → create from monitor → click on Pressure Drop 1 monitor
- In the Pressure Drop 1 Monitor criterion properties window
 - (a) Criterion option = Asymptotic
 - (b) Click on Asymptotic limit
- set [max-min] value = $1E - 4$ → Number of samples = 100

13. Create a scalar scene to show the result

- Go to Scenes folder → New Scene → Scalar
- Right click on the select function → select velocity [i]

14. Create planes to extract the results at the same location as the experimental data

- Right click on Derived Parts folder → New Part → Section → Plane
- Input Parts → Select → All the region
- Set normal → $x = 1, y = 0, z = 0$
- Display → New Geometry Displayer
- Specify the x -coordinate according to the experimental data (see below)
- Click Apply for each of the x -coordinate
 - (a) $X - 6 = -5.87m$
 - (b) $X03 = 2.59m$
 - (c) $X06 = 5.98m$
 - (d) $X14 = 13.56m$
 - (e) $X17 = 16.93m$
 - (f) $X20 = 20.32m$
 - (g) $X24 = 23.71m$
 - (h) $X27 = 27.09m$
 - (i) $X30 = 30.48m$
 - (j) $X34 = 33.87m$

15. Create the X-Y plot to monitor the result

- Right click on Plots folder → choose New Plot → click on X-Y Plot
- On the new X-Y Plot properties window, go to Parts → select all the derived parts
- Expand the X-Y Plot 1
 - (a) X-type → Type = Scalar → Scalar Function → Field Function = Velocity[i]

(b) Y-types → Y Type 1

- Type = Direction; smooth values
- Vector Quantity → Value = [0,1,0]
- Go to each Plane Section and change the X Offset accordingly

16. Insert experimental data

- Download the experimental data from the course homepage
- Expand Tools folder
- Right click on Tables folder → choose Create a New Table → click File Table → Load all the experimental data
- Go back to X-Y plot 1 monitor → right click on Data Series folder → Add Data → Select all the experimental data
- Expand Data series folder → select all the experimental data by click the first data hold shift button and click the last data → right click on the highlighted area → click swap column

17. Run the simulation

- Go to Solution → Run
- Other method: click icon which show a running man.

18. Collecting the simulation data

- After the simulation has reached its stopping criteria → Right click on XY plot 1 → Export → Save with appropriate name
- Add the simulation data to other simulation as you add experimental data

Assignment 1.12. Look at the results. How large is the pressure recovery (i.e. how much does the pressure increase from inlet to outlet).

Assignment 1.13. Choose a new turbulence model. Run the same turbulence models as for the back-step case and compare the results. Are the results different? Where do the difference appear? In the plane channel upstream of the diffuser? Compare turbulent quantities such as k and ν_t (or if you chose to use a Reynolds stress model, compare the shear stresses $v'_1 v'_2$). If the recirculation region is different, the reason is maybe connected to the turbulence level in the shear layer above the recirculation region. Or maybe the reason is that the incoming boundary layers are different. In the backstep flow, you found that one turbulence model was better than the other one(s). Is the same turbulence model best for this flow? Make other interesting comparisons!