MIET1081 Heat Transfer
Computer Guide 2024

Instructors: Kiao Inthavong, Patrick Warfield-McAlpine, Brenda Almirall, Matthew Cook

INSTRUCTIONS:
Fill in the answers in the spaces provided.
Also sketch the results that you obtain from your simulations.
You will also perform hand calculations to help assist in your learning.
This booklet will serve as a revision for your tests.

Contents

1 Using FLUENT in MIET1081

1.1 Introduction to FLUENT's Text User Interface

ANSYS-Fluent is a computational fluid dynamics program that provides colourful predictions of the physical behaviour of fluids, by numerically solving equations that govern the flow. In this course we are not interested in the details of the software, but rather we are interested in visualising the physical behaviour of fluid and heat transfer. Heat transfer cases will be available each week and we will be using FLUENT's text-user-interface, (TUI) to set up and solve the heat transfer problem.

The menu system structure is similar to the directory tree structure of UNIX operating systems. When you first start FLUENT, you are in the root menu and the menu prompt is simply an arrow with a blinking cursor.

```
>
```

To generate a listing of the submenus and commands in the current menu, simply press <Enter>. You will get the following:

adjoint/	mesh/	report/
define/	parallel/	server/
display/	plot/	solve/
file/	print-license-usage/	views/

To execute a command, just type its name (or an abbreviation). Similarly, to move down into a submenu, enter its name or an abbreviation. When you move into the submenu, the prompt will change to reflect the current menu name. For example type display and press <Enter>. To generate a listing of the submenus and commands in the current menu, simply press <Enter> again. To enter a submenu, enter its name, (e.g., type in set) and press <Enter>:

```
> display
/display> set
/display/set>
```

To go back up a menu type q or quit at the prompt and press <Enter>.

```
/display/set> q
/display>
```

You can use the up and down arrow keys on your keyboard to go through recently used commands that are stored in history.

Let's define the material of the model as wood. Type define <Enter> followed by **b-c** <Enter>. Typing **b-c** is an abbreviation of the same as typing **boundary-conditions**. A command can be matched by matching an initial sequence of its phrases. If an abbreviation matches more than one command, then the command with the greatest number of matched phrases is chosen. If more than one command has the same number of matched phrases, then the first command to appear in the menu is chosen.

Type **solid** and <Enter>. A list of options follows. Type in the responses as shown in bold font below.

```
zone id/name [body] body
material-name [copper]: Change current value? [no] yes
material-name [copper]> wood
Specify source terms? [no] n
Specify fixed values? [no] n
Frame Motion? [no] n
Use Profile for Reference Frame X-Origin of Rotation-Axis? [no] n
Reference Frame X-Origin of Rotation-Axis (m) [0] 0
Use Profile for Reference Frame Y-Origin of Rotation-Axis? [no] n
Reference Frame Y-Origin of Rotation-Axis (m) [0] 0
Mesh Motion? [no] n
Solid Motion? [no] n
Enter yes or no to continue [no] n
```

This is tedious to do repetitively. We will use the journal function that records these commands as a list of

inputs. It is simply a text file that we will use to run all our commands. Enter the following command into Notepad and save the file as week1.jou. The file should be saved in the same directory as the FLUENT case (.cas) file.

```
/define/boundary-conditions/solid body yes wood n n n n 0 n 0 n n n
```

To run the command we read the journal file. Go to File → Read → Journal and then select the file. A journal file contains a sequence of FLUENT commands, arranged as they would be typed interactively into the program or entered through the graphical user interface.

1.2   Introduction to 1D Steady Heat Conduction

Conduction is a mode of heat transfer involving transfer of energy from more energetic particles of a substance to its neighbouring less energetic ones. Although conduction can occur in solids, liquids, or gases, it is most dominant in solids through the lattice connection of electrons. In gases and liquids, conduction occurs through collisions and diffusion of molecules. The rate of heat conduction is dependent on:

- its geometry, shape,
- its thickness,
- the material (which is its conductivity), and
- the temperature difference across the medium.
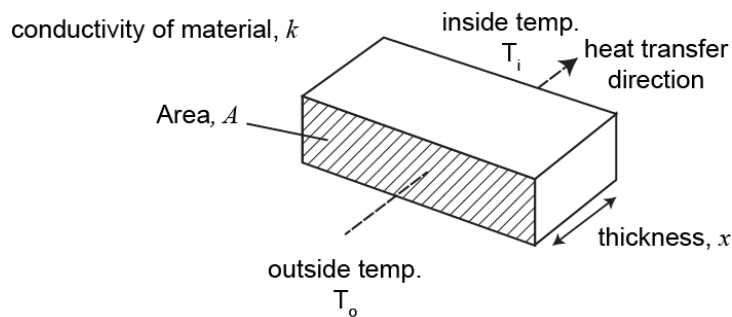
Figure 1.1: Parameters influenceing heat conduction

Consider heat conduction through a large plane wall such as a brick wall of a house. In such a geometry heat conduction can be approximated as one-dimensional since the conduction will be dominant in one direction and negligible in other directions. Fourier's law of heat conduction is used to describe this mode of heat transfer

$$\dot{Q} = -kA\frac{dT}{dx} \qquad\qquad \dot{q} = -k\frac{dT}{dx} \qquad\qquad (1.1)$$

Notes:

- The conduction heat transfer area is always normal to the direction of heat transfer.
- The thickness of the wall is independent of the heat transfer area.
- Since heat transfer occurs in the direction of decreasing temperature, the resultant temperature gradient becomes negative when temperature decreases in the direction of x i.e. increasing x. This means that we need a negative sign to ensure that heat transfer in the positive x direction gives a positive value.
- dT/dx is the temperature gradient, which describes how the temperature changes in the x-direction

The thermal conductivity, k is a measure of a material's ability to conduct heat. A high value for thermal conductivity means that the material conducts heat well, and a low value indicates that the material resists heat conduction acting like an insulator. Intuitively we know that metals get hot very quickly from exposure to heat, while water takes longer to heat up and this is confirmed by their conductivities: iron has $k = 80.2$ W/m·°C, and water has k=0.613 W/m.°C.

Thermal conductivities of some materials

Material	k, W/m·°C	Material	k, W/m·°C
Diamond	2300	Glass	0.78
Silver	429	Brick	1.31
Copper	387.6	Water, liquid	0.61
Gold	317	Wood, oak	0.173
Aluminium	202.4	Air gas	0.026
Iron	80.2	insulation foam	0.026

1.3 Using FLUENT to Predict a Steady Heat Conduction Problem

1.3.1 Open the file "CL01a.cas" file in Ansys-Fluent ensuring that the 2D option is selected.

Geometry

Side wall

Left wall

Right wall   H = 0.2 m

Side wall

L = 1.5 m
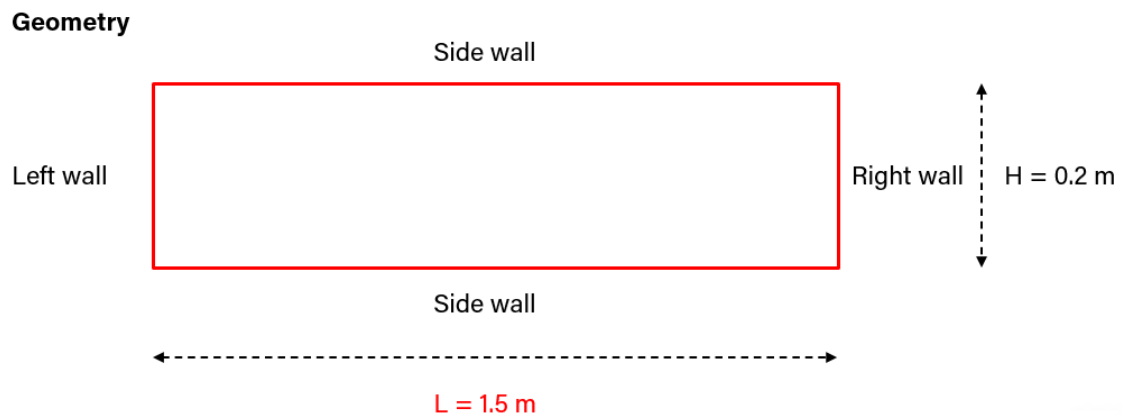
Figure 1.2: 2D geometry of a plane wall problem

1.3.2 Begin creating your journal file by defining the material. The name of the material can be one of the following brick, wood, copper, aluminum. Copy the following line, where you change the red text to the material you want:

→ `/define/boundary-conditions/solid body yes copper n n n n 0 n 0 n n n`

1.3.3 Define the boundary conditions.
- Boundary may be left, right.
- Boundary conditions may be temperature, heat-flux.
- We also need to enter the value for the boundary condition selected: T, [°C] for temperature or $\dot{q}$, [W/m²] for heat-flux

heat-flux example:

→ `/define/boundary-conditions/wall left 0 n 0 n y heat-flux n 3250 n 1`

temperature example:

→ `/define/boundary-conditions/wall right 0 n 0 n y temperature n 30 n 1`

1.3.4 Solving the model
copy the following two lines. Make sure they are separate in the journal file.

→ `/solve/initialize/init y`

→ `/solve/iterate/ 50`

1.3.5 Add a comment at the end to notify us that the simulation has completed.
$\rightarrow$ `(display "FINISHED")`

This is all the commands we need for now to run our simulation. Investigate the following cases.

Case 1 - Heat Conduction in a copper bar (k =387.6 W/m^{0}C, you don't need to enter this value explicitly since it is contained within the FLUENT material library)with a heat flux boundary condition:
- Left end with a constant heat flux 3250 W/m^2
- Right end has a constant temperature 30^{0}C

Case 2 - Heat Conduction in a copper bar (k =387.6 W/m^{0}C) with temperature boundary conditions:
- Left end has a constant temperature 250^{0}C
- Right end has a constant temperature 30^{0}C

Case 3 - Heat Conduction in a brick bar (k =1.31 W/m^{0}C)with a heat flux boundary condition:
- Left end with a constant heat flux 3250 W/m^2
- Right end has a constant temperature 30^{0}C

Case 4 - Heat Conduction in a brick bar (k = 1.31 W/m^{0}C) with temperature boundary conditions:
- Left end has a constant temperature 250^{0}C
- Right end has a constant temperature 30^{0}C

As an example, the journal file for Case 1 is:

```
/define/boundary-conditions/solid body yes copper n n n n 0 n 0 n n n
/define/boundary-conditions/wall left 0 n 0 n y heat-flux n 3250 n 1
/define/boundary-conditions/wall right 0 n 0 n y temperature n 30 n 1
/solve/initialize/init yes
/solve/iterate/ 50
/(display "FINISHED")
```

1.3.6 To visualise the results, we can use the text input to plot temperature contours. However one problem is that sometimes we don't know the contour range produced from our results. For example from the root menu type:
$\Rightarrow$ display $\Rightarrow$ contour $\Rightarrow$ temperature
And now we have to enter the minimum and maximum range values for the contour. If you cannot see the contour, try the following command:
$\Rightarrow$ display $\Rightarrow$ views $\Rightarrow$ default-view

You can use the graphical user interface. This is under Graphics and Animations.

HOMEWORK QUESTIONS

Q1 Using Fourier's heat conduction equation calculate the following

	Case 1	Case 2	Case 3	Case 4
T_{left}		250		250
$\dot{q}$	3250		3250	

Q2 Plot each of the temperature gradient for the four cases.
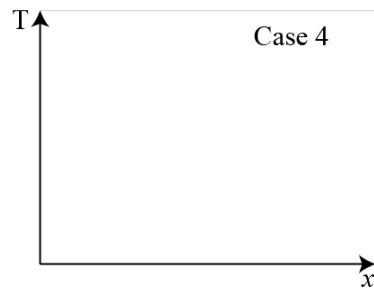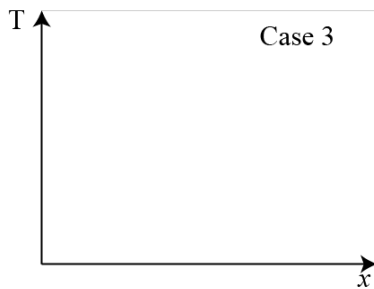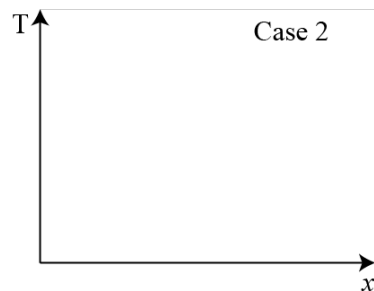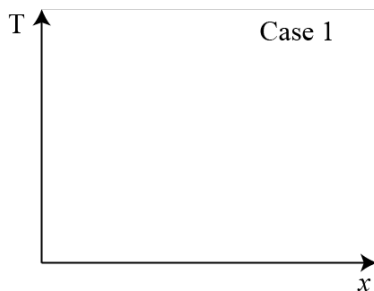
If you've forgotten how to visualise contours or xyplots visit `http://www.youtube.com/watch?v=4tgKDa0NHr0`

Q3 Why are the *dt/dx* the same for Case 2 and Case 4?

_____

_____

Q4 Which *dt/dx* has the highest gradient? Which has the flattest or smallest gradient? Why?

_____

_____

Q5 If we impose left and right wall temperature boundary conditions for the four different materials (brick, wood, steel, and copper) what do their *dt/dx* look like. Explain why in terms of Fourier's conduction equation.

_____

_____

2 Conduction and Thermal Resistance

2.1 Equations

Heat conduction through a plane wall can be described as

$$\dot{Q} = kA\frac{T_a - T_b}{L} \qquad\qquad \dot{Q} = \frac{T_a - T_b}{R_{cond}} \qquad\qquad \rightarrow \qquad\qquad R_{cond} = \frac{L}{kA} \tag{2.1}$$

If we are after a heat flux then the equations become

$$\dot{q} = k\frac{T_a - T_b}{L} \qquad\qquad \dot{q} = \frac{T_a - T_b}{R_{cond}} \qquad\qquad \rightarrow \qquad\qquad R_{cond} = \frac{L}{k} \tag{2.2}$$

For heat transfer in series, the resistance is

$$R_{series} = R_1 + R_2 + ... + R_n \tag{2.3}$$

For heat transfer in parallel, the resistance is

$$\frac{1}{R_{parallel}} = \frac{1}{R_1} + \frac{1}{R_2} + ... + \frac{1}{R_n} \tag{2.4}$$

2.2 Exercises

2.2.1 Open the File

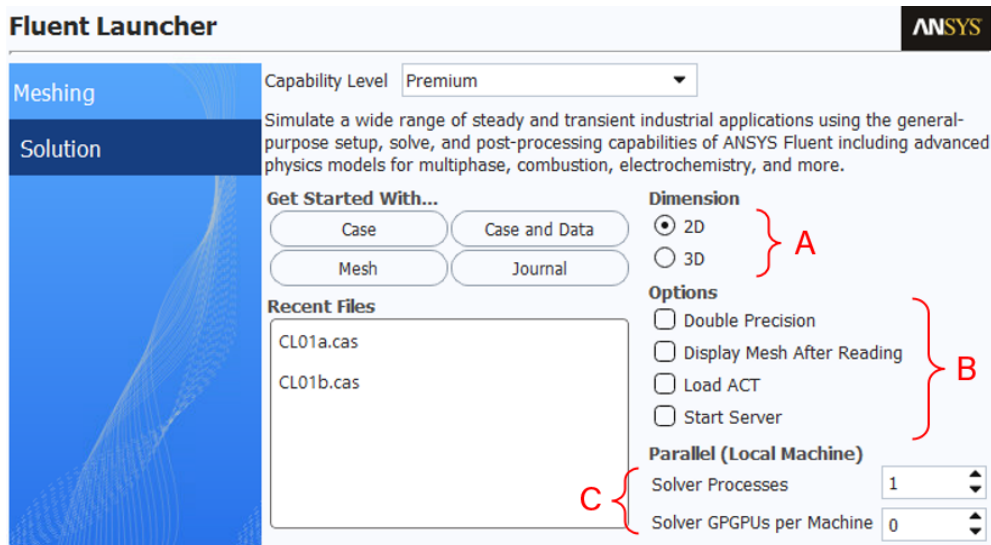Open "CLO1b.cas" file in FLUENT in 2D mode. In case you forgot, here's the splash screen:

Figure 2.1: ANSYS Fluent Launcher splash screen

A: The models we work with will all be 2D
B: Select Display Mesh After Reading
C: The computers will allow parallel processing to speed up your calculations. How many processors it allows is dependent on the computer's capabilities.

The geometry we are looking at is shown below:
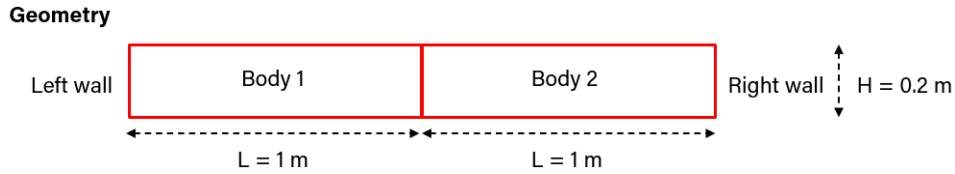
Figure 2.2: 2D geometry of a two materials in series

2.2.2 TUI Commands

Defining the material for body1 and body2:

→ `/define/boundary-conditions/solid body1 yes copper n n n n 0 n 0 n n n`

Options 1: `body1, body2`

Options 2: `copper, steel, brick`

Defining temperature boundary conditions:

→ `/define/boundary-conditions/wall left 0 n 0 n y temperature n 50 n 1`

Options 1: `left, right`

Options 2: `temperature, heat-flux`

Options 3: numerical values for boundary condition (e.g. 50)

Solving the model

copy the following two lines. Make sure they are separate in the journal file.

→ `/solve/initialize/init yes`

→ `/solve/iterate/ 50`

Add a comment at the end to notify us that the simulation has completed.

→ `/(display "FINISHED")`

2.2.3 Cases

Case 1:

Body1: Copper (k_{copper}=387.6 W/m°C), Body2: Steel (k_{steel}=16.27 W/m°C)

Temperature: Left = 190°C, Temperature: Right = 20°C

Case 2:

Body1: Copper (k_{copper}=387.6 W/m°C), Body2: Brick (k_{brick}=1.31 W/m°C)

Temperature: Left = 190°C, Temperature: Right = 20°C

Case 3:

Body1: Copper (k_{copper}=387.6 W/m°C), Body2: Brick (k_{brick}=1.31 W/m°C)

Temperature: Left = 30°C, Heat flux: $\dot{q} = 800 W/m^2$

```
/define/boundary-conditions/solid body1 yes copper n n n n 0 n 0 n n n
/define/boundary-conditions/solid body2 yes steel n n n n 0 n 0 n n n
/define/boundary-conditions/wall left 0 n 0 n y temperature n 190 n 1
/define/boundary-conditions/wall right 0 n 0 n y temperature n 20 n 1
/solve/initialize/init yes
/solve/iterate/ 50
/(display "FINISHED")
```

2.3 Questions

Q1 Using the thermal resistance equations (Eqns 2.1 to 2.4) calculate the heat transfer, the and middle interface temperature for each case. The conductivities of the materials used are: k_{copper}=387.6; k_{steel}=16.27; k_{brick}=1.31:

	Case 1	Case 2	Case 3
T_{int}			
$\dot{q}$			800 W/m^2

Q2 Plot the temperature gradient profiles for each case
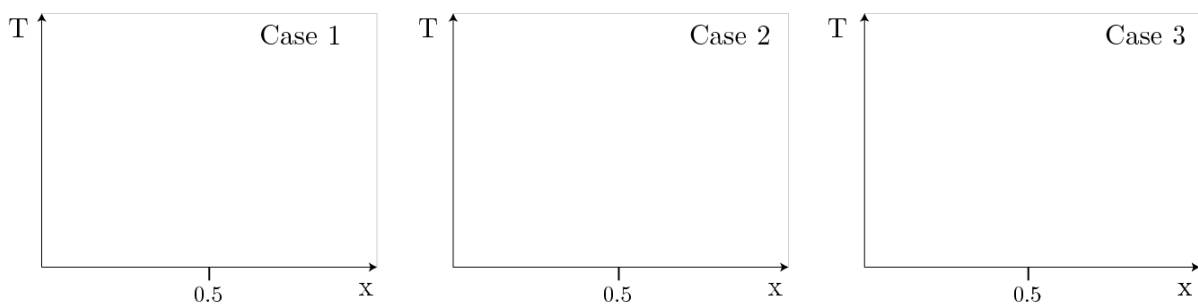
Q3 Explain what the temperature gradient of copper looks like for all the three cases and why this happens.

Q4 Calculate by hand the heat transfer rate ($\dot{q}$) for a i) copper (k = 387.6 W/m.K) and steel (k = 16.27 W/m.K), and ii) copper and brick (k = 1.31 W/m.K), in parallel mode,having the same temperature boundary conditions used earlier (190 ^{0}C, and 20 ^{0}C).

Copper and Steel: $\dot{q}$ = _____

Copper and Brick: $\dot{q}$ = _____

3 Multidimensional conduction

We have considered the heat conduction problems as one-dimensional geometries, and this is a very common assumption that can be made in practice on many occasions for crude and quick calculations. However, sometimes we encounter more complicated geometries that are two- or three-dimensional and no simple solutions are available. For cases between two surfaces that are at constant temperatures T_1, and T_2, the conduction shape factor can be used, $\dot{Q} = Sk(T_1 - T_2)$ where S: conduction shape factor, and k: thermal conductivity of the medium between the surfaces. The conduction shape factor depends on the geometry only and is applicable for pure conduction (i.e. no convection or radiation heat transfer influence).
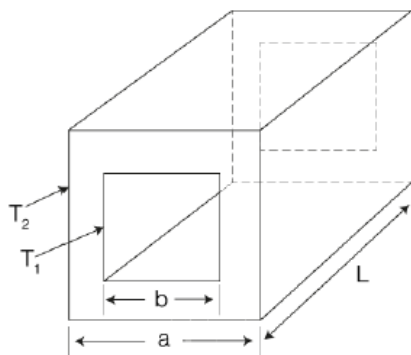
3.1 Solve the following problem

Consider a hollow concrete (k = 0.9W/mK) square duct with side widths of 5m, a wall thickness of 20cm, and a depth of L = 10m. The temperature on the outside surface is 100°C and inside surface is 5°C. In the space below calculate the heat transfer rate using the shape conduction factor:

For $a/b > 1.4$,

$$S = \frac{2\pi L}{0.93 \ln(0.948\, a/b)}$$

For $a/b < 1.41$,

$$S = \frac{2\pi L}{0.785 \ln(a/b)}$$

Q = _____ (put your answer here)

Run the simulation Open "*Lab03-square-5mx5m.cas*" file in FLUENT in 2D mode. The materials you can choose have the following properties: concrete k = 0.9 W/mK; steel k = 16.27 W/mK; copper k = 387.6 W/mK. After your simulation get the heat flux (you can also view the contour plots).
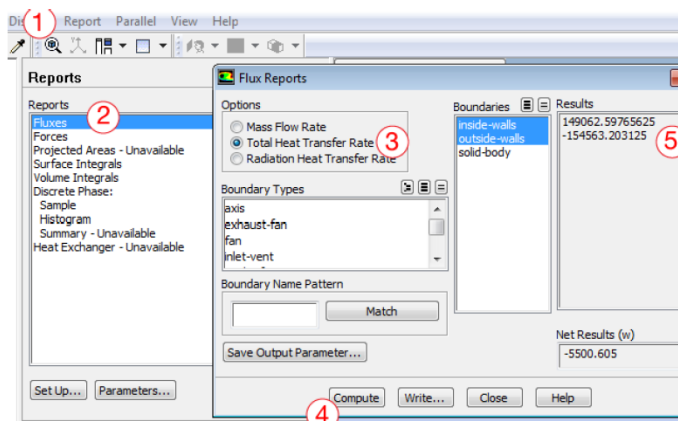
1. Report
2. Fluxes
3. Total Heat Transfer Rate
4. Compute
5. * *read your results*

3.2 Simulate the rectangular hollow duct

Now what if the square is not completely square? For example, it's a rectangular shape with dimensions of 4m x 5m. And what happens if we have a heat transfer coefficient at the boundaries instead of a known temperature? We can't use the conduction shape factor anymore because the conditions don't quite match.

If we need quick calculations and want to apply the shape conduction factor, what magnitude of error would we get? Open "Lab03-rectangle-5mx4m.cas" file in FLUENT in 2D mode. Set up your case as before (use the same conditions). Compare the heat flux value in this conduction with the value from the tutorial.

Q = _____

Difference with conduction shape factor = _____

Compare with Tutorial problem:
In the tutorial class there is a question regarding this geometry. A 5-m-wide, 4-m-high and 40-m-long kiln used to cure concrete pipes is made of 20-cm-thick concrete walls (k = 0.9 W/mK). The kiln is maintained at 40°C by injecting hot steam into it. The convection heat transfer coefficients on the inner and the outer surfaces of the kiln are 3000W/m^{2}K and 25W/m^{2}K, respectively. Determine the rate of heat loss from the kiln when the ambient air is at 4°C. You will use a technique to solve this problem, and the answer from the tutorial is 116076 W.

4 Transient Conduction

4.1 Equations

$$\text{Bi} = \frac{hL_c}{k} \qquad\qquad\qquad \tau = \frac{\alpha t}{L^2}$$

The temperature at x^* and t can be found from:

$$\theta = \frac{T - T_\infty}{T_i - T_\infty} = A_1 \exp\left(-\lambda_1^2 \tau\right) \cos\left(\lambda_1 \frac{x}{L}\right)$$
$$T = (T_i - T_\infty)A_1 \exp\left(-\lambda_1^2 \tau\right) \cos\left(\lambda_1 \frac{x}{L}\right) + T_\infty$$

where the corresponding constants are λ_1, and A_1 from Table 4.2 of the prescribed textbook.

4.2 Exercises

4.2.1 Read the File

Create a new working directory on the student H: drive. Download "TransientConduction.cas.gz" and place it in your new working directory. Open the Fluent launcher, select your working directory and start Fluent under 2D conditions.
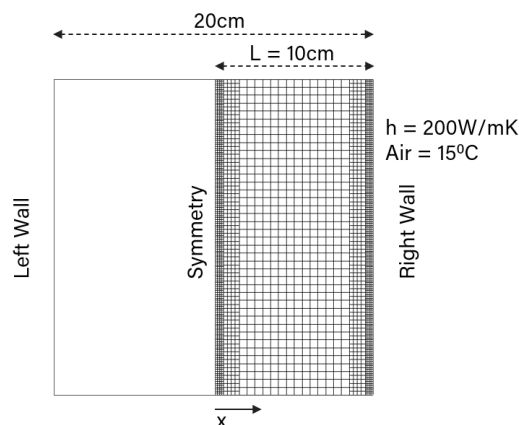
Figure 4.1: 2D Transient geometry

The computational domain makes use of a symmetry boundary condition to minimise the computational resources. The right hand side of the domain is used for the simulation.

4.3 TUI Commands

Material selection
For copper use the following command:
```
/define/b-c/solid solid yes copper n n n n 0 n 0 n n n
```
For brass use the following command:
```
/define/b-c/solid solid yes brass n n n n 0 n 0 n n n
```
Initial temperature
```
/solve/initialize/set-defaults/temperature 650
```
Transient controls (Time steps)
```
/solve/set t-c n-o-t-s 120 t-s-s 1 m-i-p-t-s 8
```
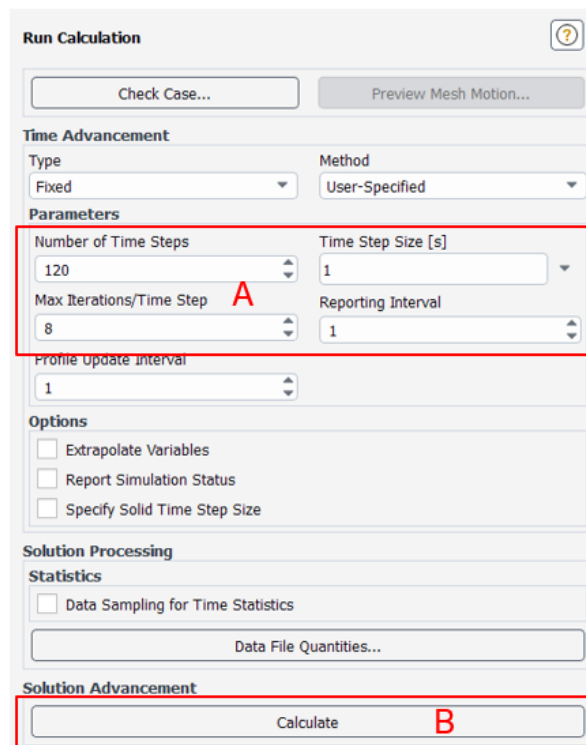
Figure 4.2: Graphics user interface. The highlighted box in red highlights control for: **(A)** Transient modelling parameters e.g. Number of Time Steps, Time Step Size [s], Max Iterations/Time Step, **(B)** Run calculation

4.4 Cases

Boundary Condition at the Right wall

- Heat transfer coefficient (h): 220 W/m^{2}K
- Free stream temperature (T_∞): 15°C

4.4.1 Case 1: Use the TUI command to select Brass as your material.

- Material: Brass
- Density (ρ): 8530 kg/m^3
- Specific heat (cp): 380 J/kgK
- Conductivity (k): 110 W/mK

4.4.2 Case 2: Use the TUI command to select Copper as your material.

- Material: Copper
- Density (ρ): 8978 kg/m^3
- Specific heat (cp): 381 J/kgK
- Conductivity (k): 397.6 W/mK

NOTE: Re-initialise before solving case 2!

4.5 Homework Questions

Q1 Using CFD determine the temperature $T(x, t)$, where $t = 120s$ and the location, x is given by its non-dimensionlised distance x^*:

x^*	Case 1	Case 2
0		
0.5		
1		

Q2 Compare your CFD results with the analytical results you obtained from completing this week's Pre-Class question (question T3.1). Comment on some possible causes of the discrepancy between the analytical and the CFD results?

Q3 Determine if we can we use the lumped system analysis for each case? What temperature $T(t)$ do you get from using the lumped system method for each case. Compare this with the CFD results.

Q4 From Question 3, comment on the relevance of the Biot number Bi and what it represents in transient conduction analysis?

Q5 Comment on what the Fourier number τ and what it represents?

5 Forced Convection over Flat Plates

The flow simulation setup will be demonstrated and you need to perform the simulations yourself with individual values for velocity. Run all your simulations in 2D and double-precision. Using 1-core CPU (i.e. in serial mode) it's okay for small cases.

5.1 Laminar flow over a flat plate

a.) Use the Fluent cas file 'flat_plate_laminar.cas.h5'. The domain is shown below. The measurement region is only along the flat plate. The pre-flow region (L = 0.2m) is used to help setup and stabilise the flow field. The start of the flat plate region begins at x = 0 m and ends at x = 0.5 m. The plate temperature is 20°C (293.15 K)and the inlet fluid temperature is 80°C (353.15 K).
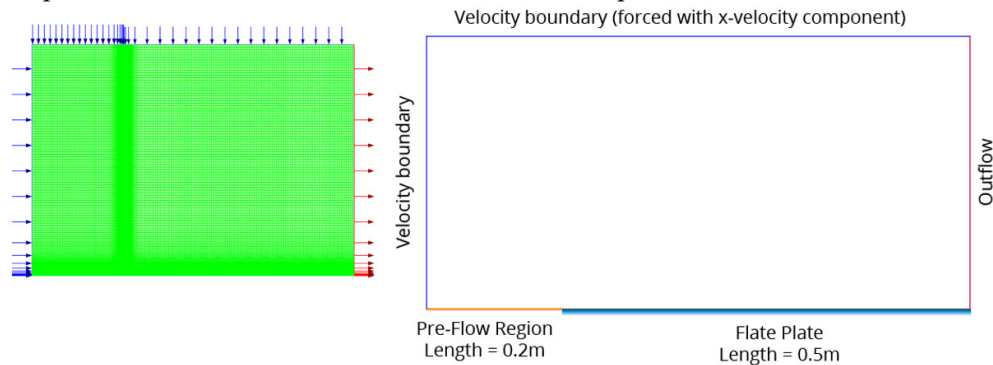
b.) Select 'Air' as your material. Note down the material properties from Ansys-Fluent.

c.) Set the velocity to the value of the last digit of one of your student numbers. E.g. s311365**2** means the inlet velocity will be 2 m/s (you cannot use a velocity value of 0 m/s). Set the velocity in 'Named Expressions'. The inlet and wall temperatures are already set at 293 K and 373 K, respectively. When the values are set, **update the Reference Values** which Ansys-Fluent uses to perform calculations. This can be done via the graphic interface as follows:
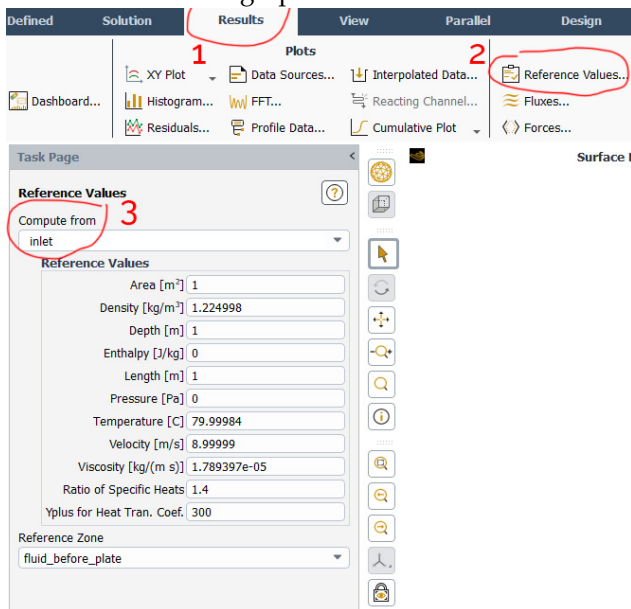
or you can use the text interface by typing in the following command in Fluent:
/report/reference-values/compute velocity-inlet inlet

d.) Using the **XY Plot** function, extract velocity, and temperature data from the lines x = 0.1, 0.3, and 0.5 (these are labelled as line_x0.1m, line_x0.3m, line_x0.5m). This is similar to placing a velocity, and temperature probe at three locations. Save these to a spreadsheet or other tools (Matlab, Python, Mathematica ... etc) for your analysis.

e.) Using the **XY Plot** function, extract the heat transfer coefficient, and skin friction coefficient from the wall plate (labelled as wall_plate), and save the data to a spreadsheet or other tools.

f.) Extract any images you need for flow and heat transfer visualisation for your video presentation. This can include vectors, and contours. Save the data file so you can access it again if you need to extract more images.

g.) Select 'miet1081-a' as your material. Note down the material properties from Ansys-Fluent. Repeat steps (d), (e), and (f) where you extract the velocity and temperature data points.

h.) Select 'miet1081-b' as your material. Note down the material properties from Ansys-Fluent. Repeat steps (d), (e), and (f) where you extract the velocity and temperature data points.

i.) As a reminder, before extracting the data, **update the Reference Values** which Ansys-Fluent uses to perform calculations. This can be done via the graphic interface (shown earlier) or you can use the text interface by typing in the following command in Fluent:
/report/reference-values/compute velocity-inlet inlet

5.2   Turbulent Flow Over a Flat Plate

a.) Use the Fluent cas file 'flat_plate_turbulent.cas.h5'. The domain is shown below. The measurement region is only along the flat plate. The pre-flow region is used to help setup and stabilise the flow field. The flat plate region begins at x = 0 m and ends at x = 5.0 m.

Velocity boundary (forced with x-velocity component)

Velocity boundary

Outflow

Pre-Flow Region
Length = 0.2m

Flate Plate
Length = 5.0m

b.) Select 'Air' as your material. Note down the material properties from Ansys-Fluent.

c.) Set the velocity to 3X m/s where X is the value of the last digit of one of your student numbers. E.g. s311365**9** means the inlet velocity will be 39 m/s. Set the velocity in 'Named Expressions'. The plate temperature is 20°C (293.15 K)and the inlet fluid temperature is 80°C (353.15 K).

d.) Extract any images you need for flow and heat transfer visualisation for your video presentation. This can include vectors and contours.

e.) Using the **XY Plot** function, extract the velocity, temperature profiles along the x positions, and the heat transfer coefficient along the wall plate (labelled as wall_plate), and save the data to a spreadsheet or other tools.

As a reminder, before extracting the data, **update the Reference Values** which Ansys-Fluent uses to perform calculations. This can be done via the graphic interface (shown earlier) or you can use the text interface by typing in the following command in Fluent:
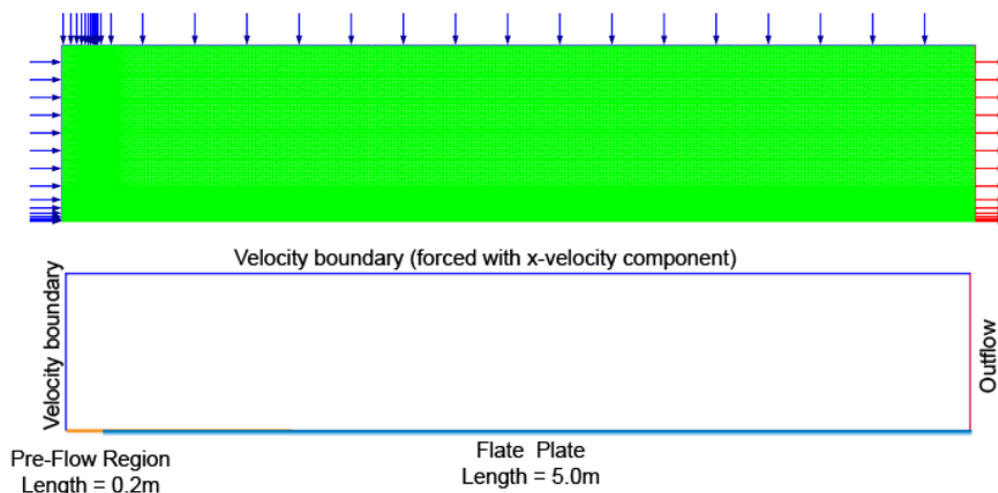**/report/reference-values/compute velocity-inlet inlet**

6 Flow over a Cylinder

6.1 Introduction

The flow over a circular cylinder is one of the most widely investigated problems because of the many fluid dynamics properties that it exhibits. As the Reynolds number (Re) increases, the flow exhibits a series of different fluid structures. This includes separation, and vortex shedding in its wake, which all contribute to drag, lift and heat transfer.

Figure 6.1 shows the varied and rich fluid dynamics that occur at different Re numbers. The flow field over the cylinder is symmetric at low values of Reynolds number. As the Reynolds number increases, flow begins to separate behind the cylinder and vortex shedding occurs. Vortex shedding is an unsteady phenomenon. You will run simulations in steady state or unsteady (time dependent) solver to capture these effects, as appropriate.

In this tutorial we investigate the influence of Re on the flow characteristics and how this influences the heat transfer. This forms part of the external forced convection heat transfer topic.
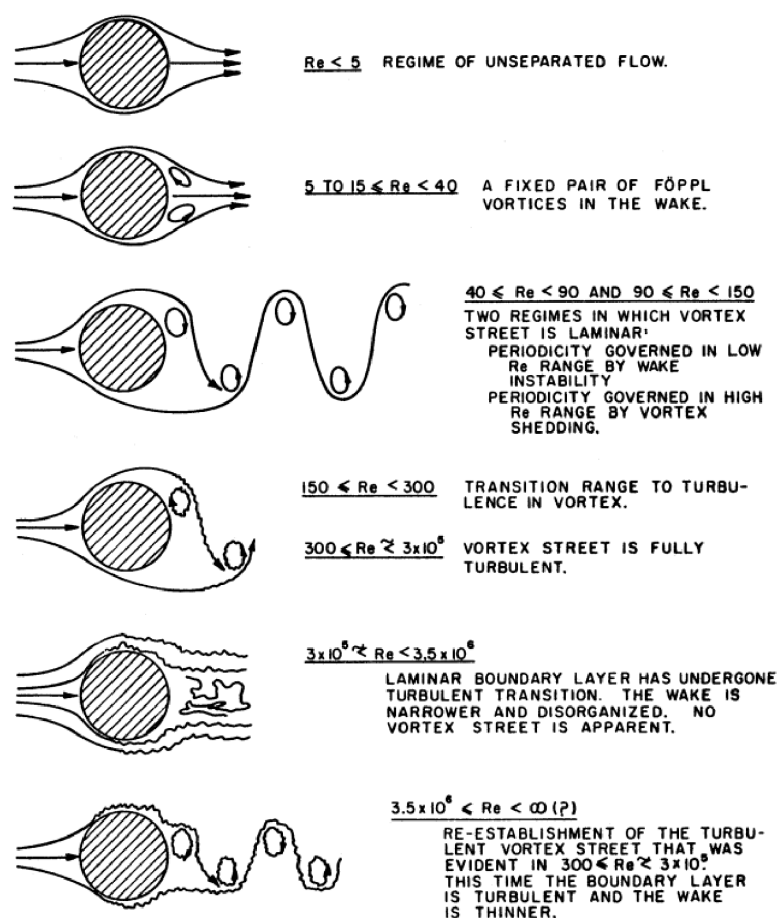
Figure 6.1: Regimes of fluid flow across a smooth circular cylinder - image from Lienhard (1966) Synopsis of lift, drag, and vortex frequency data for rigid circular cylinders, Technical Extension Service

The lab report requirement consists of two parts: i) experimental measurements and its analysis from the wind tunnel experiments, and ii) CFD modelling.

The aims of this report are:

- to visualise and understand the fluid dynamics associated with flow over a cylinder through the application of CFD modelling using FLUENT.
- to perform experimental data analysis to obtain drag coefficient, and compare this value with CFD simulations
- to write concise, clear, and well documented reports.

6.2 Model Setup

An infinitely long circular cylinder of diameter D = 0.01m is placed in a uniform crossflow with velcity U, modelled in 2D as shown in Figure 6.2, "flowCylinder-2022.cas.h5". There are two boundary conditions to define. Firstly the flow velocity which is defined at the inlet. Also when considering heat transfer effects, we have to include temperature values for the inlet velocity, and then the cylinder wall. The heat transfer then becomes an external forced convection flow over a cylinder.

In the labs you need to simulate the following cases:

- Re = 1 (steady, laminar)
- Re = 20 (steady, laminar)
- Re = 100 (unsteady, laminar)
- Re = wind tunnel lab experiment, (steady laminar and turbulent)
- Re = $1e^6$ (steady turbulent)
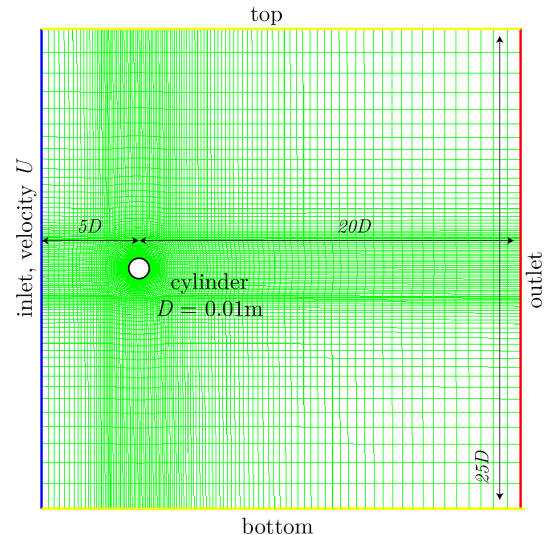
Figure 6.2: Fluent geometry setup

6.3 Defining Re number

The Re number is a dimensionless number which we use to match flow properties. This means that even if the flow conditions exhibit different material properties, velocities and dimensions - as long as the Re numbers are the same - then the fluid flow characteristics should remain the same. The Re number is defined as

$$Re = \frac{\rho D U}{\mu}$$

where D = 0.01m.

The fluid material we are using is a customised one, called *miet1081* where we will change its properties in order to easily calculate the Re number.

For example, for Re = 1, we define the material property as
$\rho = 1$, D = 0.01, $\mu = 0.01$
which produces

$$Re = \frac{\rho D U}{\mu} \Rightarrow 1 = \frac{1(0.01)}{0.01}U$$

This means that U = 1 to obtain Re = 1 and hence gives us a direct and convenient method for defining Re based on changes in the velocity.

Based on the Reynolds number the flow may be laminar or turbulent, and we assign this to the setup file with:

`/define/models/viscous laminar y` or `/define/models/viscous ke-standard y`

6.4 Defining material properties

In the previous classes we changed the material type for a given body or region/zone. This time we maintain the material type (being *miet1081*), but instead we explicitly define its density, and dynamic viscosity.

`/define/material/change-create/miet1081 miet1081 y constant dens n n y constant visc n n n`

where dens = density [kg/m^3]; visc = dynamic viscosity [kg/m.s]

6.5 Defining boundary conditions

The flow setup is determined by boundary conditions which are applied to the inlet boundary and the cylinder wall shown in Figure 6.2

At the inlet a velocity needs to be defined based on the Re number required, and its temperature needs to be defined.

```
/define/boundary-conditions/velocity-inlet inlet n n y y n vel n 0 0 n temp
```

where you interchange the red text with a numerical value, e.g. vel = 1 m/s; temp = $20\,^0$C

At the cylinder wall, we will use a temperature boundary condition to replicate many cases that resemble a heat exchanger. This is similar to what we used in Week-1.

Temperature [^{0}C] example:

```
/define/boundary-conditions/wall cylinder 0 n 0 n y temperature n 20 n n n 1
```

6.6 Initialize and Iterate

We apply the following lines. Note that there is a new line here which is to normalise our results in order to obtain the drag coeficient. The other new line will display the Re number for the setup.

```
/solve/initialize/compute-defaults velocity-inlet inlet
/solve/initialize/init yes
/report/reference-values/compute velocity-inlet inlet
/report/reference-values velocity 1 /report/reference-values density 1
/report/surface-integrals/a-w-avg inlet , check-re-number n
```

6.7 Putting it all together

We will use the journal file to setup the case only, and instead will run the simulation manually using the graphical interface. An example for running Re = 1 is given:

```
/define/models/viscous laminar y
/define/material/change-create/miet1081 miet1081 y constant 1 n n y constant 0.01 n n n
/define/boundary-conditions/velocity-inlet inlet n n y y n 1 n 0 0 n 20
/define/boundary-conditions/wall cylinder 0 n 0 n y temperature n 20 n n n 1
/solve/initialize/compute-defaults velocity-inlet inlet
/solve/initialize/init yes
/report/reference-values velocity 1
/report/reference-values density 1
/report/reference-values/compute velocity-inlet inlet
/report/surface-integrals/a-w-avg inlet , check-re-number n
/solve iter 750
```

6.8 Running the simulation

To run the simulation using the graphical interface.

i) Select the Run Calculation option on the left panel

ii) Set the number of iterations

iii) Click calculate. (see Figure 6.3)

iv) After the solution has converged save the data. Click File → Write → Data. This will save the iterations that has been performed.

The number of iterations you choose will depend on the convergence of the drag coefficient. Monitor the drag coefficient in Fluent's console until it converges. If it hasn't repeat the above iterations step and continue running the iterations. Note that
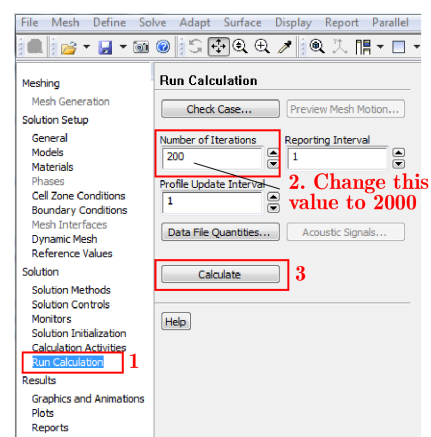

Figure 6.3: Using the graphical interface for manually starting the simulation.

if you intialize the case by using your setup journal file, then
the solution iterations and the results are reset.

```
                                         drag coefficient

iter   continuity  x-velocity  y-velocity     energy        Cd-1      time/iter
 800   9.8836e-05  2.5498e-05  9.3206e-06  8.0878e-08   1.5292e+01  0:00:32  200
 801   9.9876e-05  2.5452e-05  9.2961e-06  7.8189e-08   1.5290e+01  0:00:26  199
 802   9.9112e-05  2.5405e-05  9.2717e-06  7.6086e-08   1.5289e+01  0:00:21  198
 803   9.9662e-05  2.5357e-05  9.2483e-06  7.8832e-08   1.5287e+01  0:00:16  197
 804   9.8267e-05  2.5312e-05  9.2232e-06  7.7611e-08   1.5285e+01  0:00:13  196
 805   9.8581e-05  2.5265e-05  9.1987e-06  7.7093e-08   1.5283e+01  0:00:10  195
 806   9.8626e-05  2.5218e-05  9.1747e-06  7.6839e-08   1.5282e+01  0:00:08  194
 807   9.8350e-05  2.5172e-05  9.1508e-06  7.7842e-08   1.5280e+01  0:00:07  193
 808   9.7694e-05  2.5127e-05  9.1273e-06  7.7655e-08   1.5278e+01  0:00:44  192
 809   9.6680e-05  2.5080e-05  9.1038e-06  8.0957e-08   1.5277e+01  0:00:35  191
 810   9.7365e-05  2.5034e-05  9.0798e-06  7.8243e-08   1.5275e+01  0:00:28  190
```

Figure 6.4: Monitor the drag coefficient in Fluent console.

6.9   Post Processing

The steps to produce contours, vectors, streamlines, and xy-
plots are all similar. Specific instructions are given below:

  i) **Plot Vectors**: Click (1) *Graphics and Animations* in the left panel; 2) *Select Vectors*; 3) Click *Setup*. A
     vectors options panel opens up and here you setup the details of the vector plot. See Figure below.
     In Step 7, change the Scale to reduce or enlarge the vectors. For this setup try Scale = 0.001.
     The value Skip removes every other vector. For this setup use a value of Skip = 1.
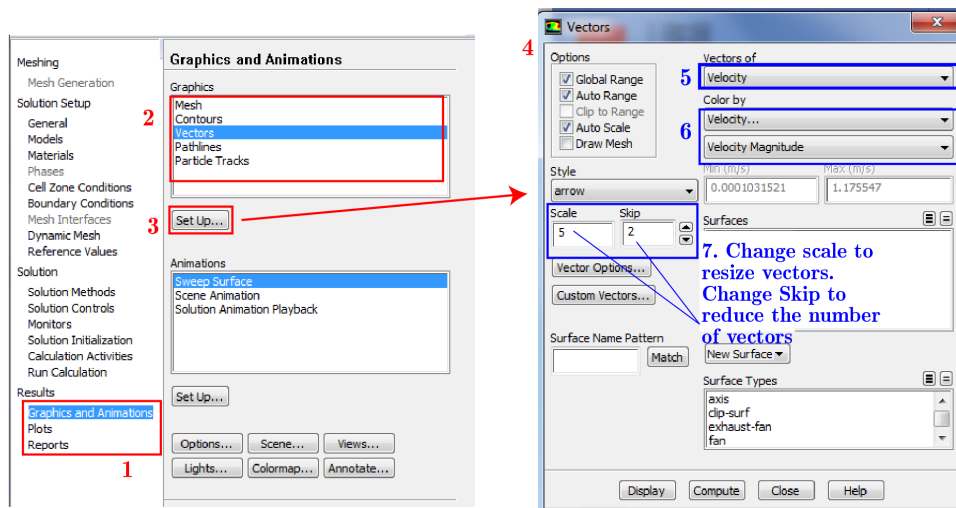
Figure 6.5: Post Processing Steps.

 ii) **Plot Contours**: Same procedure as above but in step 2, select Contours. In Step 5 and 6 Select Pressure
     or Velocity contours as needed.
iii) **xy-Plots**: 1) Make sure only the Node values is ticked. 2) Select the y-axis function you want to plot
     (e.g.Pressure Coefficient or Skin Friction Coefficient). Skin Friction can be found in Wall Fluxes → Skin
     Coefficient Coefficient 3) Select the cylinder surface.

6.10   Cases to Investigate

6.10.1   Re = 1, steady, isothermal, laminar

  i) For isothermal scenario, set the temperature of the cylinder wall equal to the inlet temperature.
 ii) Plot vectors in the vicinity of the cylinder wall. What do they look like and explain why.
iii) Plot flow streamlines across the cylinder. In Fluent these are referred to as Pathlines. Click "*Graphics
     and Animations*" → "*Pathlines*" → "*Setup*". The Pathlines options panel opens up to setup shown below
     in Figure 6.7. Select the line colours by velocity (1), then where the lines are released from (2) and
     finally Display (3).
     What do you notice about how the streamlines pass over the cylinder?

Figure 6.6: Making xy-plots

Figure 6.7: Pathlines setup options panel.
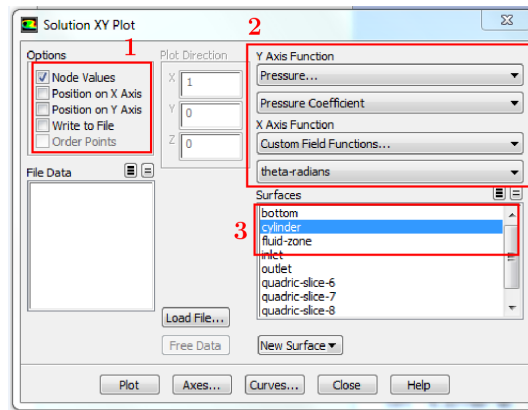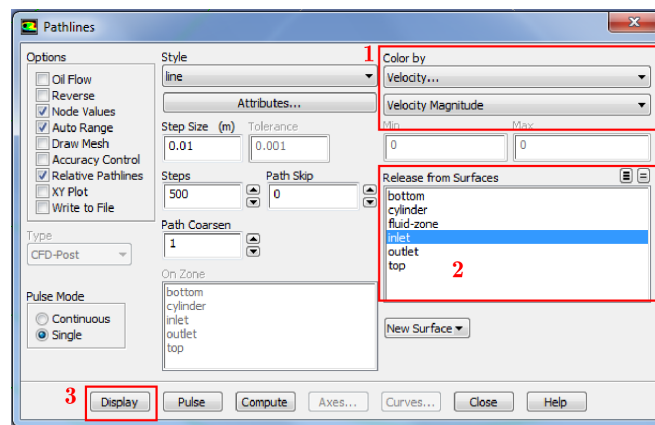
iv) Plot a pressure contour plot. What is the pressure at the front and rear of the cylinder? Why?

v) Record the drag coefficient.

vi) Plot an xyPlot of the pressure coefficient, and the skin friction coefficient over the cylinder. The pressure coefficient can be found under the Pressure variable. The skin friction coefficient can be found under the Wall Fluxes variable.

6.10.2 Re = 20, steady, isothermal, laminar

i) Plot vectors in the vicinity of the cylinder wall. What do they look like and explain why.

ii) Plot a pressure contour plot. What is the pressure at the front and rear of the cylinder? Why?

iii) Plot a temperature contour plot. What is the pressure at the front and rear of the cylinder? Why?

iv) Plot an xyPlot of the pressure coefficient, and the friction coefficient over the cylinder.

6.10.3 Re = 100, unsteady, isothermal, laminar

i) Firstly run a steady flow to establish a steady state solution. Record its Cd, and generate a velocity contour plot.

ii) Plot vectors in the vicinity of the cylinder wall. Where do they separate?

iii) Plot an xyPlot of the pressure coefficient, and the friction coefficient over the cylinder.

iv) *YOU MUST RUN STEADY CASE BEFORE RUNNING UNSTEADY CASE*

v) We have solved the steady flow for Re=100, and now will continue the simulation but in an unsteady solution. This means we will look at the flow behaviour at incremental times defined by a time step Δt. If we choose a Δt that is too large we may not see the fluctuating vortices, and if we choose one that is too small, then this will take much longer than needed. We have to determine the fluctuating vortex frequency by using the dimensionless Strouhal (Sr) number. Experiments have found that the

Strouhal number for flow past a cylinder is roughly 0.165 for a Re=100. This means that

$$Sr = 0.165 = \frac{fD}{U}$$

D = 0.01, and let U = 1m/s (If you use a different velocity, then your Sr will change)

$$0.165 = \frac{f(0.01)}{1} \rightarrow f = 16.5$$

A shedding cycle time is then $\frac{1}{16.5}$, and the suitable time step to capture the frequency is then

$$\Delta t = \frac{1}{16.5(25)} \approx 0.0025s$$

In order to sufficiently capture the vortex shedding you should have about 25 time steps in one shedding cycle, and we should look at 5 cycles. Create an unsteady flow journal file with the following lines:

```
define/models/viscous laminar y
define/models/unsteady-2nd-order y
display/contour dynamic-press , ,
solve/animate/define define-mon , 1 n y 5 contour , ,
```

Run this command and setup the unsteady simulation by clicking on "Run Calculation". Enter values in boxes 2, 3, and 4 as needed. In this case we need *Time Step Size* = 0.0025s; *Number of Time Steps* = 125 (giving us 5 cycles to visualise); and *Max Iterations/Time Step* = 20 (this is the optimum value). Then click "Calculate" when ready. This simulation will take longer than any other simulation performed before, so please be patient.
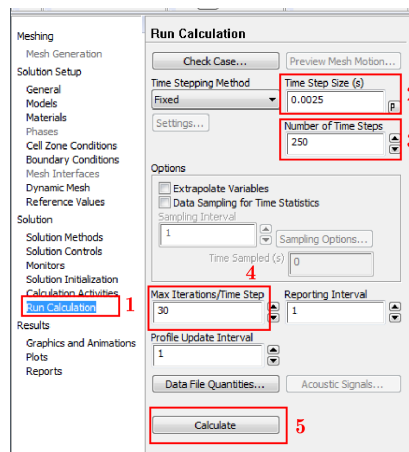
Figure 6.8: Setup window for unsteady iterations.

vi) From the unsteady simulation, plot velocity or pressure contours at four different time steps to demonstrate the vortex shedding phenomena.

vii) *Post-processing unsteady results* Follow the steps shown in the Figure 6.9. If there is no file shown in Sequences, but you have made the files earlier, then you can read in the file by clicking Read, and the .cxa file within the same folder as the Fluent case file. Use the play and stop buttons to view the results.

6.10.4 Re = 1,000,000, steady, isothermal, turbulent

For a turbulent case, we have to specify the turbulent conditions at the boundaries. The material property, inlet and cylinder boundaries are re-defined as:

note we abbreviate some commands (`boundary-conditions = b-c`; `change-create = c-c`)

```
define/material/c-c/miet1081 miet1081 y constant 1000 n n y constant 0.0001 n n n
define/b-c/velocity-inlet inlet n n y y n 10 n 0 0 n 20 n n n y 5 0.25
define/b-c/wall cylinder 0 n 0 n y temperature n 20 n n n 0 n , n 1
```

Figure 6.9: Solution Playback window for unsteady simulation

i) Plot vectors in the vicinity of the cylinder wall. What do they look like and explain why.
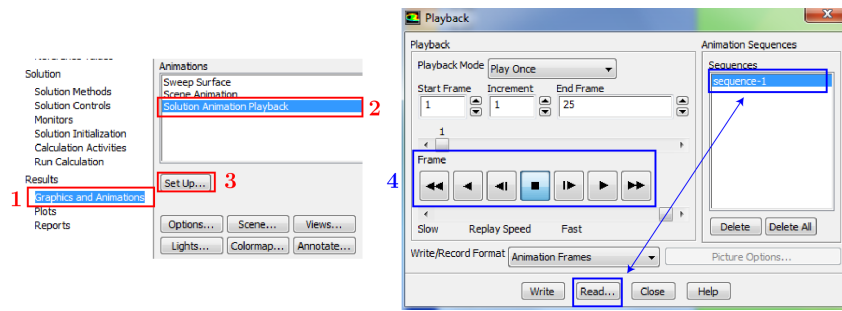ii) Plot a pressure contour plot. What is the pressure at the front and rear of the cylinder? Why?
iii) Plot a temperature contour plot. What is the pressure at the front and rear of the cylinder? Why?
iv) Plot an xyPlot of the pressure coefficient, and the friction coefficient over the cylinder.

6.10.5 Re = 20, and 1,000,000 heat transfer comparison

When we introduce heat transfer, we now must consider another dimensionless number, the Prandtl number (Pr), which we will cover in the topic of Forced Convection (Week 4-Week 9). The Prandtl number is a ratio of the momentum diffusivity (kinematic viscosity) to thermal diffusivity and it influences the thermal boundary layer. This means that the Prandtl number must remain constant between the two heat transfer cases we are investigating. The Pr is defined as

$$Pr = \frac{C_p \mu}{k}$$

where C_p is specific heat, (SI units : J/(kg K)), μ is dynamic viscosity, and k is thermal conductivity. Since μ is also used in the Re number we need to ensure that this value is constant for both cases to ensure that the material properties do not influence the thermal boundary layer.

	μ	D	ρ	U	Re
Re20	0.00001	0.01	1	0.02	20
Re1e6	0.00001	0.01	100	10	1,000,000

We also set the cylinder wall temperature to 100^OC. The command lines for material properties, cylinder wall, and velocity inlet becomes

Re=20

```
define/material/c-c/miet1081 miet1081 y constant 1 n n y constant 0.00001 n n n
define/b-c/velocity-inlet inlet n n y y n 0.02 n 0 0 n 20
define/b-c/wall cylinder 0 n 0 n y temperature n 100 n n n 1
```

Re=1,000,000

```
define/material/c-c/miet1081 miet1081 y constant 100 n n y constant 0.00001 n n n
define/b-c/velocity-inlet inlet n n y y n 10 n 0 0 n 20 n n n y 5 0.25
define/b-c/wall cylinder 0 n 0 n y temperature n 100 n n n 0 n , n 1
```

i) Plot temperature contour plot.
ii) Plot velocity contour plot.
iii) Discuss the boundary layers and how this relates to heat transfer.

7 Developing internal pipe flow

7.1 Introduction

Internal forced convection heat transfer occurs in various applications that involve pipe and duct flows, such as blood flow, oil and gas pipelines, and airconditioning and heating units. Convection heat transfer is complicated by the presence of fluid motion, and the presence of a boundary layer.

In this exercise, a hot fluid flows through a straight circular pipe (with cold walls) of constant radius of $r = 0.2$m and a length of L=0.8m. The inlet velocity begins at the inlet with a constant profile and as it progresses downstream begins to form a parabolic shape due to the momentum transfer between fluid particles caused by viscosity, and the no-slip boundary condition at the wall. Similarly a constant temperature profile is introduced at the inlet and as the fluid progresses downstream, a thermal boundary develops. The computational model considers half the circular pipe in 2D shown in Figure 8.1a. The effects of Prandtl number, and Reynolds number on the developing flow profile is investigated.
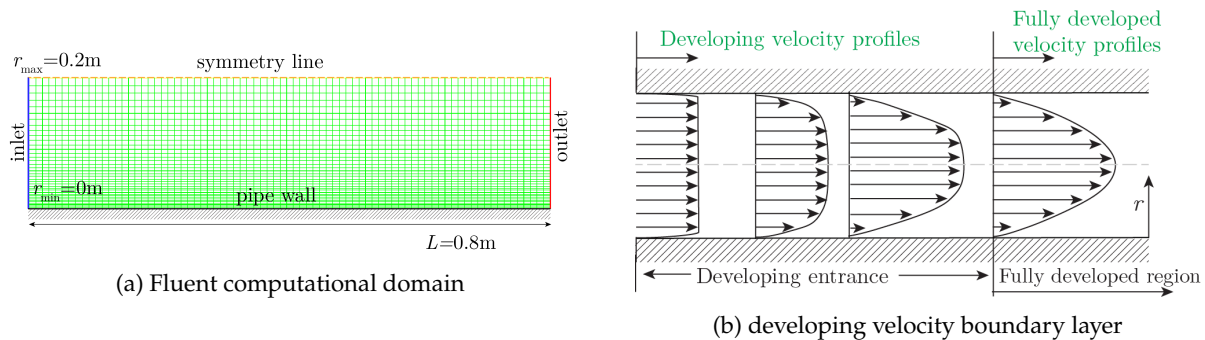

(a) Fluent computational domain

(b) developing velocity boundary layer

Figure 7.1: Developing internal pipe flow

7.2 Preliminary calculations

We will investigate the influence of three different materials, air, water, and oil, and observe how its fluid properties affects its flow behaviour.

We make use of Fluent's material library database and select the predefined material, which obviates the need to explicitly specify each material property. The values extracted from Fluent are given in the table below:

	μ [kg/ms]	C_p[J/kgK]	ρ [kg/m^3]	k[W/mK]
Air	1.8×10^{-5}	1006	1.225	0.02
Water	0.001	4182	1000	0.6
Oil	0.486	1910	884	0.144

The Prandtl number is defined as

$$\text{Pr} = \frac{C_p \mu}{k}$$

and the Reynolds number is

$$\text{Re} = \frac{\rho U D}{\mu}$$

Calculate the resulting Pr and based on the geometry having a diameter of $D = 2r_{max} = 0.4$m, calculate the required inlet velocity to achieve a Reynolds number of **Re=100** for the three fluids.

	Pr	U, inlet velocity [m/s]
Air		
Water		
Oil		

7.3 Model setup

As an example let's calculate air. The Pr becomes:

$$\text{Pr} = \frac{1006 \times 1.8 \times 10^{-5}}{0.02} = 0.905$$

and the inlet velocity needs to be

$$\text{U} = \frac{100 \times 1.8 \times 10^{-5}}{1.225 \times 0.4} = 0.0037\text{m/s}$$

```
/define/boundary-conditions/fluid pipefluid yes air no no no no 0 no 0 no no no
/define/boundary-conditions/wall bottomwall 0 no 0 no yes temperature no 5 no no no 1
/define/boundary-conditions/velocity-inlet inlet no no yes yes no 0.00365 no 0 no 25
/solve/initialize/compute-defaults velocity-inlet inlet
/solve/initialize/init yes
/report/reference-values/compute velocity-inlet inlet
/report/surface-integrals/a-w-avg inlet , , , , check-re-number n
```

For other cases, change the text or numbers highlighted in red to what it needs to be for the case you are investigating. The options are described below

- **air**: this is the fluid material selection, and can be air, water-liquid, or oil.
- **0.00365**: this is the inlet velocity in m/s
  We will keep the wall and inlet fluid temperature constant:
- **5**: this is the temperature of the cylinder wall surface
- **25**: this is the temperature of the inlet fluid

7.4 Questions

Q1 Plot the velocity and temperature contours for the three fluids below:

Velocity contour - AIR   Temperature contour - AIR

Velocity contour - WATER   Temperature contour - WATER

Velocity contour - OIL   Temperature contour - OIL

Q2 Before looking at the *xy*-plots in Fluent, draw the velocity and temperature profiles at the inlet position (*x*=0m, select 'inlet'), midpoint(*x*=0.4m), and outlet (*x*=0.8m) locations on the graphs below.

Velocity Profiles

U

0.4   x

Temperature Profiles

T

0.4   x
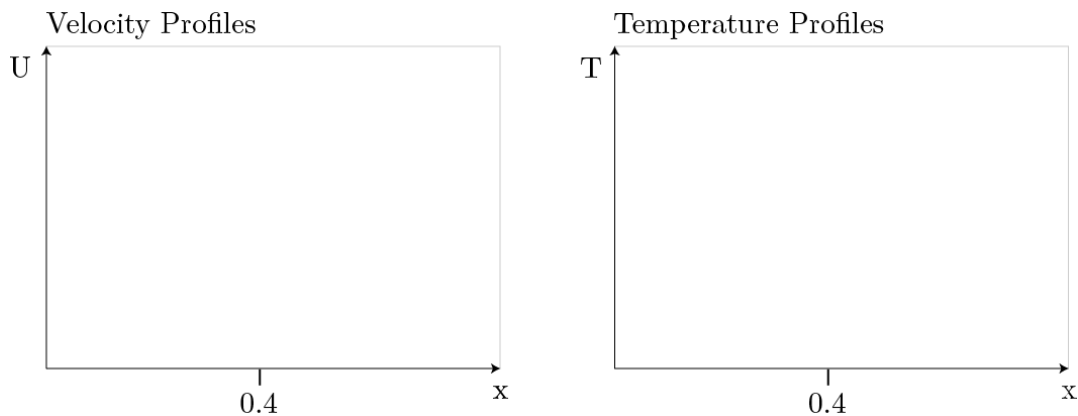
Q3 Describe the relative thicknesses of the velocity boundary layers for the three fluids.

_____

_____

Q4 Describe the relative thicknesses of the thermal boundary layers for the three fluids.

_____

_____

Q5 Explain the why there are differences or similarities between the relative boundary layer thicknesses between the three fluids.

_____

_____

Q6 What would the thermal boundary layer thickness look like for oil, if the Reynolds number is now Re = 1. Explain why.

_____

_____

8 Developed internal pipe flow

8.1 Introduction

In this exercise, a hot fluid flows through a straight circular pipe (with cold walls) of constant radius of $r = 0.1$m and a length of L=10m. The inlet velocity begins at the inlet with a constant profile and becomes fully developed at some section downstream. Two real fluids, air and water, pass through the pipes. The inlet temperature is constant at 25°C while the pipe surface has a constant surface wall temperature of 5°C. The fluid travels at a Reynolds number, Re number of 100. The computational model is similar to that from last week's class.

8.2 Preliminary calculations

Since we are using a real fluid, we make use of Fluent's material library database and select the predefined material. The values extracted from Fluent are given in the table below:

	μ [kg/ms]	C_p[J/kgK]	ρ [kg/m^3]	k[W/mK]
Air	$1.79{\times}10^{-5}$	1006	1.225	0.0242
Water	0.001	4182	998	0.6

The Prandtl number,

$$Pr = \frac{C_p\mu}{k}$$

and Re number are

$$Re = \frac{\rho UD}{\mu}$$

Calculate the resulting Pr and based on the geometry having a diameter of $D = 2r_{max} = 0.2$m, calculate the required inlet velocity to achieve a Reynolds number of **Re=100** for the two fluids. This is the same as last week's work which we calculated the Pr. However for the Re, the velocities need to change. Note that this week's pipe diameter (D=0.2m) is half of that from last week (D=0.4m), so by calculation our velocity needs to be doubled to maintain a Re = 100 (see last week's exercises if you are not sure)

	Pr	U, inlet velocity [m/s]
Air	0.744	0.0073
Water	6.97	0.0005

8.3 Model setup

We use the same journal file from last week. This time our pipe geometry is much longer and we want to investigate the fully developed length and fully developed region.

```
/define/boundary-conditions/fluid pipefluid yes air no no no no 0 no 0 no no no
/define/boundary-conditions/wall bottomwall 0 no 0 no yes temperature no 5 no no no 1
/define/boundary-conditions/velocity-inlet inlet no no yes yes no 0.0073 no 0 no 25
/solve/initialize/compute-defaults velocity-inlet inlet
/solve/initialize/init yes
/report/reference-values/compute velocity-inlet inlet
/report/surface-integrals/a-w-avg inlet , , , , , , , check-re-number n
```

For other cases, change the text or numbers highlighted in red to what it needs to be for the case you are investigating. The options are described below

- **air**: this is the fluid material selection, and can be air, or water-liquid.
- **0.0073**: this is the inlet velocity in m/s
  We will keep the wall and inlet fluid temperature constant:

- **5**: this is the temperature of the cylinder wall surface
- **25**: this is the temperature of the inlet fluid

8.4 Extra simulations

We can also apply a constant surface heat flux condition instead of the constant surface temperature. This is done by changing the wall boundary condition. We replace the script line:

```
/define/boundary-conditions/wall bottomwall 0 no 0 no yes temperature no 5 no no no 1
```

with

```
/define/boundary-conditions/wall bottomwall 0 no 0 no yes heat-flux no 20 no no no 1
```

The new line changes the boundary condition from constant wall surface temperature to constant heat flux with a value of $20 W/m^2$.

8.5 x-y plotting, symmetry view

In this exercise, you need to export velocity and temperature profiles at different pipe locations downstream. Then normalise the variable against its maximum value. Using air as an example, specific instructions based on velocity plots are given below:
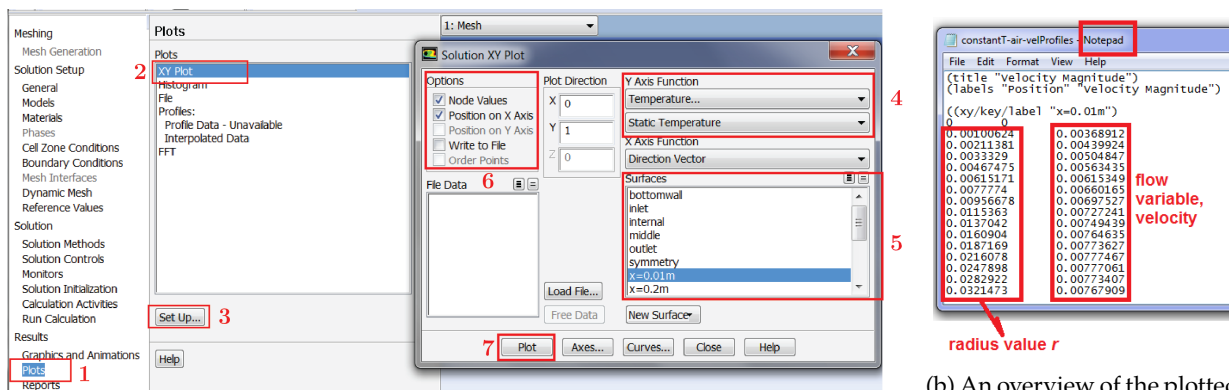

(a) Generate and export xy-plots

(b) An overview of the plotted data opened by notepad

For **Step 6** you can select to write the data out as a .txt file that can be opened in notepad or Matlab. Check the box for "Write to File". Note that if you want to plot on the screen then this box should be left unchecked. **Step 7** will now read "Write" instead of it plotting to the screen.

Save the file and then you will be able to open it in Notepad to view the data. Use a spreadsheet software to analyse. To view the entire pipe we can view the opposing symmetry side of the pipe. Click on "**Display**" → "**Views**", then select "**Symmetry**" → "**Apply**". Then when you view your contours it will show a full pipe.
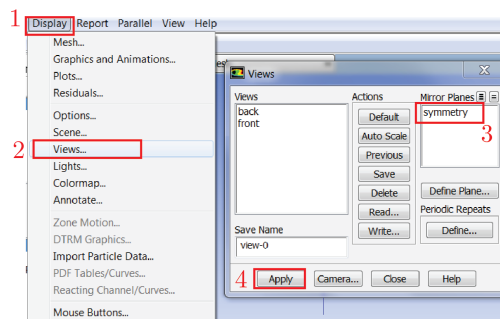
8.6 Questions

Investigate the velocity and temperature profiles at different x-distance locations.

- Q1 What do you notice about the fully developed temperature profiles?
- Q2 What would happen if the fluid temperature < surface temperature?
- Q3 What do you notice about the velocity profiles for water and air in the fully developed region?

9 Natural Convection

9.1 Introduction

***** *Before you start the simulations **Sketch your convection currents first.** Then we will run the simulations and compare with your sketches.*****

Many heat transfer applications involve natural convection as the primary mechanism of heat transfer. As its name suggests it is a nature at work, occurring naturally without any forced influences. In this lab we use CFD to visualise different natural convection currents. The Grashof number is a dimensionless number. As before, this means it is ratio of two flow properties. It represents the ratio of the buoyancy force to an opposing viscous force acting on the fluid.

$$\mathrm{Gr} = \frac{g\beta(T_s - T_\infty)L_c^3}{\nu^2} \qquad\qquad \mathrm{Ra} = \mathrm{Gr}\,\mathrm{Pr} = \frac{g\beta(T_s - T_\infty)L_c^3}{\nu^2}\mathrm{Pr}$$

Natural convection heat transfer correlations are usually expressed with the Pr involved. When we combine the Gr with the Pr we get the Rayleigh number.

9.2 Model setup

We want to investigate the influence of the Rayleigh number on the convection currents. We use air as the base material but alter its viscosity to get the desired Ra number. use an artificial viscosity. Real air has a viscosity of about 1.79e-5. (Note that we could change the temperatures which also affects the thermal expansion coefficient)

9.3 Cases to investigate

CASE 1: The physical model considered here is a two dimensional square cavity of length L with a heated thin plate of length h placed horizontally (see figure) or vertically at its center. The plate is isothermally maintained at a higher temperature T_s = 320K. The vertical walls are cooler at a constant temperature T_∞ = 300K while the horizontal walls are insulated.

The Rayleigh can be rewritten with the kinematic viscosity ν and the Pr number expanded out:

$$\mathrm{Ra} = \mathrm{Gr}\,\mathrm{Pr} = g\beta(T_s - T_\infty)L_c^3 \; \frac{\rho^2}{\mu^2} \; \frac{C_p\mu}{k}$$

Taking the characteristic length as L = 0.2m, density as ρ = 1.225, C_p = 1006, and conductivity as k = 0.0242 the required viscosity can be determined by setting the Ra number to 10e-7:

$$1 \times 10^7 = (9.81)\left(\frac{1}{310}\right)(320 - 300)(0.2^3)\left(\frac{1.225^2}{\mu^2}\right)\left(\frac{1006\mu}{0.0242}\right) \rightarrow \quad \mu \approx 3.6 \times 10^{-5}$$
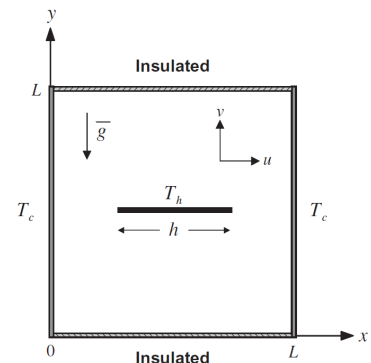
Set the viscosity of air with an artificial value of 3.6e-5. The default script command lines are:

```
/define/boundary-conditions/wall top-bottom 0 n 0 n y heat-flux n 0 n n n 1
/define/boundary-conditions/wall left-right 0 n 0 n y temperature n 300 n n n 1
/define/operating-conditions/gravity yes 0 -9.81
/define/material/c-c/air air n n n y constant 3.6e-5 n n n
/solve/initialize/init yes
/report/surface-integrals/a-w-avg cavityfluid , rayleigh n
```

Change the numbers highlighted in blue or red to the text or value needed to achieve the required Rayleigh number.

- **3.6e-5**: this is the dynamic viscosity, µ
- **0 -9.81**: the first value is the x-coordinate, and the second value is the y-coordinate for gravitational acceleration.

i) Run the case for a horizontal plate at Ra = 10^7. Plot its temperature contour and compare it with your sketched natural convection currents.

ii) Run the case again but for Ra = 10^5. Change the fluid material properties by increasing its viscosity to 3.6e-3 script. Change the line for the material property as

```
/define/material/c-c/air air n n n y constant 3.6e-3 n n n
```

Plot its vector plot on the *cavityfluid* surface (select *cavityfluid* surface), and also the temperature contour (don't select any surface). Compare what you see with your sketched natural convection currents.

CASE 2: This time we want to simulate a vertical plate inside the square cavity. Instead of re-building the entire geometry again, we can simply change the boundary conditions and take advantage of the symmetrical features of the geometry. Make the *top-bottom* wall a temperature boundary while the *left-right* wall becomes adiabatic with a heat-flux condition of 0 W/m^2. Also change the script line for *gravity* so that it reads -9.81 0 to change it into the x-coordinate. Run for the two Ra-number cases, Ra=10^5 and Ra=10^7.

Plot its vector plot on the *cavityfluid* surface (select *cavityfluid* surface), and also the temperature contour (don't select any surface). Compare what you see with your sketched natural convection currents.

CASE 3: A rectangular enclosure is investigated. The side walls are adiabatic. A hot bottom wall has T_{bottom} = 320K and a colder upper wall has T_{top}=300K. The following script is used to run case-3.

<div align="center">

top

left | *cavityfluid* | right

bottom

</div>
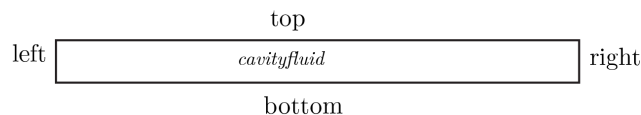
```
/define/boundary-conditions/wall top 0 n 0 n y temperature n 300 n n n 1
/define/boundary-conditions/wall bottom 0 n 0 n y temperature n 320 n n n 1
/define/boundary-conditions/wall left 0 n 0 n y heat-flux n 0 n n n 1
/define/boundary-conditions/wall right 0 n 0 n y heat-flux n 0 n n n 1
/define/oper-cond/gravity yes 0 -9.81
/define/material/c-c/air air n n n y constant 3.6e-6 n n n /solve/initialize/init yes
/report/surface-integrals/a-w-avg cavityfluid , rayleigh-rectangle n
```

Plot vector and temperature contour plots and compare with what currents you sketched.

CASE 4: As in Case-3, rotate the geometry by changing the boundary conditions and the gravity term so that the case now becomes a vertical enclosure. Plot vector and temperature contour plots and compare with what currents you sketched.
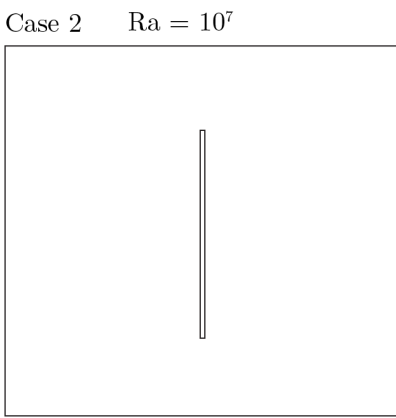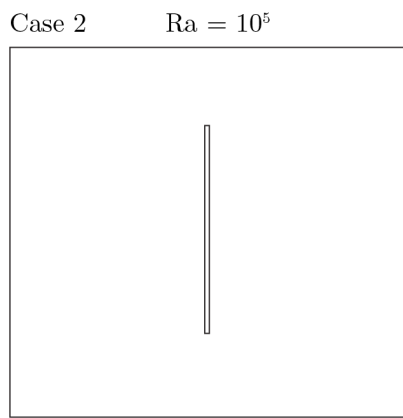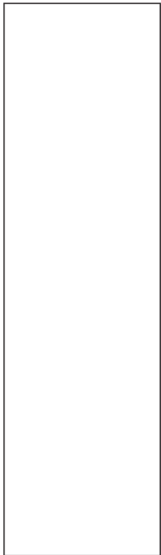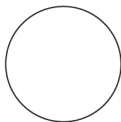
9.4 Sketches

Sketch your convection currents first! Then run the simulations and see how many you got right.

Case 1   Ra = 10^5

Case 1   Ra = 10^7
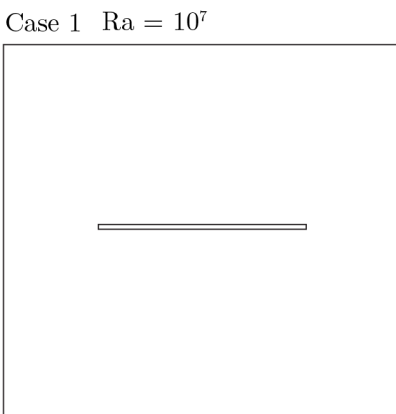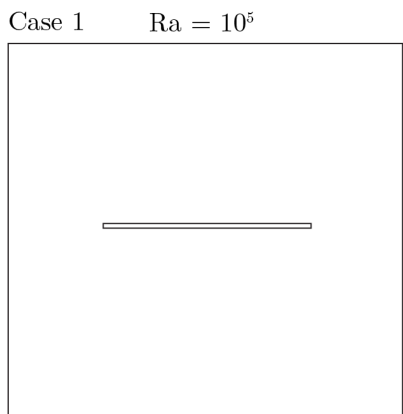
Case 4 (hot right wall)

Case 2   Ra = 10^5

Case 2   Ra = 10^7

Case 3 (hot bottom surface)

Case 5 (hot cylinder in cold air)