
Hands-on session 2 – Laminar Elbow


Abstract

This session will introduce you to the solution procedure of a CFD simulation. For this purpose, a simple 2D elbow mixing problem under laminar flow is studied. Monitors of relevant physical quantities will be set up in order to check the convergence of the solution. Diverse numerical discretization schemes and mesh refinements will be analyzed and compared.

Goal

The aim of this hands-on session is to solve a laminar flow (with scalar transport) in a 2D elbow geometry and compare the results obtained with different discretization schemes and mesh refinements in order to get familiar with the following aspects of the software and of a CFD study:

1. set-up numerical schemes;
2. define monitors of physical quantities;
3. check convergence of a simulation;
4. get sensitivity to the order of accuracy of discretization schemes;
5. get sensitivity to mesh refinements.

Author	CFDLab@Energy		Page 1 of 33
CFD for Nuclear Engineering	Session 2		

Hands-on session 2 – Laminar Elbow.....	1
1 Introduction	3
2 Model set-up.....	4
2.1 Analyze the engineering problem	4
2.2 Grid check.....	4
2.3 Models	7
2.4 Materials	8
2.5 Boundary conditions.....	9
3 Solver Execution.....	11
3.1 Initialization.....	11
3.2 Solution Methods	11
3.3 Monitoring the solution.....	12
3.4 Start the calculation	15
4 Post-processing	16
4.1 Flow field	16
4.2 Profiles	18
4.3 Integral quantities	20
5 Revision of the Model	2224
5.1 Enabling Second-Order Discretization.....	22 24
5.2 Adapt (refine) the mesh.....	26 25
6 Home assignments	3130

1 Introduction

The mixing elbow configuration is encountered in piping systems in power plants and process industries. It is important to predict the flow field and temperature field in the neighborhood of the mixing region. The problem to be considered is shown schematically in Figure 1 and it is assumed 2D (i.e., the geometry is infinite in the z-direction and the solver assumes 1 m thickness to compute areas and volumes). A cold fluid at 26 °C enters through the large pipe and mixes with a warmer fluid at 40 °C in the elbow. The average properties of the fluid are Density = 1000 kg/m³, Specific heat = 4216 J/kgK, Thermal conductivity = 0.677 W/mK and Viscosity = 0.0008 kg/ms. The duct dimensions are in **cm**, and the fluid properties and boundary conditions are given in SI units. The Reynolds number at the main inlet is 200, while it is 250 at the secondary inlet, so that the flow is laminar.

Note: It is advised to prepare an excel file to summarize the fluid properties and boundary conditions for the resolved variables (pressure, x- and y-velocity, temperature). Some of these data may require calculation depending on the information given in the case description.

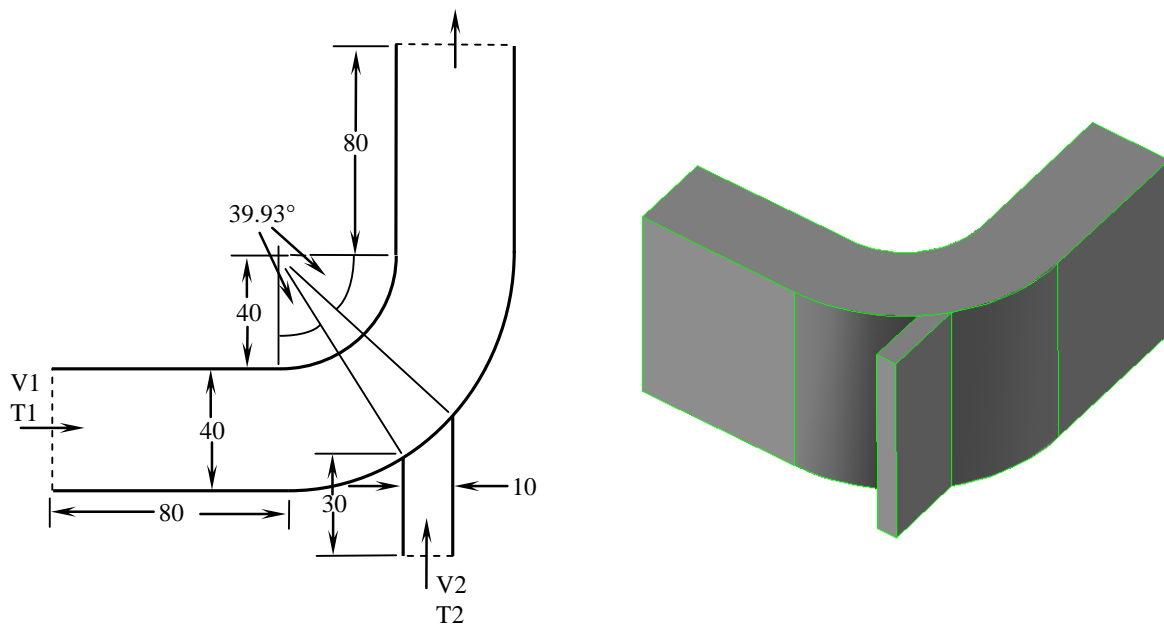



Figure 1: Layout (measures in cm)

Author	CFDLab@Energy		Page 3 of 33
CFD for Nuclear Engineering	Session 2		

2 Model set-up

2.1 Analyze the engineering problem

According to the problem description, determine the following:

- Velocity at the main inlet
- Velocity at the secondary inlet (in the elbow)
- Outlet average temperature

Question: how should we compute the outlet average temperature to obtain a meaningful value?

2.2 Grid check

2.2.1 Start ANSYS Fluent 2D

Select the 2D version of ANSYS Fluent with “double precision” activated.

2.2.2 File -> Read -> Mesh...

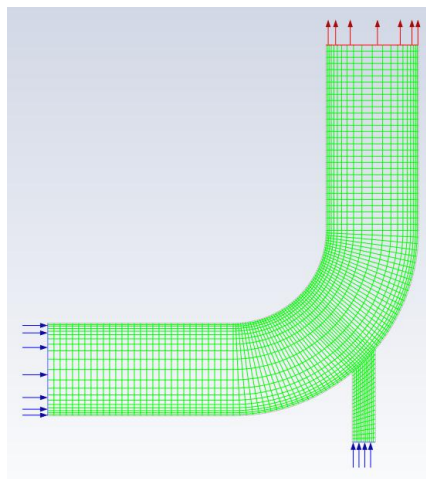
Open the mesh file “Elbow_2D.msh”.


2.2.3 Display the grid

Setup -> General -> Mesh -> Display...

Please, refer to the Session 2 for all the details about displaying the mesh.

Note: You can use the left mouse button to check which zone number corresponds to each boundary. If you click the left mouse button on one of the boundaries in the graphics window, its zone number, name, and type will pop-up in the FLUENT graphics window. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.



Author	CFDLab@Energy		Page 4 of 33
CFD for Nuclear Engineering	Session 2		

2.2.4 Check the grid size and quality

Domain -> Mesh -> Info -> Size

Setup -> General -> Mesh -> Check

Setup -> General -> Mesh -> Report Quality

Note: In order to visualize mesh skewness and measure characteristic dimensions of the geometry, we need to initialize the simulation with some values.

Solution -> Initialization (double click) -> Standard Initialization -> Initialize

Results -> Graphics -> Contours (double click)

Name: Skewness

Options: select ONLY Filled + Global Range + Auto Range

Contours of: Mesh... -> Cell Equiangle Skew

Colormap Options...

Number Format -> Type: float

Click Apply and Close

Click Save/Display and Close

Results -> Graphics -> Contours (double click)

Name: Orthogonal Quality

Options: select ONLY Filled + Global Range + Auto Range

Contours of: Mesh... -> Orthogonal Quality

Colormap Options...

Number Format -> Type: float

Click Apply and Close

Click Save/Display and Close

Note: deselecting “Node Values” will deactivate the interpolation between nodes and display the cells center value uniformly over the respective cells.


Question: where are the low quality elements located?

Results -> Plots -> Histogram (double click)

Histogram of: Mesh... -> Cell Equiangle Skew

Click Print and check the output in the Console window

Click Plot and check the output in the Graphics window

Author	CFDLab@Energy	 Page 5 of 33
CFD for Nuclear Engineering	Session 2	

Results -> Plots -> Histogram (double click)

Histogram of: Mesh... -> Orthogonal Quality

Click Print and check the output in the Console window

Click Plot and check the output in the Graphics window

Question: is the overall quality of the mesh acceptable?

2.2.5 *Scale the mesh*

Results -> Reports -> Surface Integrals (double click)

Report Type: Area

Surfaces: select “inflow1”

Click Compute and check the results


Scale the mesh according to the problem description.

Domain -> Mesh -> Scale...

Please refer to Session 2 for all the details.

2.2.6 *File -> Write -> Case & Data...*

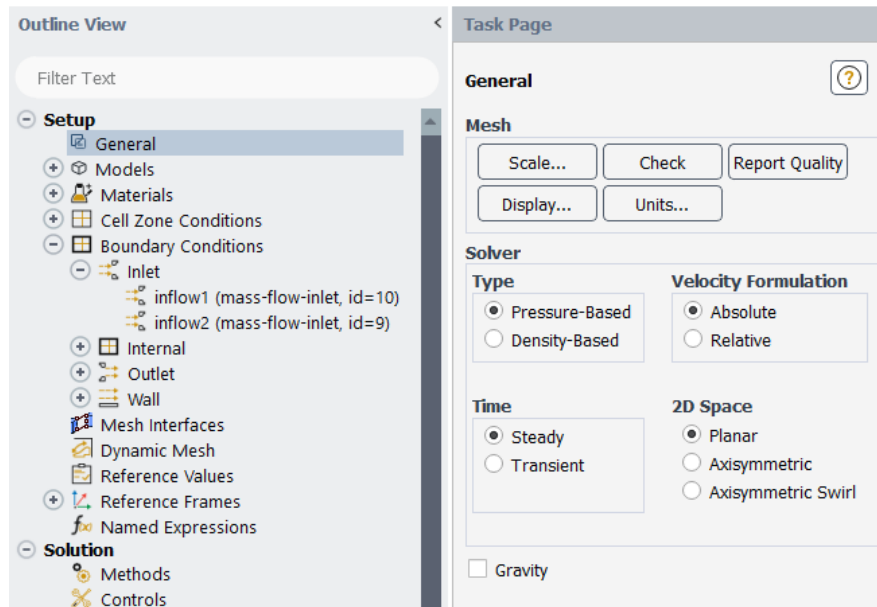
IT IS HIGHLY RECOMMENDED TO OFTEN SAVE YOUR CASE AND DATA FILES!

Author	CFDLab@Energy		Page 6 of 33
CFD for Nuclear Engineering	Session 2		

2.3 Models

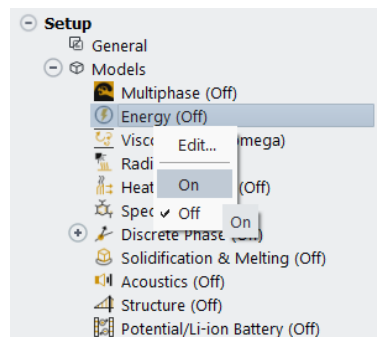
2.3.1 Solver

In this session we solve a steady state problem by using a pressure based solver:




2.3.2 Energy equation

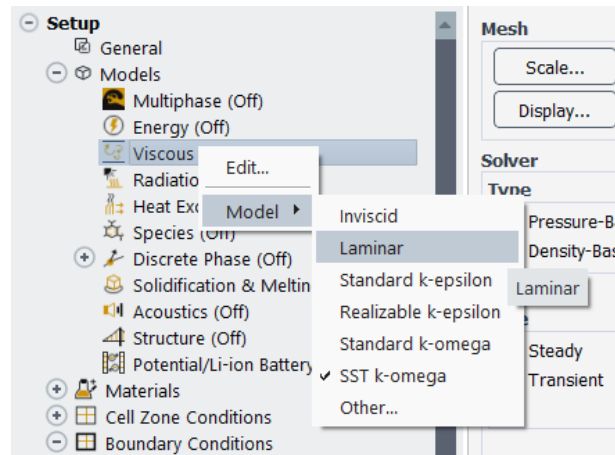
Enable heat transfer by activating the energy equation. In the tree, right-click Energy under Models and from the submenu that opens, click On:



2.3.3 Turbulence modelling

The flow is laminar so the turbulence model should be turned off. Right-click Viscous under Models and from the submenu that opens, go under Models and click Laminar:

Author	CFDLab@Energy	 Page 7 of 33
CFD for Nuclear Engineering	Session 2	



2.4 Materials

In the present case, the working fluid properties are given and do not depend on the temperature and pressure (they are considered constant). Therefore, we create a new material named “myfluid”.

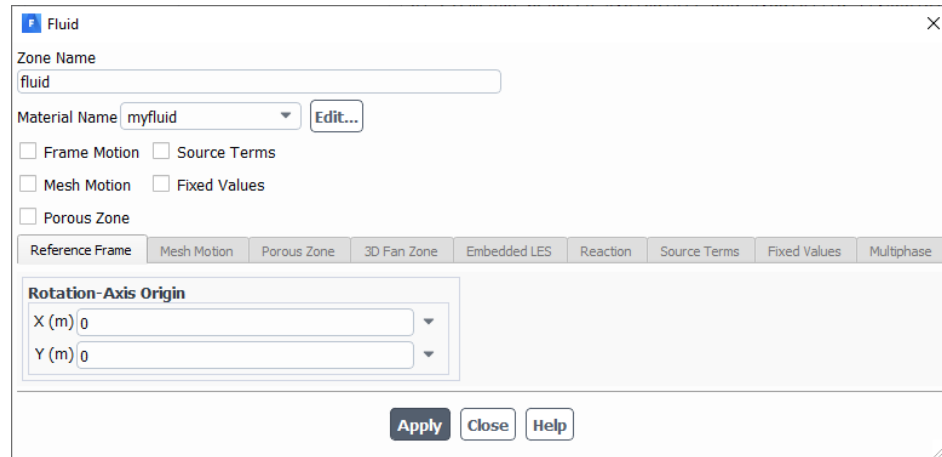
Setup -> Materials -> Fluid -> air (double click)

Click Change/Create and Overwrite air.

2.5 Boundary conditions

2.5.1 Setup -> Cell Zone Conditions -> Fluid -> fluid (double click)

If not already set-up, specify “myfluid” as the fluid material by selecting “myfluid” in the Material Name drop-down list. Then click Apply and Close.



2.5.2 Set the boundary conditions

1. Set the boundary conditions at the main inlet “inflow1”:

First change the type to velocity-inlet under the menu appearing when performing the following:

Setup -> Boundary Conditions -> Inlet -> inflow1 (right-click)

In the dialog box of the boundary conditions --- Setup -> Boundary Conditions -> Inlet -> inflow1 (double click) --- perform the following:

Momentum tab

Velocity Specification Method: Components

X-Velocity: 0.0002 m/s


Y-Velocity: 0 m/s

Thermal tab

Temperature: 299.15 K (26 °C)

Click Apply and Close

Note: The Reynolds number is typically based on the hydraulic diameter of a duct/channel. Here the hydraulic diameter of a channel of infinite depth reduces to twice the height of the channel. Check your data under the analysis of the problem accordingly.

Author	CFDLab@Energy		Page 9 of 33
CFD for Nuclear Engineering	Session 2		

2. Set the boundary conditions at the secondary inlet “inflow2”:

First change the type to velocity-inlet under the menu appearing when performing the following:

Setup -> Boundary Conditions -> Inlet -> inflow2 (rick-click)

In the dialog box of the boundary conditions --- Setup -> Boundary Conditions -> Inlet -> inflow2 (double click) --- perform the following:

Momentum tab

Velocity Specification Method: Components

X-Velocity: 0 m/s

Y-Velocity: 0.001 m/s

Thermal tab

Temperature: 313.15 K (40 °C)

Click Apply and Close


Note: A “Velocity Specification Method” set to “Magnitude, Normal to Boundary” could also be used as an alternative to the specification of each velocity components.

3. Set the boundary conditions to pressure outlet for the outflow” zone. The values reported Thermal panel will be used in the event that flow enters the domain through this boundary (backflow condition). Set Backflow Total Temperature to 306.93 K.

Question: Why this value is proposed?

4. For the walls, in the Thermal panel, keep the default settings for a Heat Flux of 0 (adiabatic boundary condition).

2.5.3 File -> Write -> Case & Data...

Author	CFDLab@Energy		Page 10 of 33
CFD for Nuclear Engineering	Session 2		

3 Solver Execution

3.1 Initialization

Initialize the flow field using the “Hybrid Initialization”. This feature solves a simplified potential flow problem that provides an estimate of the flow field without inertial and viscous effects. It usually provides a good starting point for the simulation.

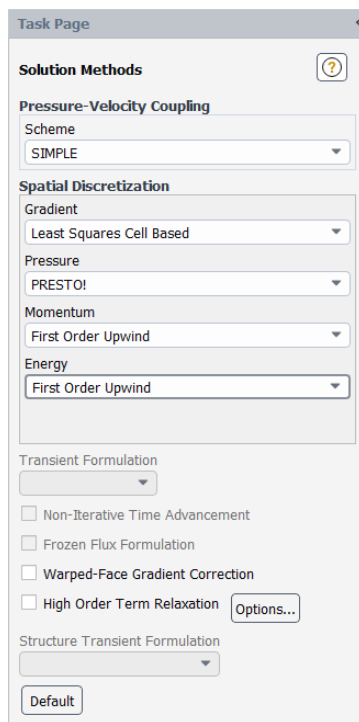
Solution -> Initialization -> Hybrid Initialization -> Initialize

Note: If a popup shows up to ask if you would like to discard the data, click OK.

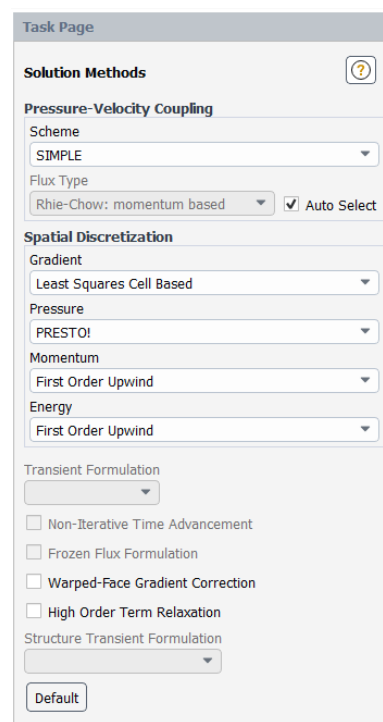
3.2 Solution Methods

3.2.1 Solution -> Methods (double click)

Set the various options such as in the following:



2020 R2




2021 R2

Note: since version 2021 R2 a new option “Flux Type” is available. Select “Auto Select” for convenience.

3.2.2 Solution -> Controls (double click)

Leave default values for the Under-Relaxation Factors.

Author	CFDLab@Energy		Page 11 of 33
CFD for Nuclear Engineering	Session 2		

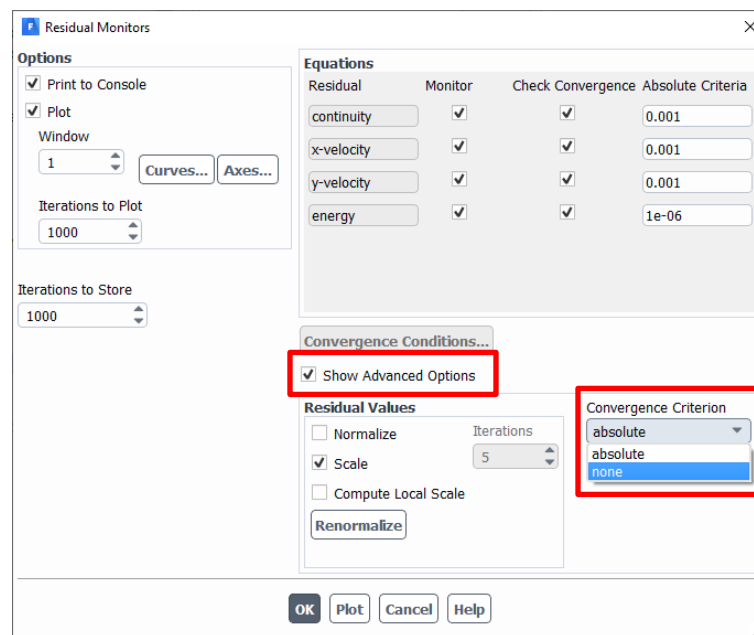
3.3 Monitoring the solution

Properly monitor the convergence of a simulation is FUNDAMENTAL to performing correctly a CFD study. By default, FLUENT assigns absolute criteria for the residuals of the solved equations that are almost NEVER suitable. The solution of the system of equations is iterative and the residuals represent, in plain language way, “the imbalance between what is solved and what is expected”. However, although the residuals are scaled or normalized, universal criteria for proper convergence are not achievable. The convergence criteria (on residuals) usually depend on the number of cells, physical models implemented, extension of the domain, etc.

You MUST therefore deactivate the convergence criteria in the Residual panel. This is the first step to perform in order to correctly monitor the solution.


3.3.1 Solution -> Monitors -> Residual (double click)

Set the Convergence Criterion to none as below (activate Show Advanced Options):



3.3.2 Set-up physical monitors of the solution

Since it is not possible to check the convergence of the simulation from the values of the residuals, a different approach is required. Depending on the boundary conditions of the simulation (in particular inlet and outlet) the user should monitor the quantities that are a “consequence” of the flow field and not quantities that are already “defined” by boundary conditions. Here for instance, the static pressure is assigned at the outlet so it is useless to monitor such quantity. Instead, the static (or total) pressure at the inlet(s) will be a consequence of the various losses (inertial and viscous) occurring in the simulated domain. Since these losses depend on the flow field, any change in the velocity field will result in a change in pressure at the inlet. This quantity is therefore FUNDAMENTAL to understand the “physical” convergence of the flow field.

Author	CFDLab@Energy	 Page 12 of 33
CFD for Nuclear Engineering	Session 2	

Note: On the contrary, when pressure is defined on both the inlet(s) and outlet(s), the quantity that will be a “consequence” of the flow field will be the mass flow rate.

On the other hand, the thermal mixing is only one-way coupled with the velocity field (fluid properties do not depend on temperature) therefore an additional monitor is required to check the convergence of the energy equation. Again, the quantity to monitor will be one that is a “consequence” according to the set-up of the thermal boundary conditions. Here, we can check the average outlet temperature but it will converge quite rapidly because the problem is adiabatic. Instead, we will check for the standard deviation of the outlet temperature that is representative of the strength of mixing that, in turn, depends on the flow field.

1. Set-up the monitor of the pressure at the inlet

Solution -> Report Definitions (double click)

New -> Surface Report -> Mass-Weighted Average...

Surface Report Definition

Name: in-p

Report Type: Mass-Weighted Average

Options:

- ☒ Per Surface
- Average Over: 1

Report Files [0/0]

Report Plots [0/0]

Create:

- ☐ Report File
- ☒ Report Plot
- Frequency: 1
- ☐ Print to Console
- ☐ Create Output Parameter

Field Variable: Pressure... Static Pressure

Surfaces: default-interior, fluid, inflow1, inflow2, outflow, sec-int, wall-ext, wall-inlet, wall-int1, wall-int2

Buttons: OK, Compute, Cancel, Help

2. Set-up the monitor of the standard deviation of temperature at the outlet

Solution -> Report Definitions (double click)

New -> Surface Report -> Standard Deviation...

Surface Report Definition

Name: out-std-t

Report Type: Standard Deviation

Options:

- ☐ Per Surface
- Average Over: 1

Report Files [0/0]

Report Plots [0/0]

Create:

- ☐ Report File
- ☒ Report Plot
- Frequency: 1
- ☐ Print to Console
- ☐ Create Output Parameter

Field Variable:

- Temperature...
- Static Temperature

Surfaces:

- default-interior
- fluid
- inflow1
- inflow2
- outflow
- sec-int
- wall-ext
- wall-inlet
- wall-int1
- wall-int2

New Surface

OK Compute Cancel Help

3.3.3 File -> Write -> Case & Data...

3.4 Start the calculation

3.4.1 Solution -> Run Calculation (double click)

Give 250 for the Number of Iterations and click Calculate.

3.4.2 Check for convergence

As mentioned previously, there are no universal metrics for judging convergence. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore, it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities and checking for mass and energy balances.

Here, it is reasonable to say that the simulation is converged after 150 iterations because the monitors of pressure and standard deviation of temperature have reached a “plateau” and do not change anymore. Corresponding residual values at 150 iterations are the following:

- Continuity: 2E-6
- X-Velocity: 3E-8
- Y-Velocity: 1E-7
- Energy: 4E-10

With the default convergence criteria, the simulation would have stopped at iteration 89 and represents the early stage of the “plateau” trend. This could be acceptable for this “lucky” case study but it is evident that it is risky.

To check mass and energy balances, go to

Results -> Reports -> Fluxes (double click)

Options: Mass Flow Rate

Boundaries: select all

Click Compute and check net results (should be a very low value)

Results -> Reports -> Fluxes (double click)


Options: Total Heat Transfer Rate

Boundaries: select all

Click Compute and check net results (should be a very low value)

Note: Further information about checking and controlling convergence will be given in next lessons and hands-on sessions.

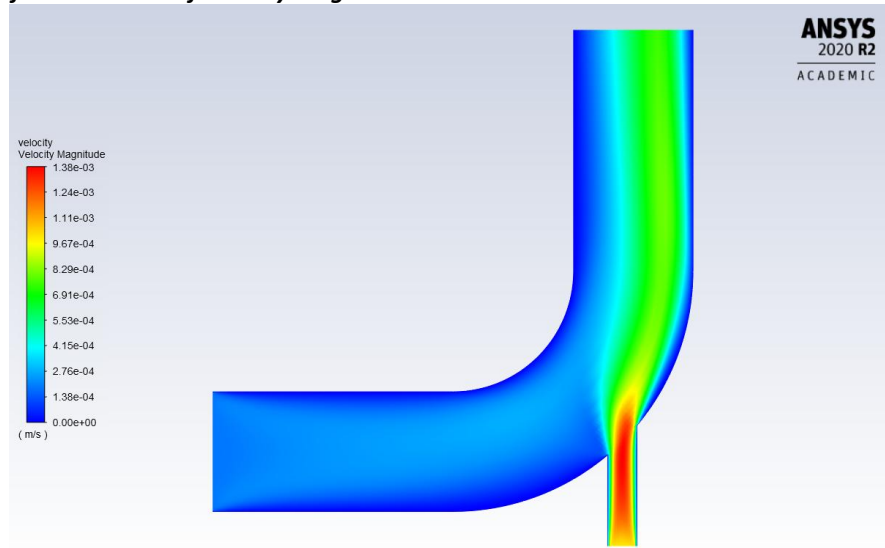
3.4.3 File -> Write -> Case & Data...

Author	CFDLab@Energy		Page 15 of 33
CFD for Nuclear Engineering	Session 2		

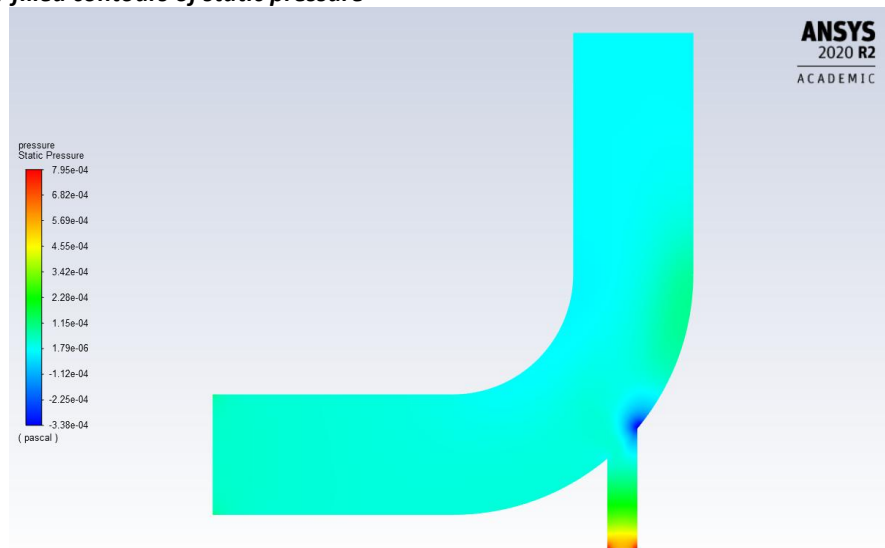
4 Post-processing

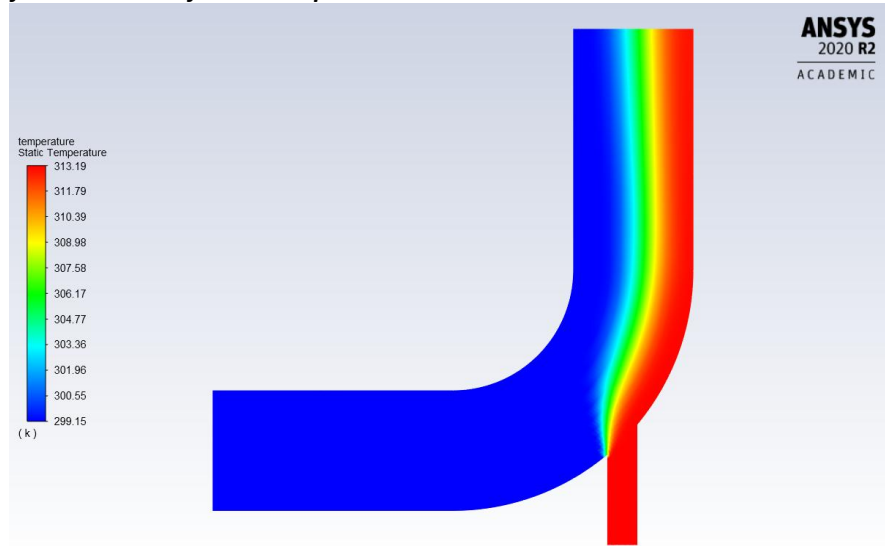
4.1 Flow field

4.1.1 Display filled contours of velocity magnitude

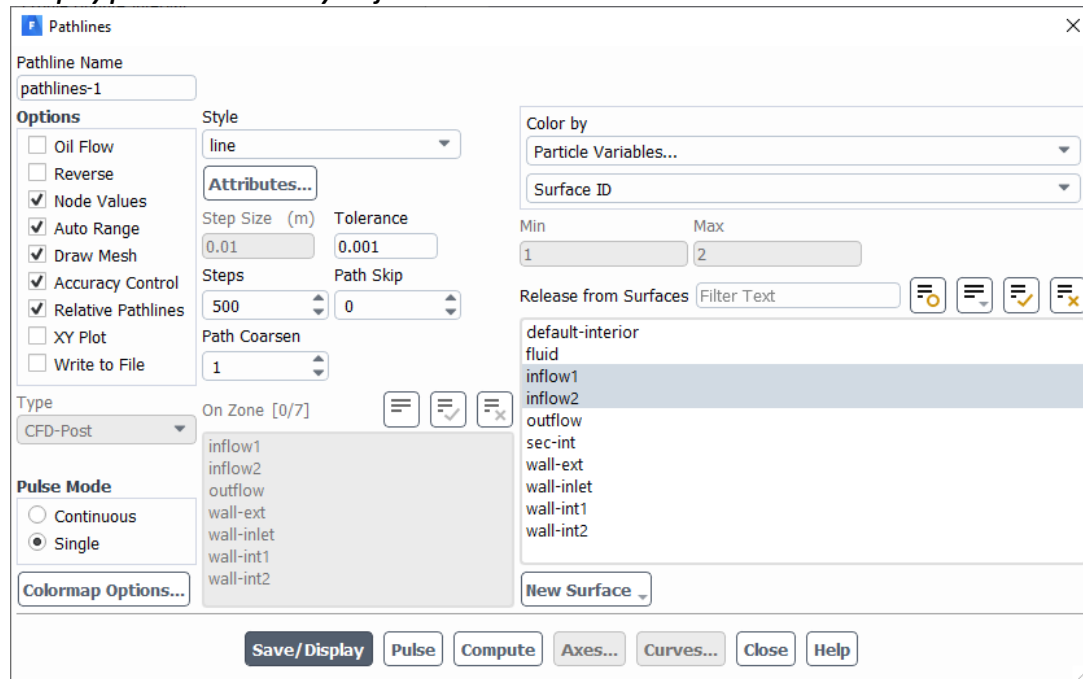



4.1.2 Display filled contours of static pressure

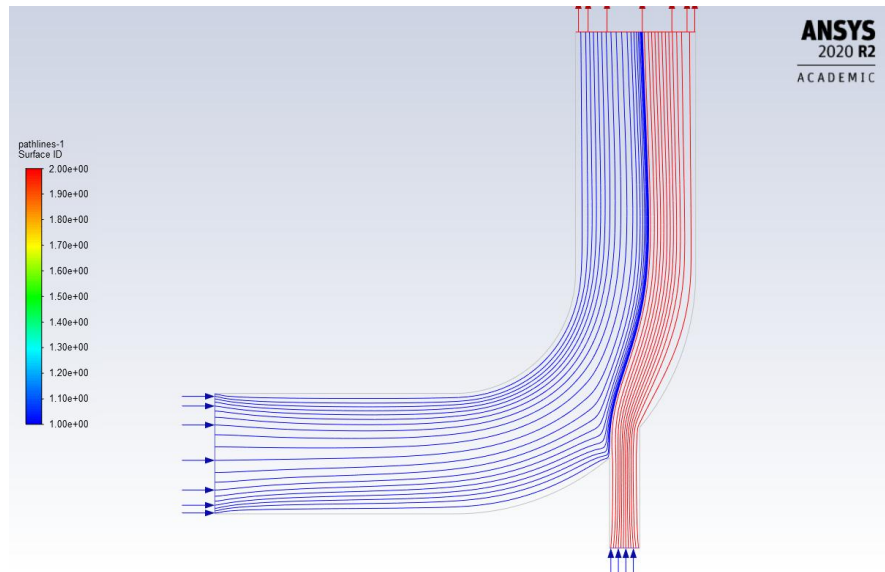


4.1.3 Display filled contours of static temperature

Please refer to Session 2 or to the session's recording for details on creating filled contours.

4.1.4 Display pathlines colored by Surface ID

Author	CFDLab@Energy	 Page 17 of 33
CFD for Nuclear Engineering	Session 2	



Question: What is the main mixing process going on in this case study?

4.2 Profiles

4.2.1 Display velocity profile at the outlet

Results -> Plots -> XY Plot (double click)

XY Plot Name: outlet-velocity

Y Axis Function: Velocity... -> Velocity Magnitude

Surfaces: select "outflow"

Click Curves...

Line Style -> Pattern: ----

Click Apply and Close

Click Save/Plot

Options: activate "Write to File" and "Order Points"


Click Write...

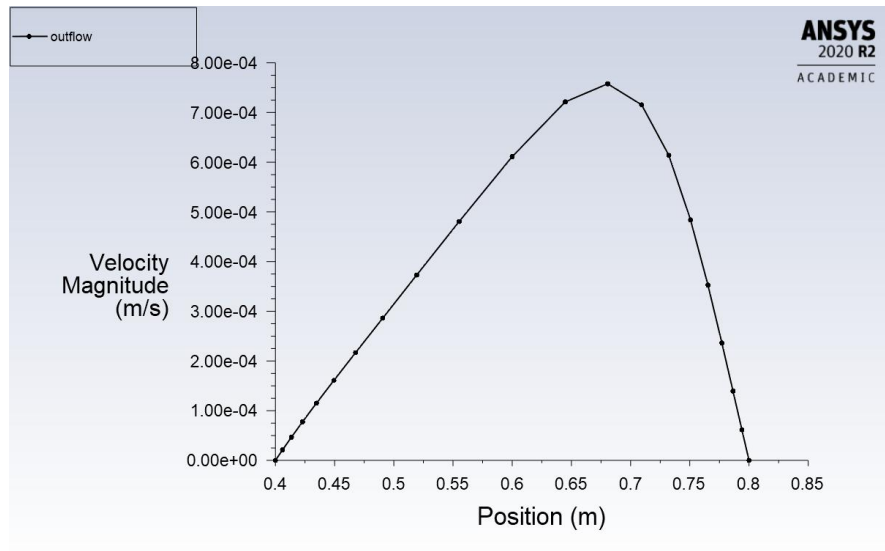
XY File: outlet-velocity_1st-order.xy

Click OK

Click Close

Note: the outlet profile is now saved in a text file that can be imported in Excel or reused later in FLUENT.

Author	CFDLab@Energy		Page 18 of 33
CFD for Nuclear Engineering	Session 2		



4.2.2 Display static temperature profile at the outlet

Results -> Plots -> XY Plot (double click)

XY Plot Name: outlet-temperature

Y Axis Function: Temperature... -> Static Temperature

Surfaces: select "outflow"

Click Curves...

Line Style -> Pattern: ----

Click Apply and Close

Click Save/Plot


Options: activate "Write to File" and "Order Points"

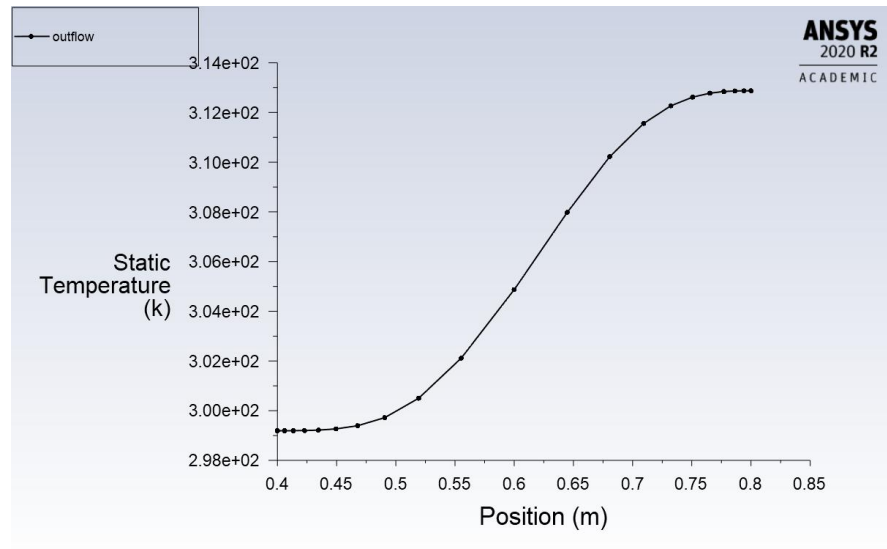
Click Write...

XY File: outlet-temperature_1st-order.xy

Click OK

Click Close

Author	CFDLab@Energy		Page 19 of 33
CFD for Nuclear Engineering	Session 2		



4.3 Integral quantities

4.3.1 Compute the pressure loss between the main inlet and the outlet

Results -> Reports -> Surface Integrals (double click)

Report Type: Mass-Weighted Average

Field Variable: Pressure... -> Total Pressure

Surfaces: select "inflow1" and "outflow"

Click Compute and check the results in the Console window

Subtract (manually) the outlet value to the inlet value.

4.3.2 Compute the average temperature at the outlet

Results -> Reports -> Surface Integrals (double click)


Report Type: Mass-Weighted Average

Field Variable: Temperature... -> Static Temperature


Surfaces: select "outflow"

Click Compute and check the results in the Console window.

Compare the computed value with the expected one.

Author	CFDLab@Energy		Page 20 of 33
CFD for Nuclear Engineering	Session 2		

Note: there is a bug in ANSYS FLUENT 2020 R2 that removes the “Temperature...” entry in the Field Variable dropdown menu. To recover the entry, a workaround is to write your case & data and read it afterwards or select and deselect a turbulence model.

Author	CFDLab@Energy		Page 21 of 33
CFD for Nuclear Engineering	Session 2		

5 Revision of the Model

Save your simulation with first-order upwind discretization scheme with a proper name for re-use. For instance
File -> Write -> Case & Data...

Case/Data File: Elbow_2D_1st-order.cas.h5

5.1 Enabling Second-Order Discretization

5.1.1 Set-up the simulation

Solution -> Methods (double click)

Spatial Discretization -> Momentum: Second Order Upwind

Spatial Discretization -> Energy: Second Order Upwind

5.1.2 Run the simulation

Solution -> Run Calculation (double click)

Run 250 additional iterations

Note: This second simulation will start from the previous simulation results unless it is initialized again. However, it is more convenient to start from the previous solution.


5.1.3 Check for convergence

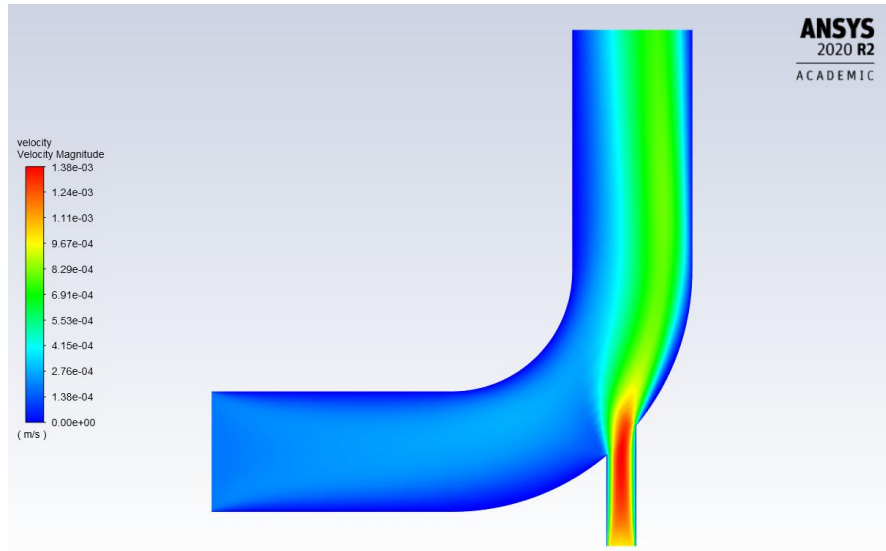
The simulation is converged after 100 iterations (global iteration number 350).

5.1.4 Post-processing

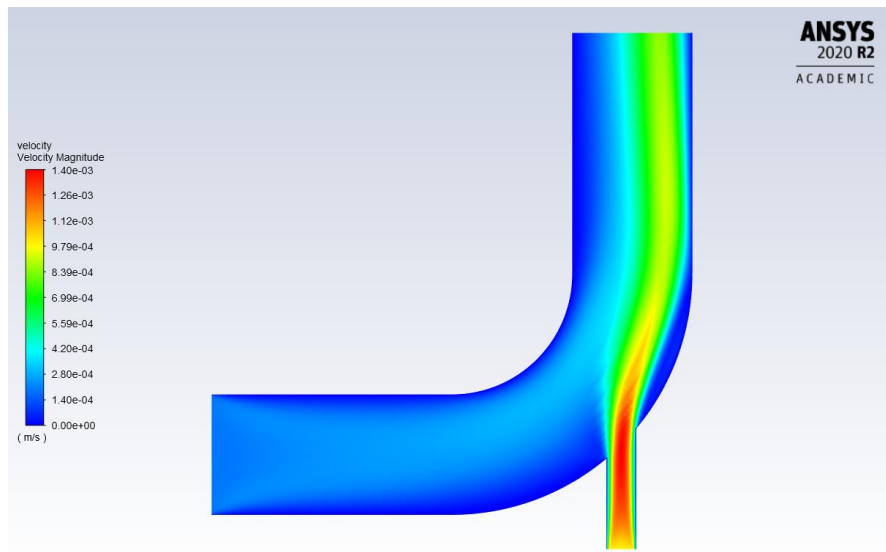
Perform the following:

- Compare the velocity field
- Compare the temperature field
- Compare the outlet velocity profile
- Compare the outlet temperature profile

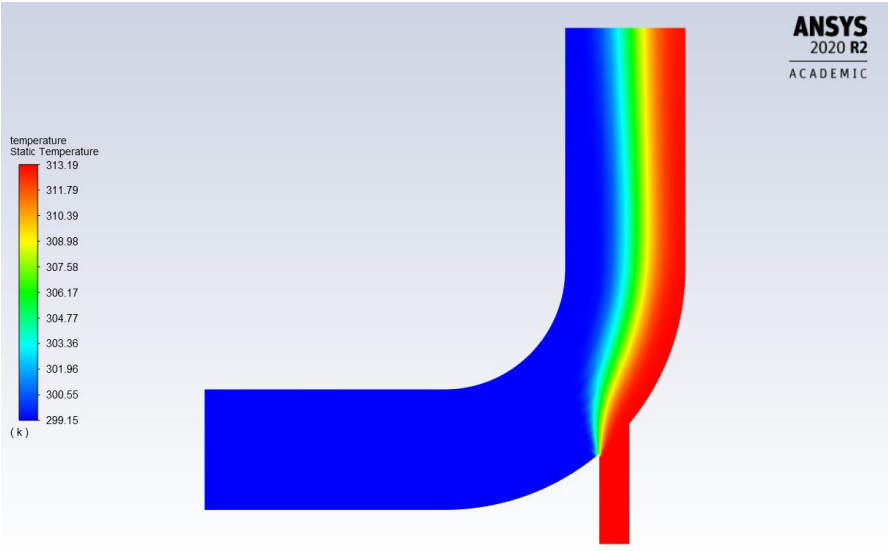
Author	CFDLab@Energy		Page 22 of 33
CFD for Nuclear Engineering	Session 2		



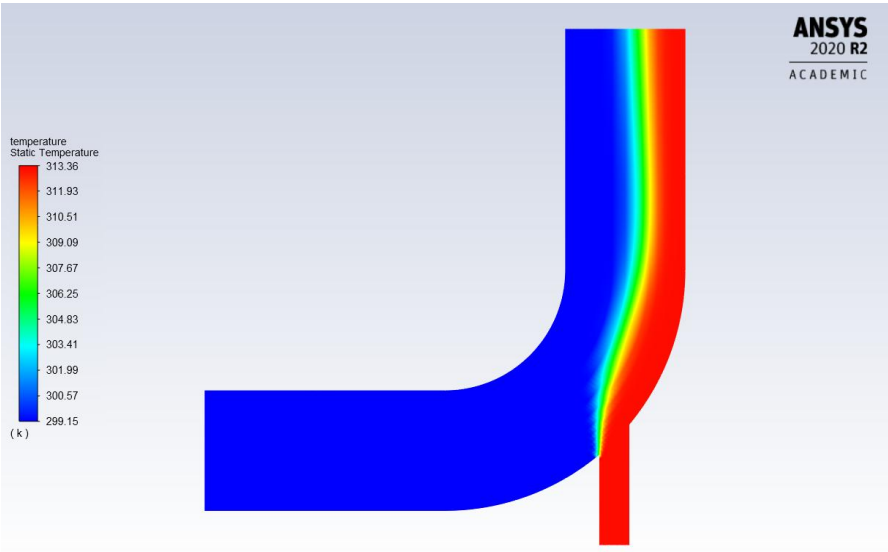
First-Order Upwind



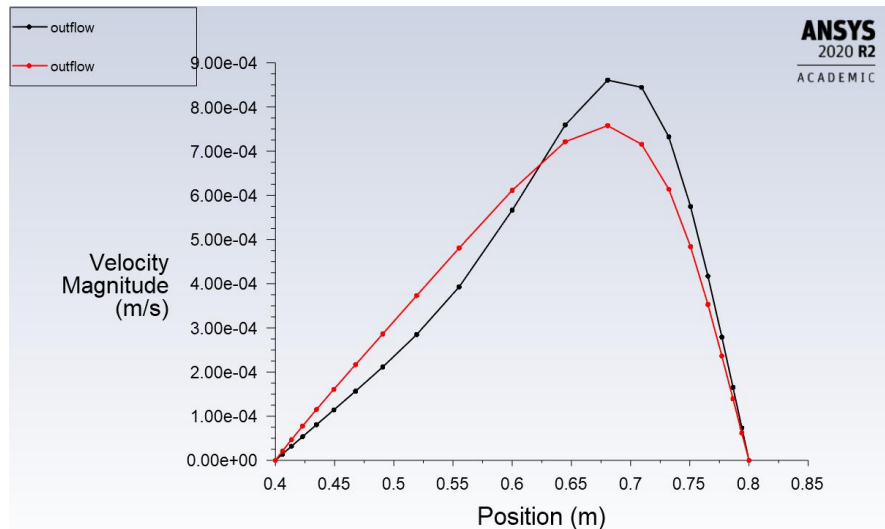
Second-Order Upwind



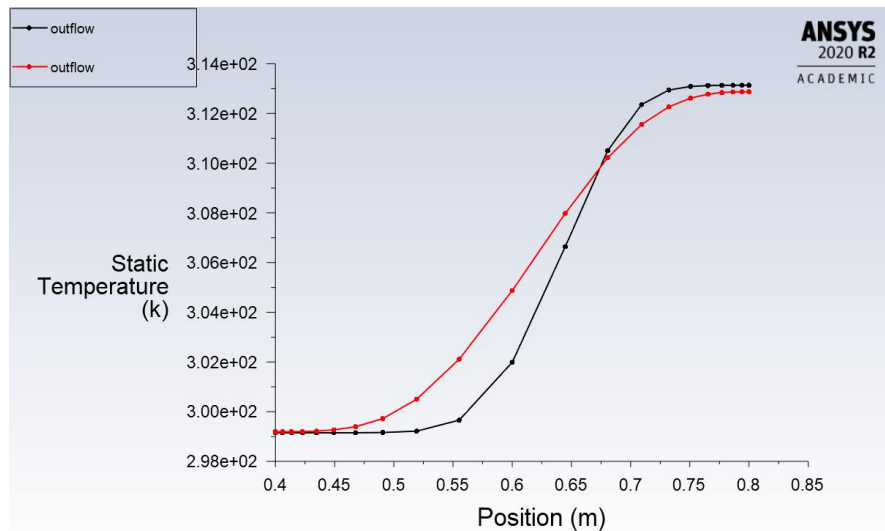
First-Order Upwind



Second-Order Upwind




Red: 1st-order Black: 2nd-order



Red: 1st-order Black: 2nd-order

Note: You can load a previous XY Plot using the “Load File...” button and selecting the corresponding file. Then in the “Curves...” settings you can apply the pattern for a given curve changing the “Curve #”. Remember to click Apply before changing curve number to keep the changes made.

Write the profiles to “outlet-velocity_2nd-order.xy” and “outlet-temperature_2nd-order.xy”.

Author	CFDLab@Energy		Page 25 of 33
CFD for Nuclear Engineering	Session 2		

5.2 Adapt (refine) the mesh

An alternative to increasing the order of accuracy of discretization schemes is to refine the mesh. Both approaches act on the truncation errors originating from the discretization but the former is usually more efficient, though slightly less stable.

Save the second-order upwind case to a different name and read the first-order upwind case.

File -> Write -> Case & Data...

Case/Data File: Elbow_2D_2nd-order.cas.h5

File -> Read -> Case & Data...

Case File: Elbow_2D_1st-order.cas.h5

5.2.1 Set-up the simulation

The mesh refinement features of FLUENT have been updated in the recent releases of the software and tends to be buggy. In addition, refinement in 2D will be deprecated in future releases.

In the following, a simple adaption of the mesh will be performed, i.e., all the mesh will be refined uniformly (one quad cell will be split into four quad cells). First we need to select all the cells for refinement. In FLUENT there is no explicit function for that but it can be done indirectly by selecting the region outside an empty region (i.e., everything). Then the selected cell region is used to refine the mesh. The following steps will perform such actions.

Domain -> Adapt -> Manual Refine/Coarsen...

Cell Registers -> New -> Region...

Options: Outside

Save/Display

Refinement Criterion: region_0

Click Adapt and ignore the warnings

You should now have a mesh containing 10560 cells. You can also display the mesh to check what happened.

5.2.2 Run the simulation

Solution -> Run Calculation (double click)


Run 250 additional iterations

5.2.3 Check for convergence

The simulation is converged after 150 iterations (global iteration number 400). Note also the slower convergence rate due to the increased number of cells.

File -> Write -> Case & Data...

Case/Data File: Elbow_2D_1st-order_refined.cas.h5

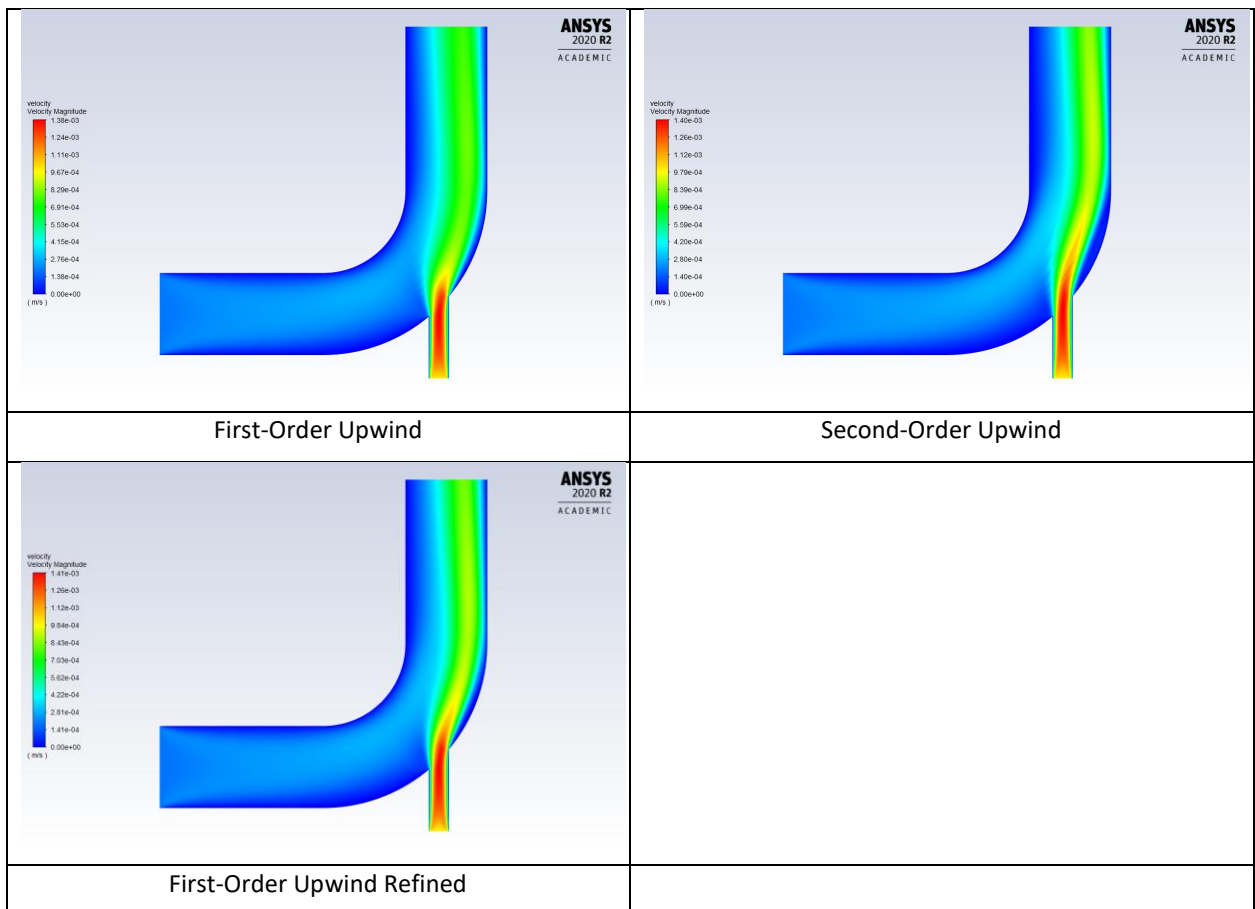
Author	CFDLab@Energy		Page 26 of 33
CFD for Nuclear Engineering	Session 2		

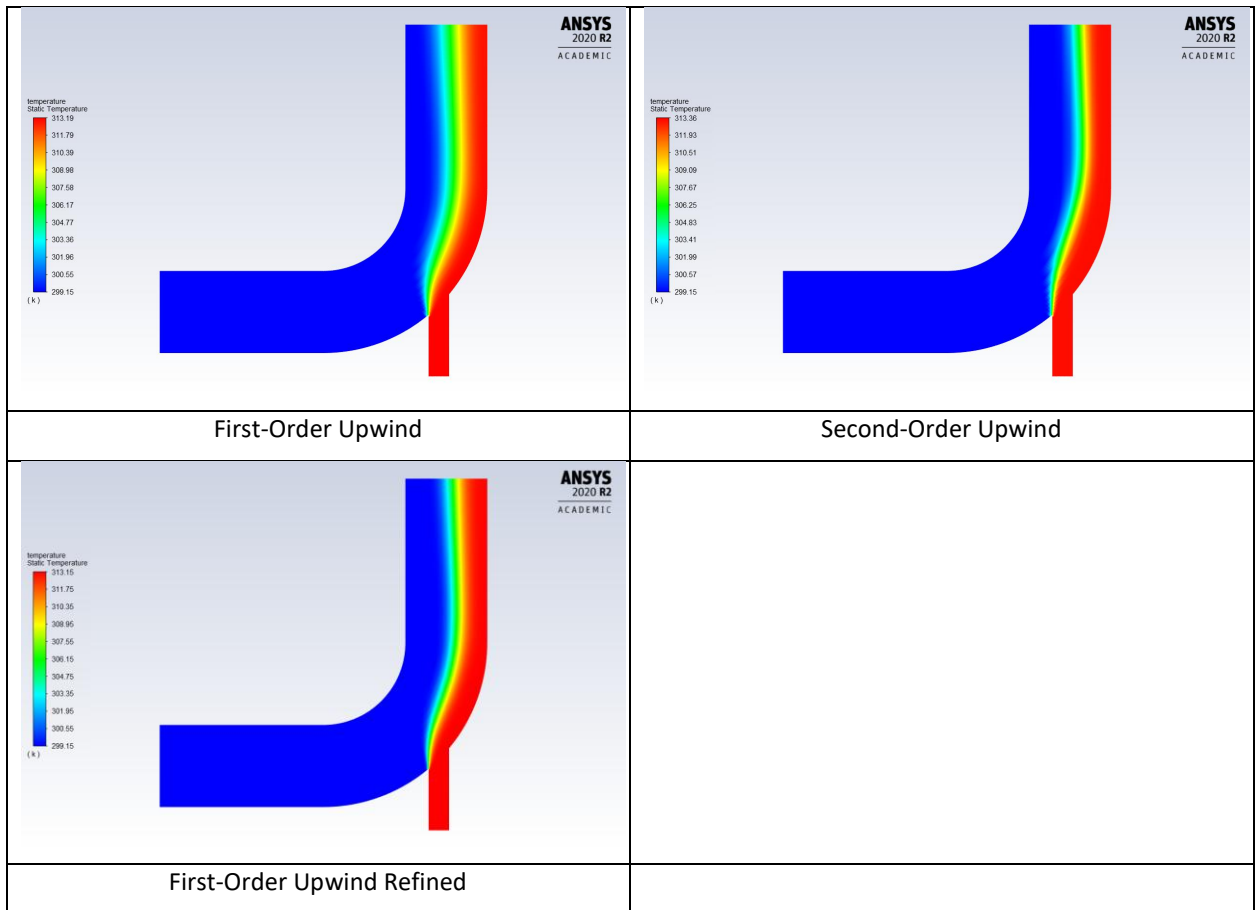
5.2.4 Post-processing

Perform the following:

- Compare the velocity field
- Compare the temperature field
- Compare the outlet velocity profile
- Compare the outlet temperature profile

Note: When doing a comparison between solutions, it is important to check that the interval range of the quantities match. Here for the filled contours of velocity we did not checked that the ranges match so for each picture there is a slightly different color scale. In future be aware of this problem and fix your interval ranges accordingly.





Now plot the velocity and temperature profiles at the outlet and write the data to “outlet-velocity_1st-order_refined.xy” and “outlet-temperature_1st-order_refined.xy”.

To compare the three profiles obtained in the different case settings, you can import the profiles into excel or use another FLUENT tool.

Results -> Plots -> Data Sources


Load File...

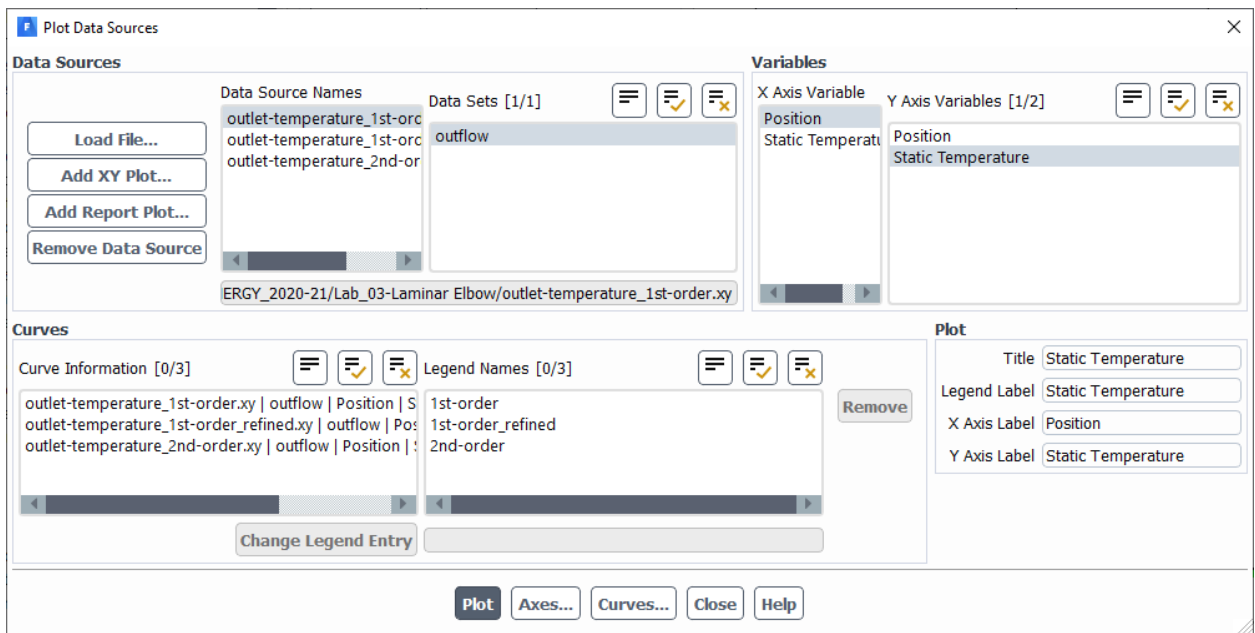
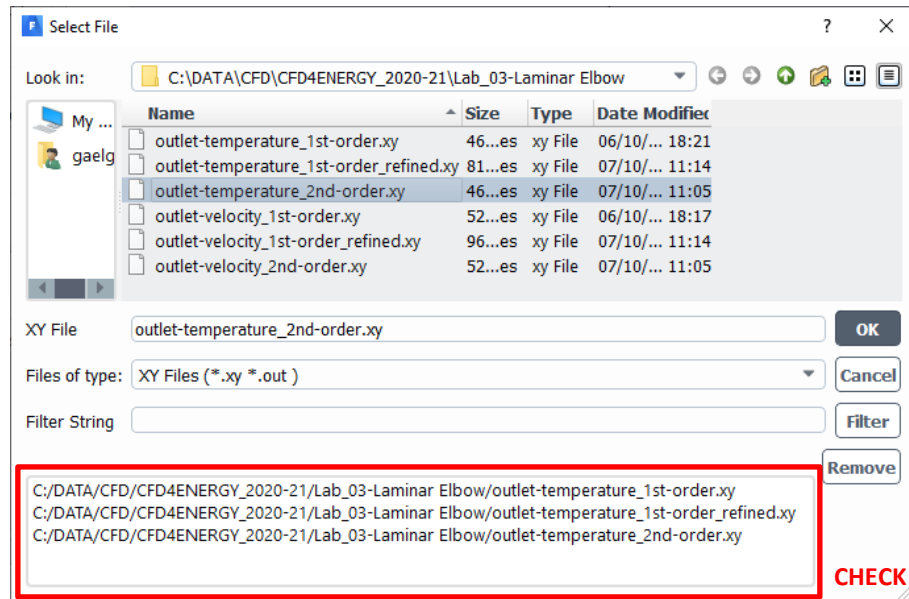
Select (using one left click for each entry) the various profiles to plot (see image below)

Check that the three profiles are selecting looking at the list at the bottom of the window

Click OK

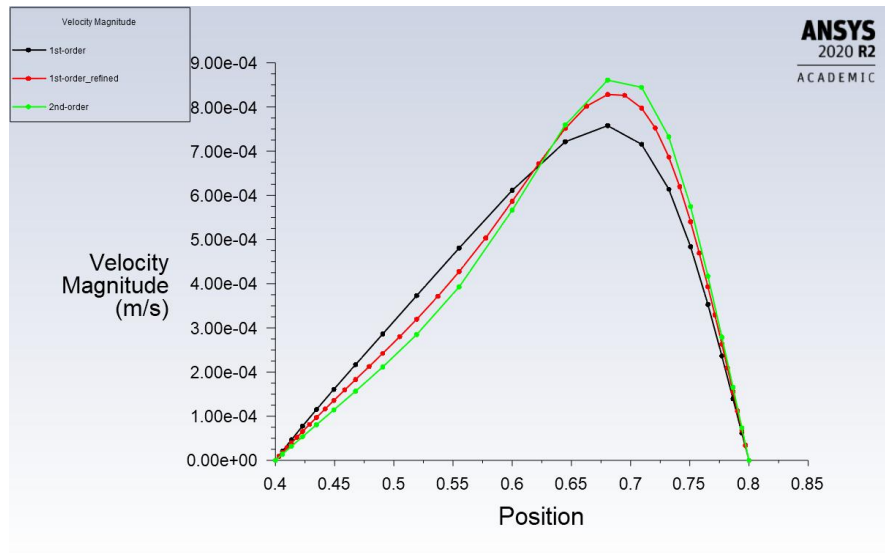
Change Legend Names using the “Change Legend Entry” tool

Author	CFDLab@Energy		Page 28 of 33
CFD for Nuclear Engineering	Session 2		

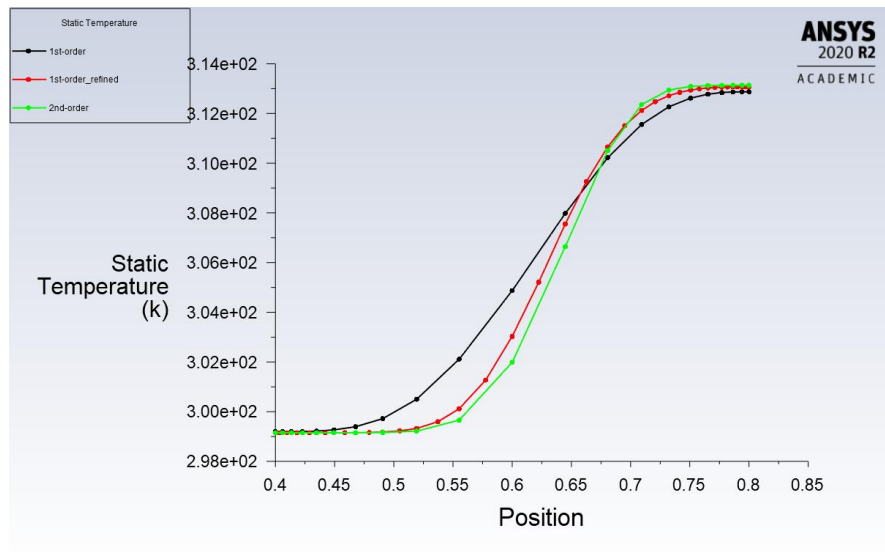


Change the line pattern in “Curves...”

Click Plot



Black: 1st-order Red: 1st-order_refined Green: 2nd-order



Black: 1st-order Red: 1st-order_refined Green: 2nd-order

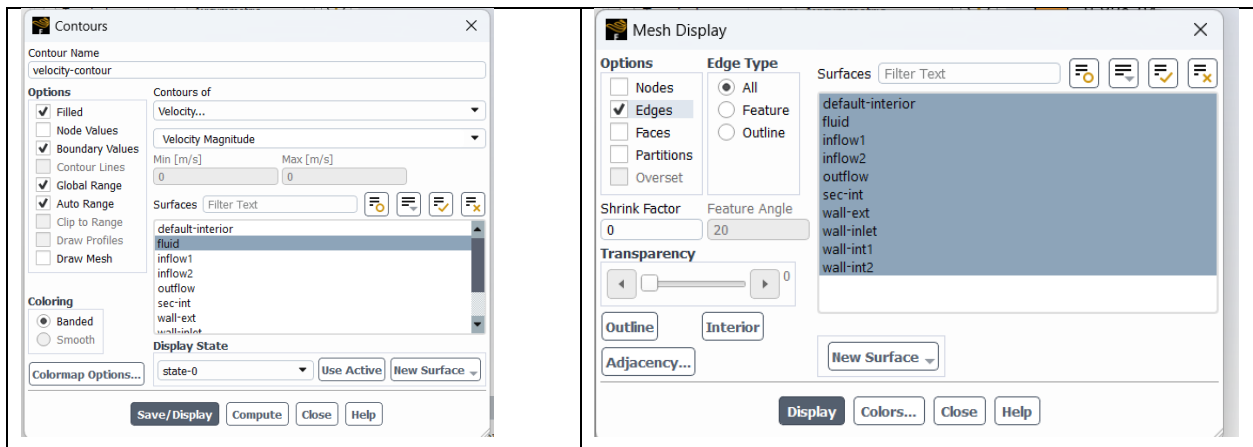
6 Home assignments

- Switch to Second-Order Upwind on the refined mesh and compare the results
- Question: Which approach is more efficient to increase the accuracy of the results: refining the mesh or increasing the order of accuracy?
- Question: Is the new solution (refined 2nd-order) optimal, i.e., fine enough?


7 Extras

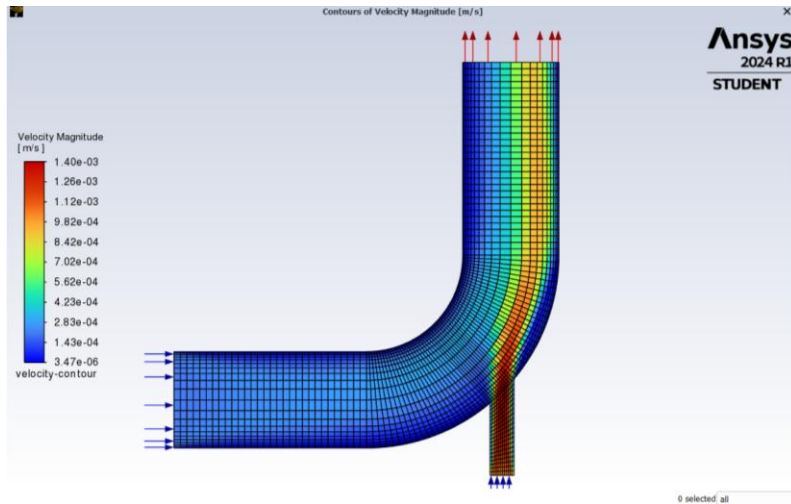
How to show a contour plot together with the grid?

In CFD the value of the main variables is located in the center of the cell so it is often necessary to visualize the solution and the cell boundaries. In Fluent the procedure would be to open a contour, click on “Draw Mesh” and on the pop-up window select the appropriate surfaces, check “Edges” and click “Display”. This will show the grid but remove the contour from the visualization. If you click on “Save/Display” in the Contours window while all the surfaces are selected the grid will be colored with the solution and the grid won’t be visible. One trick would be to deselect all the Surfaces and click on “Save/Display”.



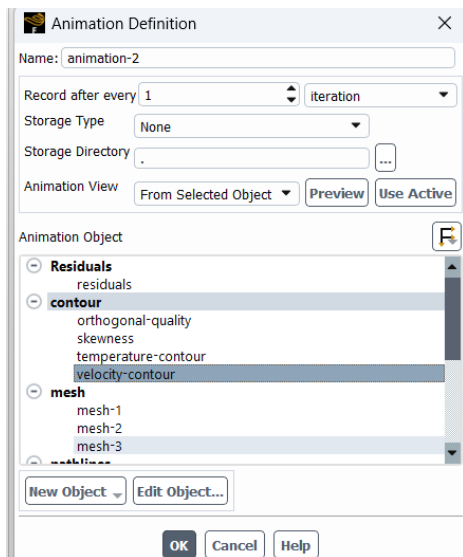
The visualization would be the following:

Author	CFDLab@Energy		Page 31 of 33
CFD for Nuclear Engineering	Session 2		




Can I see the evolution of the contours during the simulation?

Another useful way to show the results is monitoring the solution on a contour as the simulation advances. To do so we need first to create a “Solution Animation”. Go to Solution->Calculation Activities->Solution Animations (double-click)



Here it can be decided how often the record is done and the storage type. If we do not want to save the video but simply visualize, then “Storage Type” should be “None”. Select the contour that you would like to update at each Iteration/Time Step.

Author	CFDLab@Energy		Page 33 of 33
CFD for Nuclear Engineering	Session 2		