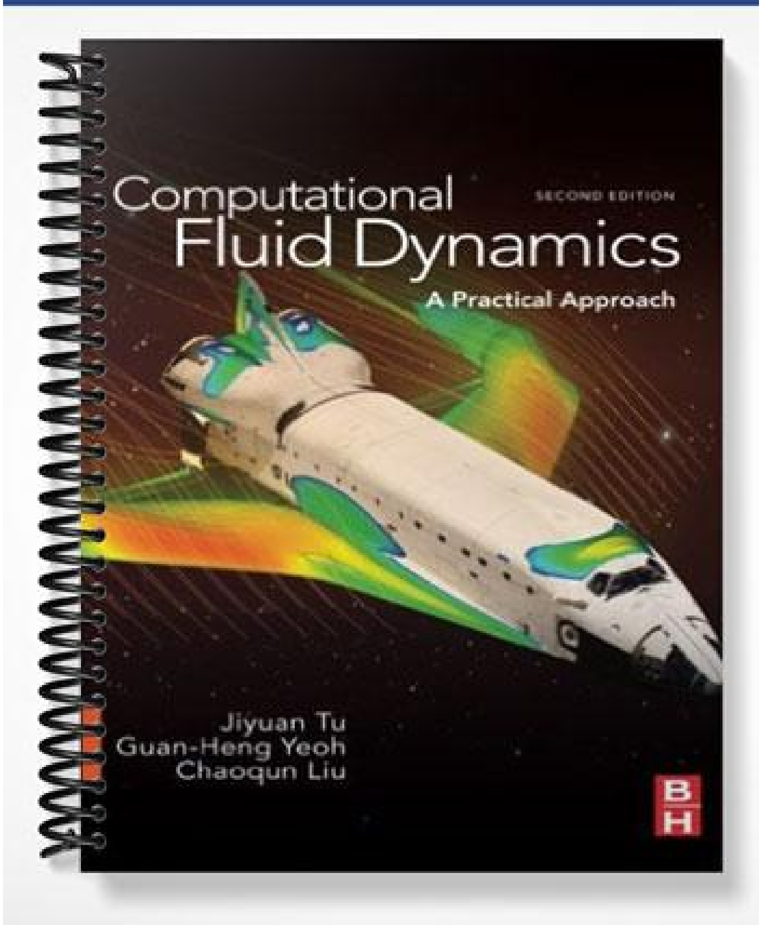
SOLUTIONS MANUAL



REVIEW QUESTIONS – CHAPTER 2

2.1 How are commercial codes allowing CFD analysis to be carried out with ease for the novice user?

CFD commercial codes employ more user-friendly environments and provide functionality for a variety of applications. Without any prior basic CFD knowledge, a first-time user can perform the necessary operations that involve setting up a fluid problem, solving the numerical problem and managing some graphical representation of the results attained.

2.2 What are the main elements involved in a complete CFD analysis?

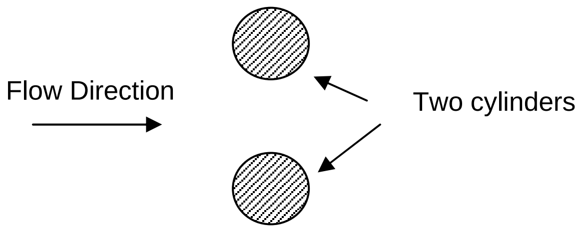
The main elements involved are:

- the pre-processing step that involves creating the computational domain and geometry from the physical situation;
- the solver which implements the numerical techniques and applies it across the computational domain;
- the post-process step, which involves presenting the data in graphical format for analysis.

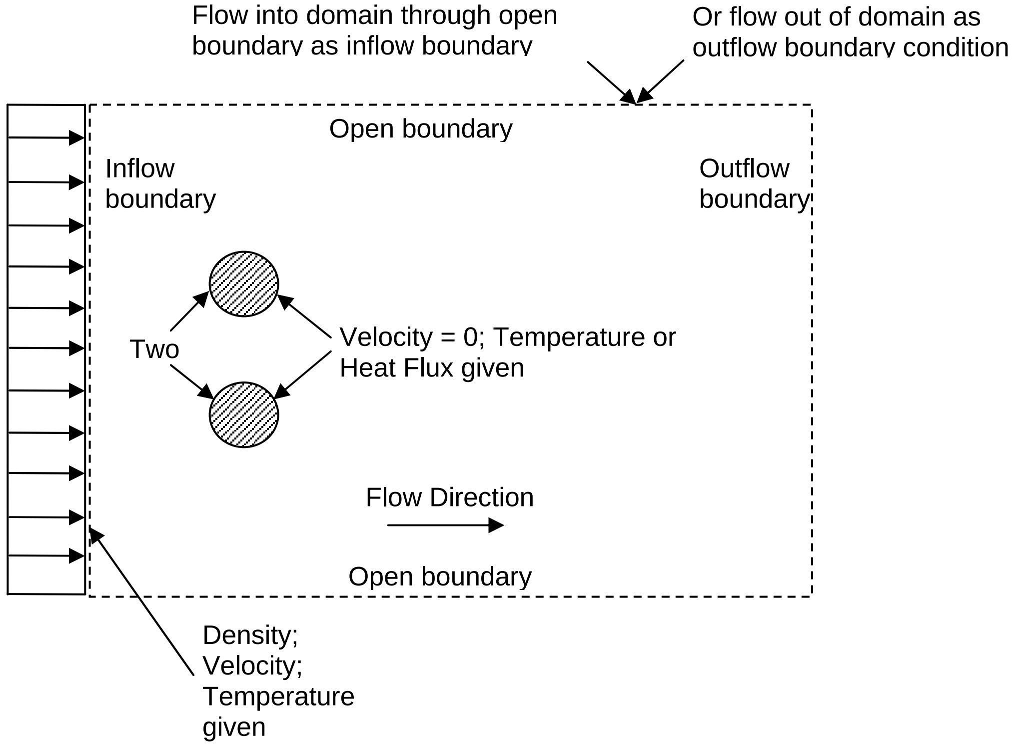
2.3 Why is it important to correctly define the computational domain for the fluid flow problem? Give an example of this.

The computational domain must represent the physical domain correctly so that the results will reflect the true nature of the real case. The following is an example of the many ways the physical domain of a flow passing two cylinders can be converted into a computational domain. Note that the computational domain requires boundary definitions to enclose the domain.

Physical domain



Computational Domain



2.4 What is the consequence of using a very fine mesh (i.e. a very large number of cells) compared with using a coarse mesh (i.e. a small number of cells)?

A very fine mesh uses a large number of cells, which means that more calculations are required to iterate the governing equations to resolve for all the cells. Therefore using a coarser mesh will lead to a shorter computational simulation time. However a coarse mesh will exhibit large cell sizes which lead to less accurate results.

2.5 What is the main difference between a structured and unstructured mesh and when are they applied to physical domains?

For relatively simple geometries such as a square box, structured cells that generally comprise a regular distribution of rectangular cells is a better option. For more complex geometries such as the human body, structure cells are difficult to apply and the use of unstructured cells is appropriate.

2.6 What types of boundary conditions can be imposed on the computational domain?

The type of boundary condition that needs to be imposed on the computational domain is dependent on the physical model. The following are some of the available conditions:

- Inflow - pressure, velocity, mass-flow
- Outflow - pressure, velocity, mass-flow
- Open - symmetry, axis, cyclic, wall

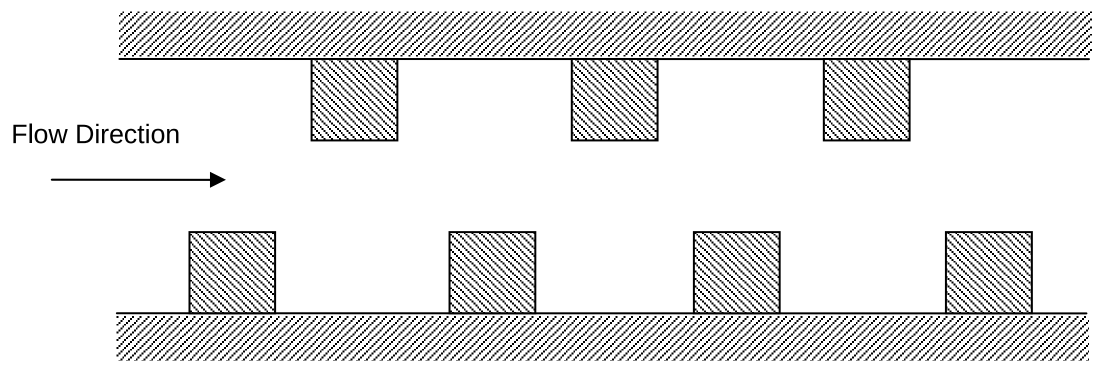
2.7 What type of boundary can be used for a computational boundary that represents an open physical boundary?

The most common boundary conditions for an open physical boundary is symmetry, axis and cyclic.

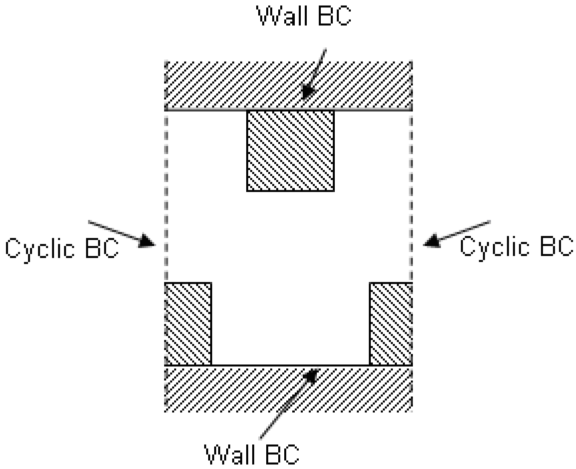
2.8 What advantages can a *symmetry* boundary condition and a *cyclic* boundary condition provide and when can they be applied?

Symmetry or cyclic boundary conditions can be applied on repeating geometries, where a single computational domain is used to represent the entire domain through symmetry or cycles.

Physical Domain



Computational Domain



The simplified computational domain is now smaller and a very fine mesh resolution can be applied. The computations are now applied to one representative domain rather than the entire physical domain that would take a much longer time.

2.9 What is the main purpose of a *CFD solver*?

The purpose of the CFD solver is to set up numerical algorithms and to obtain a solution for the governing equations. The solver consists of applying different models and numerical schemes to control the accuracy and stability of the simulation followed by computationally solving the problem.

2.10 What are the advantages of providing intelligent values for the initial solution?

Although initial conditions can be applied arbitrarily, an intelligent initial value that is close to the final solution will quicken the iterative procedure whereby the solution will converge in shorter computational time. If, however, the initial values are far away from reality, the computations will take longer for the solution to converge and may even lead to the iterative procedure misbehaving and possibly “blowing-up” or diverging. Normally an intelligent initial guess is based on the user’s experience.

2.11 What is an iteration process and how is it performed?

The iteration process involves the solving of the algebraic set of equations leading to a set of solved values for all the variables involved. Some well known iterative solvers are Algebraic Multi Grid (AMG) algorithm, conjugate gradient methods, Strongly Implicit Procedure (SIP) of Stone’s method and TDMA line-by-line procedure. The goal of the iteration is to reduce the magnitude of the differences in the solution within each iteration step. The iteration process stops when the differences, called residuals, reach a certain limit or criteria. More descriptions of iterative solvers are provided in Chapter 4.

2.12 What does the convergence criterion control?

A solution is considered to have converged when the differences in the solution variables, also called residuals or imbalances, is reduced through each iteration step until an acceptable level is reached. This acceptable level is called the convergence criteria. More details of residuals are discussed in Chapters 4 and 5.

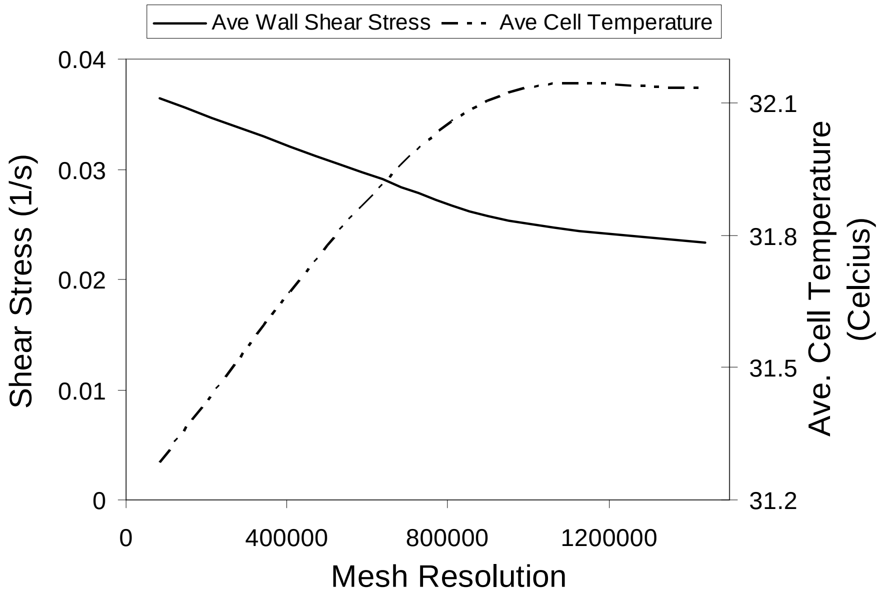
2.13 What is the main purpose of the *post-processing* stage?

The purpose of the post-processing stage is to translate, interpret and analyse the raw numerical data into graphical formats that are visually appealing and that convey the results in the correct manner.

2.14 What are the advantages of using X-Y plots? Give examples of what CFD results X-Y plots can capture.

X-Y plots are usually two-dimensional graphs that present the variation of one dependent variable against another independent variable. They are the most precise and quantitative way to present the numerical data since they are easily identifiable. The reader can interpret the results easily without resorting to any mental or arithmetic interpolation.

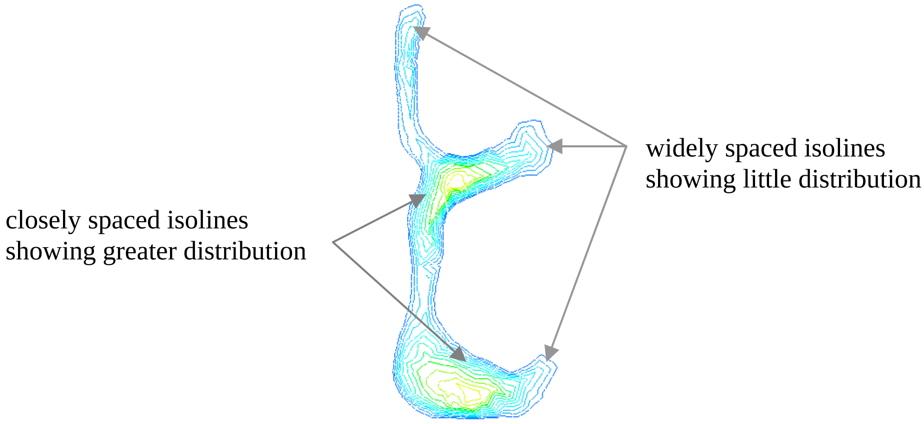
Examples of X-Y plots may involve line profiles of velocity, pressure, shear stress, temperature and skin-friction coefficient.



Example of two x-y plots on a single chart

2.15 Why are contour plots best used to display the distribution of a variable?

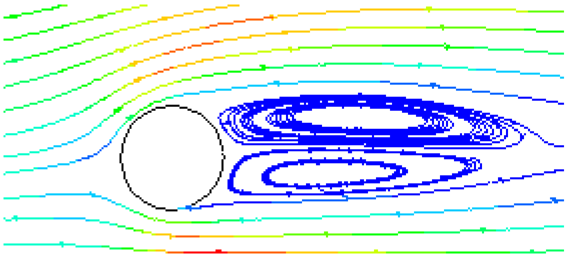
Contour plots provide a global description of the fluid flow encapsulated in one view. A contour line (also known as *isoline*) is a line that indicates a property which is constant along that line. Generally the numerical value of a variable from one contour line to an adjacent contour line is also held constant and it is the distance between each isoline that is useful in showing the distribution of the variable over the geometry. For example the below velocity magnitude contour plot of a nasal cavity cross-section shows a little distribution at the tips of the geometry where the lines are widely spaced apart, while in the middle regions where the lines are close to each other, there is a large greatest round the edges.



Velocity magnitude contour plot of a nasal cavity cross-section

2.16 What is the meaning of a streamline? What advantages do they have over other plot types?

A streamline is a path that is traced out if a massless particles moves with the flow. In other words any point along the streamline represents the parallel line to the mean velocity vector at that point. The streamline plot can reveal important features that could be obscured in some isolated flow regimes when using other types of plots. For example the below figure shows a streamline plot of a flow around a cylinder, which shows the wake-developing vortices behind the cylinder which may be obscured in other plots such as contours or x-y plots.



Streamline plot of a flow over a cylinder showing wake-developing vortices