EDISON_CFD

EDucation-research Integration through Simulation On the Net

첨단 사이언스·교육 허브 개발 (EDISON) 사업

Implementing Boundary Conditions

이신형
서울대학교 공과대학

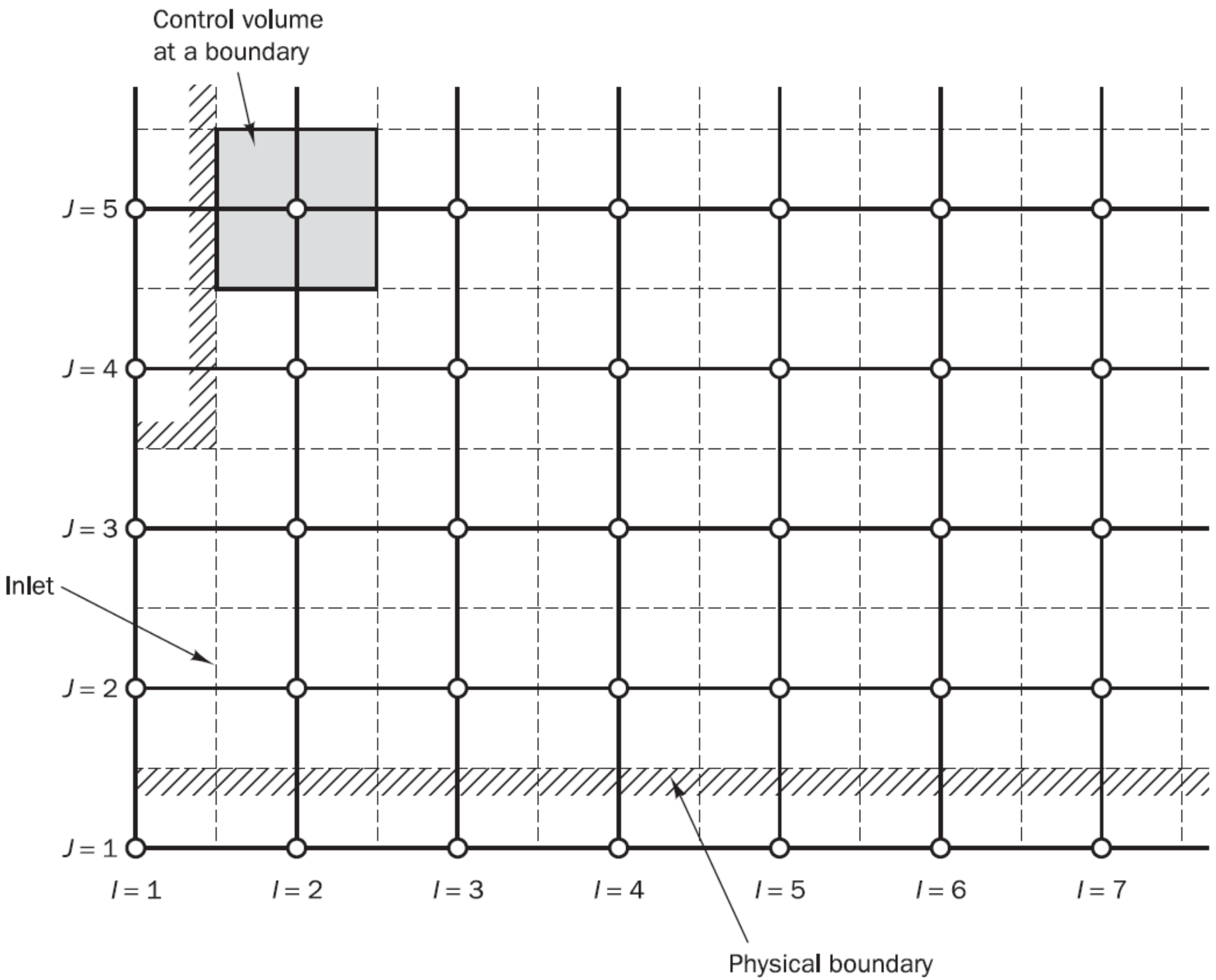
Introduction

All CFD problems are defined in terms of *initial* and *boundary conditions*. It is important that the user specifies these correctly and understands their role in the numerical algorithm. In transient problems the initial values of all the flow variables need to be specified at all solution points in the flow domain. However this involves no special measures other than initialising the appropriate data arrays in the CFD code. This chapter describes the implementation in the discretised equations of the finite volume method of the most common boundary conditions:

- ***inlet***
- ***outlet***
- ***wall***
- ***prescribed pressure***
- ***symmetry***
- ***periodicity (or cyclic boundary condition)***

Introduction

In constructing a staggered grid arrangement, we set up additional nodes surrounding the physical boundary, as illustrated in the figure. The calculations are performed at internal nodes only (I = 2 and J = 2 onwards). Two notable features of the arrangement are (i) *the physical boundaries coincide with scalar control volume boundaries* and (ii) *the nodes just outside the inlet of the domain (along I = 1 in the figure) are available to store the inlet conditions*. This enables the introduction of boundary conditions to be achieved with small modifications to the discretised equations for near-boundary internal nodes.



Control volume
at a boundary
J = 5
J = 4
J = 3
Inlet
J = 2
J = 1
I = 1
I = 2
I = 3
I = 4
I = 5
I = 6
I = 7
Physical boundary

Inlet BC

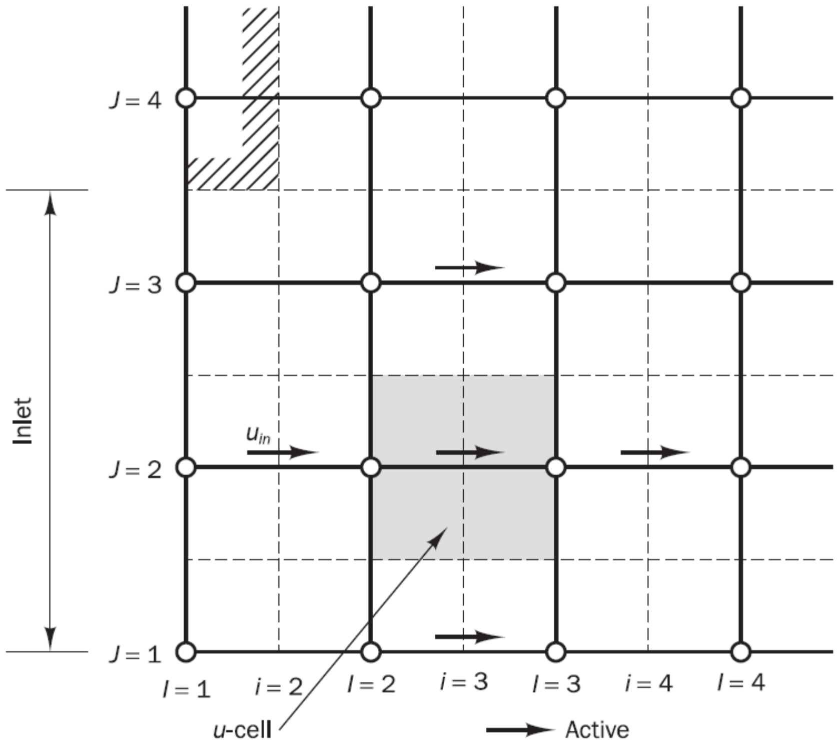
The *distribution of all flow variables* needs to be specified at inlet boundaries. Consider the case of an inlet perpendicular to the x-direction. The flow direction is from the left to the right in the diagrams.

Figures below show the grid arrangement in the immediate vicinity of an inlet for u- and v-momentum, scalar and pressure correction equation cells.

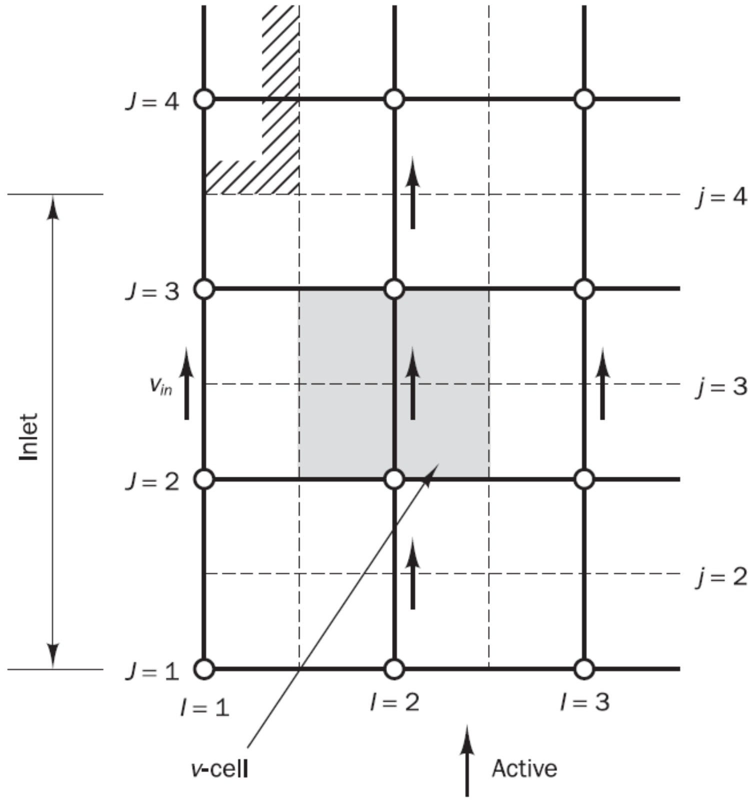
The *grid extends outside the physical boundary and the nodes along the line $I = 1$ (or $i = 2$ for u-velocity) are used to store the inlet values of flow variables* (indicated by u_{in}, v_{in}, φ_{in} and p'_{in}).

Just downstream of this extra node we start to solve the discretised equation for the first internal cell, which is shaded.

Inlet BC



J = 4
J = 3
J = 2
J = 1
Inlet
u_{in}
I = 1
i = 2
I = 2
i = 3
I = 3
i = 4
I = 4
u-cell
Active



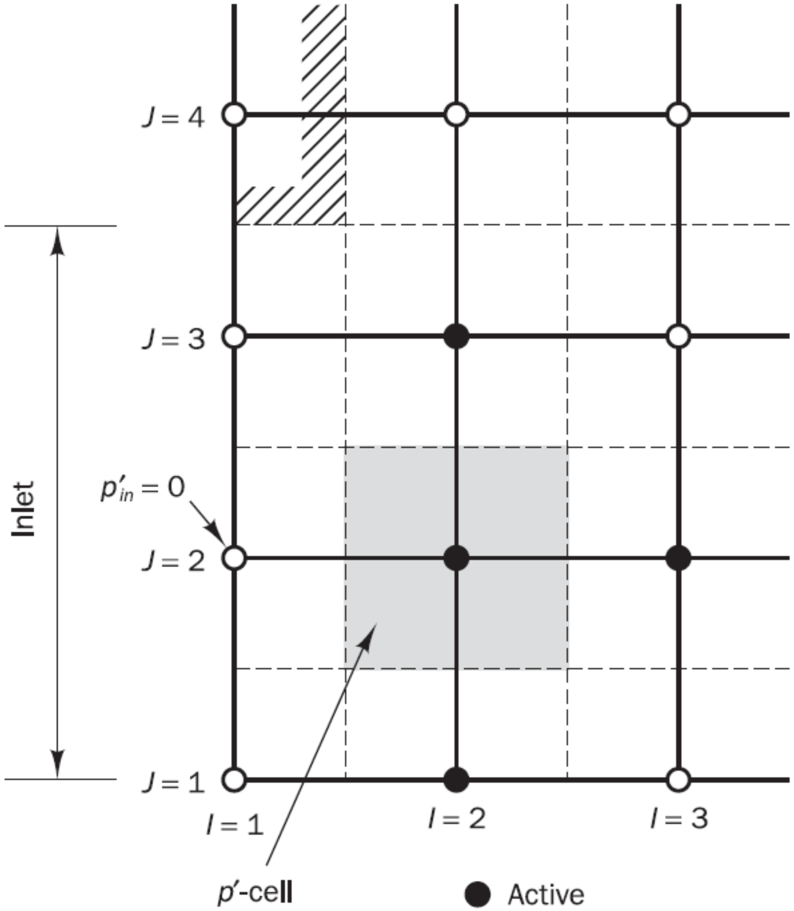
J = 4
J = 3
J = 2
J = 1
j = 4
j = 3
j = 2
Inlet
v_{in}
I = 1
I = 2
I = 3
v-cell
Active

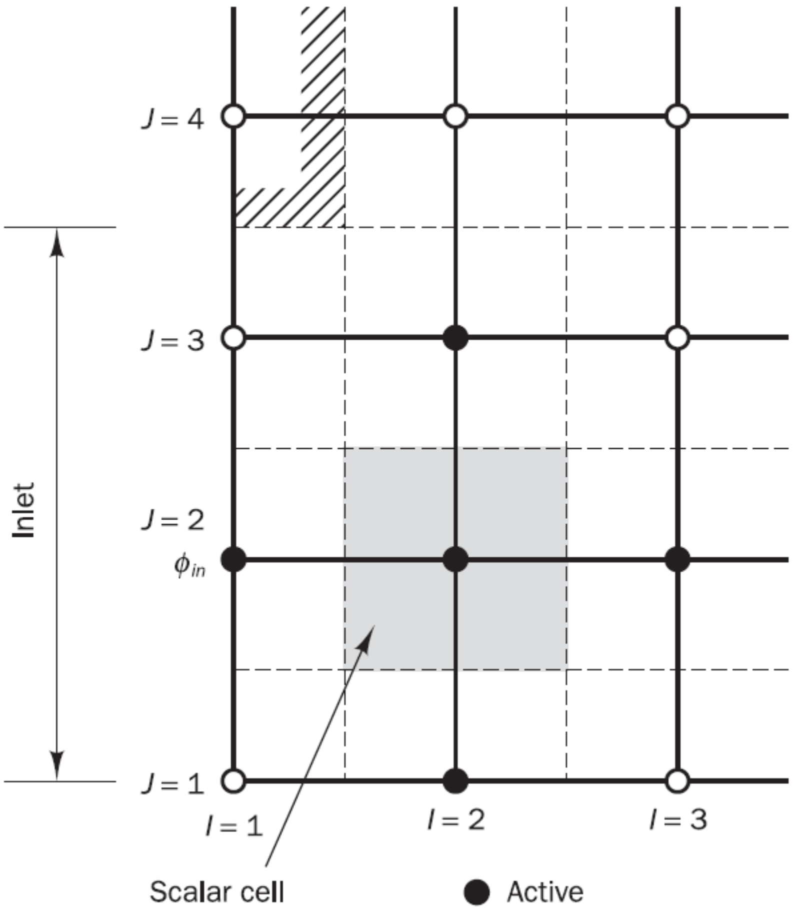
Inlet BC

In the figure, the active neighbour velocities are given by means of arrows and the active face pressures by open circles.

The figures indicate that all links to neighbouring nodes remain active for the first u-, v- and φ-cell, so to accommodate the inlet boundary condition for these variables, it is unnecessary to make any modifications to their discretised equations.

Inlet BC

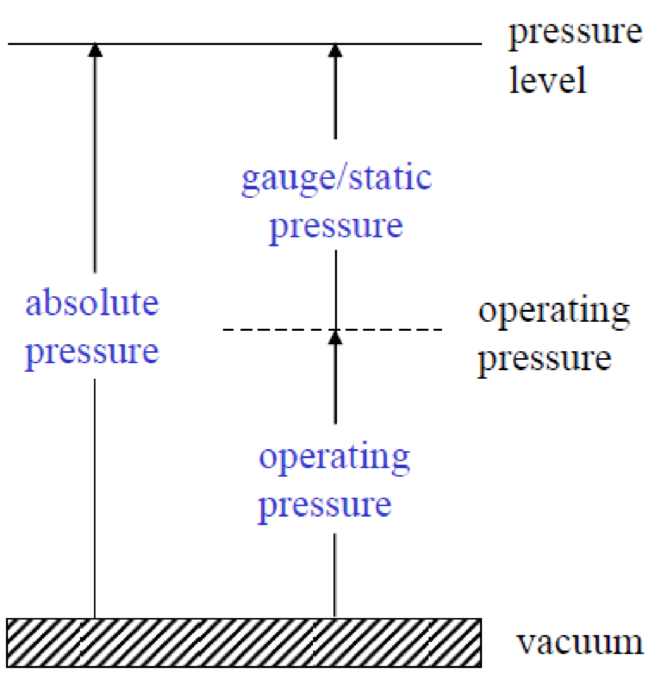

J = 4
J = 3
p′in = 0
J = 2
J = 1
I = 1
I = 2
I = 3
Inlet
p′-cell
Active


J = 4
J = 3
J = 2
ϕin
J = 1
I = 1
I = 2
I = 3
Inlet
Scalar cell
Active

- Reference (operating) pressure

The pressure field obtained by solving the pressure correction equation does not give absolute pressures. It is common practice to *fix the absolute pressure at one inlet node and set the pressure correction to zero at that node.*

Having specified a reference value, the absolute pressure field inside the domain can now be obtained.



- k and ε

The most accurate simulations can only be achieved by supplying measured inlet values of turbulent kinetic energy k and dissipation rate ε. However, such data are often not available.

Some CFD codes often estimate k and ε with an approximate formulae

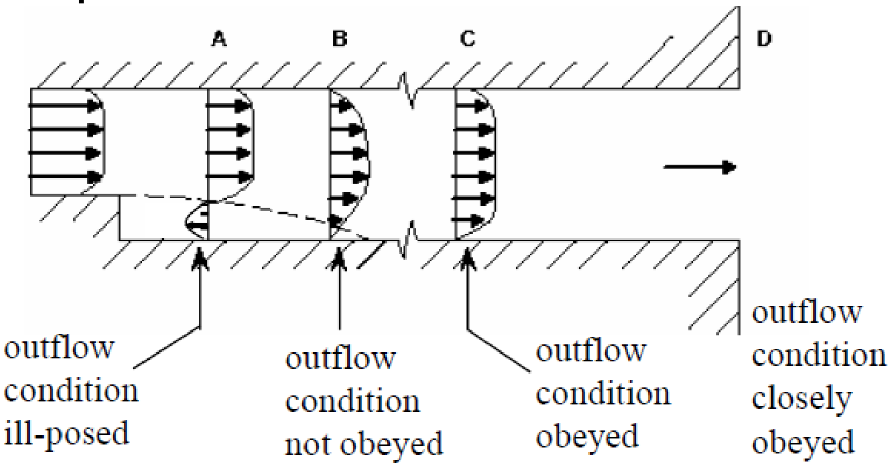
$$k = \frac{2}{3}(U_{ref} T_i)^2 \quad \varepsilon = C_\mu^{3/4} \frac{k^{3/2}}{\ell} \quad \ell = 0.07L$$

, based on a turbulence intensity, T_i, – typically between 1% and 6% – and a length scale.

Outlet BC

If the location of the outlet is selected *far away* from geometrical disturbances, the flow eventually reaches a *fully developed state* where *no change occurs in the flow direction*. In such a region we can place an outlet surface and state that the *gradients of all variables (except pressure) are zero in the flow direction*.

In other words, locate the outlet surface perpendicular to the flow direction and take gradients in the direction normal to the outlet surface equal to zero.

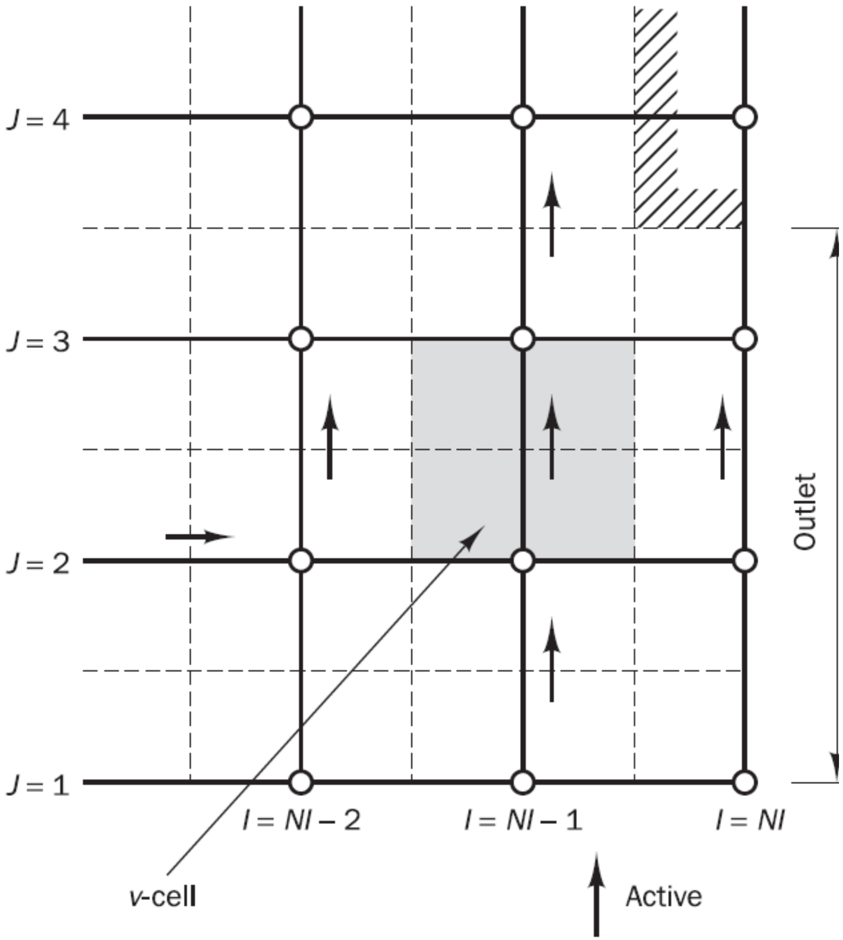


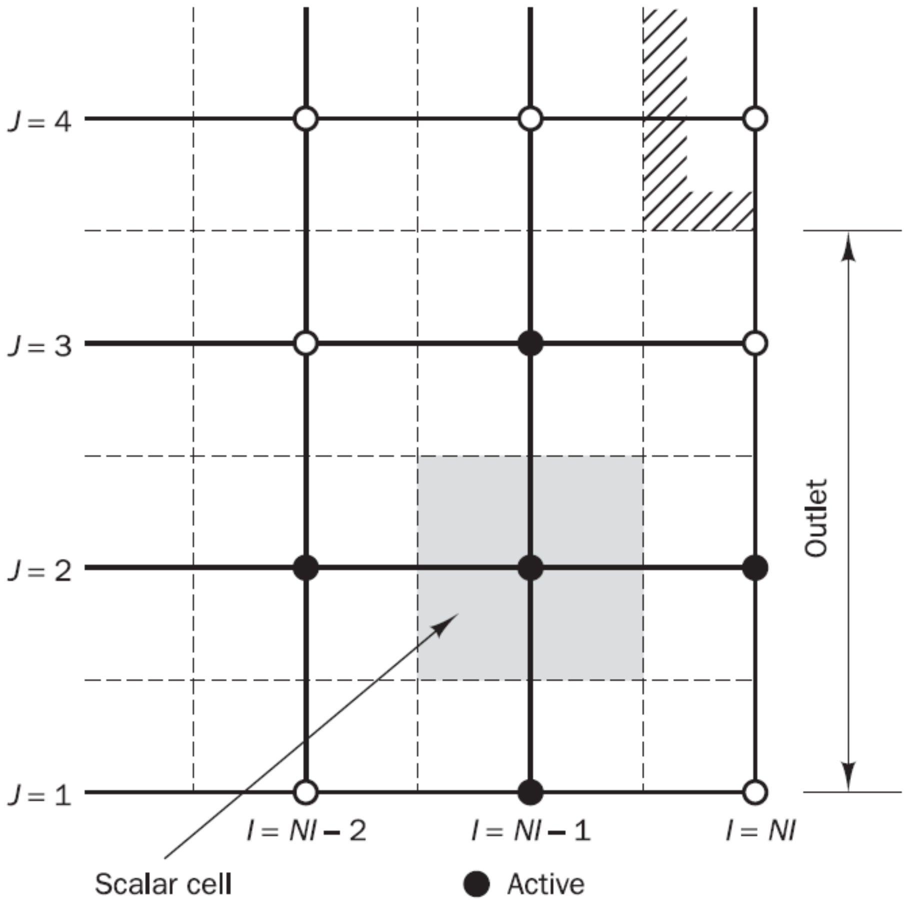
Outlet BC

If NI is the total number of nodes in the x-direction, equations are solved for cells up to I (or i) = NI − 1. Before the relevant equations are solved the values of flow variables at the next node (NI), just outside the domain, are determined by *extrapolation from the interior on the assumption of zero gradient at the outlet plane*. For the v- and scalar equations this implies setting

$$v_{NI,j} = v_{NI-1,j} \qquad \phi_{NI,J} = \phi_{NI-1,J}.$$

Outlet BC


J = 4
J = 3
J = 2
J = 1
I = NI – 2
I = NI – 1
I = NI
Outlet
v-cell
Active

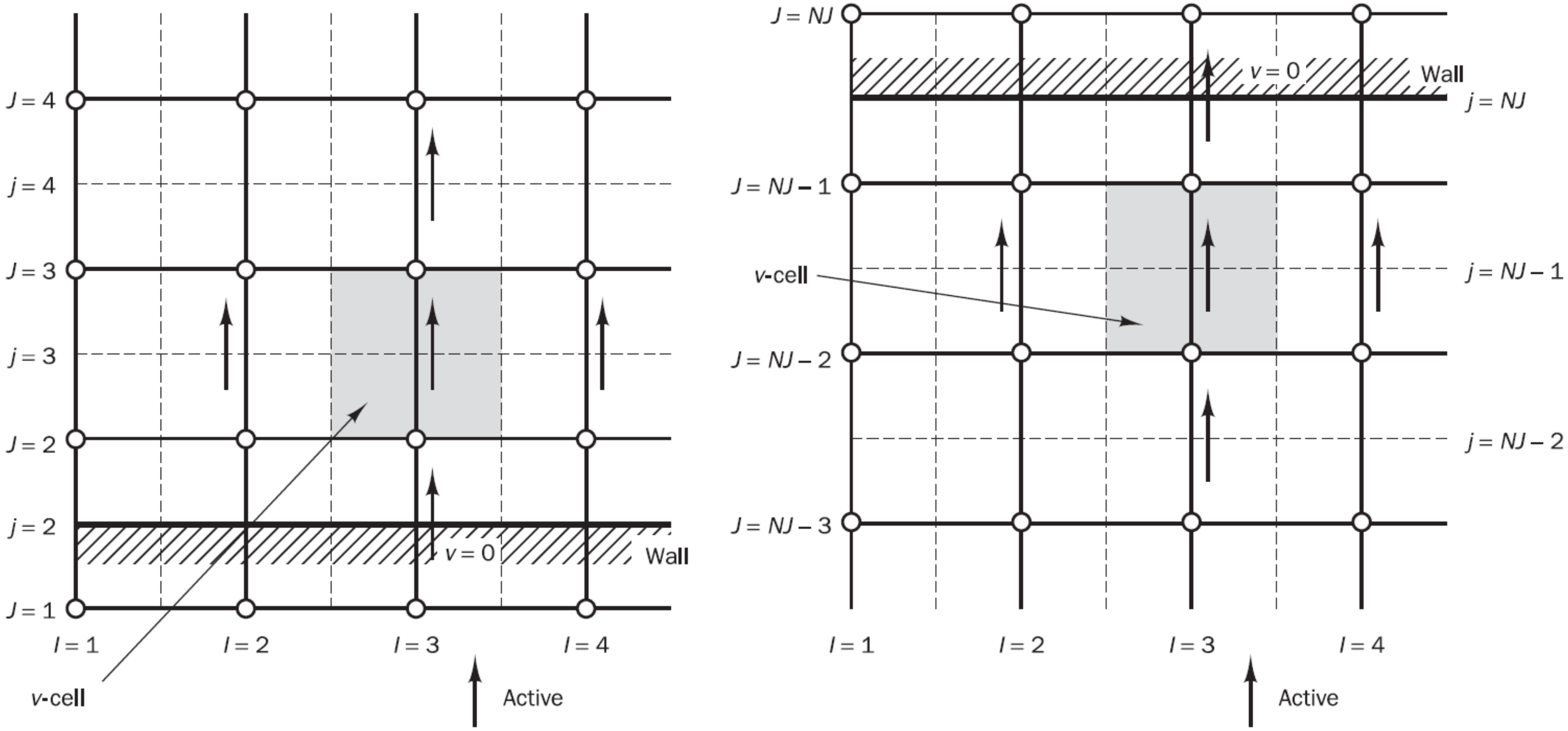

J = 4
J = 3
J = 2
J = 1
I = NI – 2
I = NI – 1
I = NI
Outlet
Scalar cell
Active

Wall BC

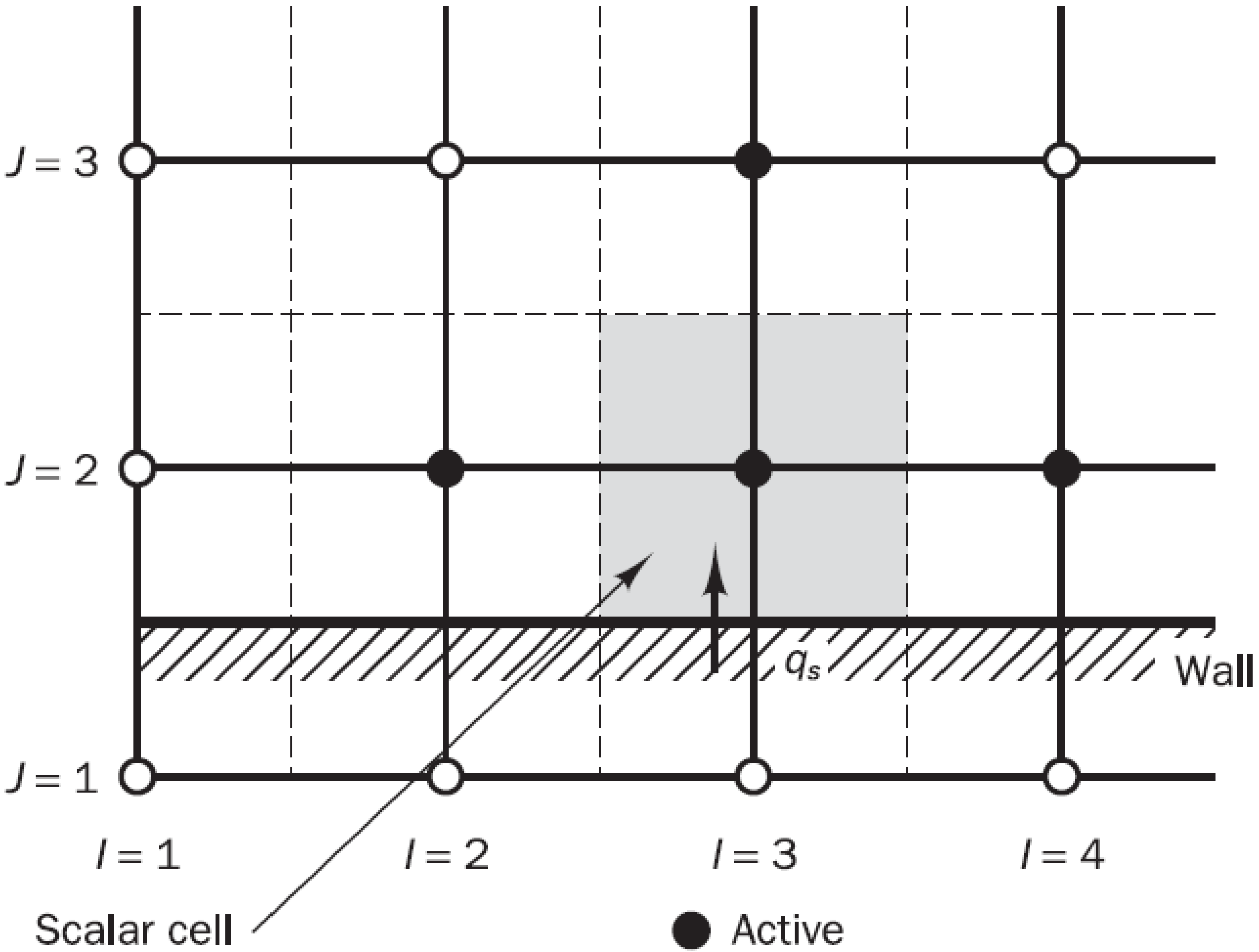
The *no-slip condition* ($u = v = 0$) is the appropriate condition for the velocity components at solid walls.

The normal component of the velocity can simply be set to zero at the boundary ($j = 2$), and the discretised momentum equation at the next v-cell in the flow ($j = 3$) can be evaluated without modification.

Wall BC


J = 4
j = 4
J = 3
j = 3
J = 2
j = 2
J = 1
I = 1
I = 2
I = 3
I = 4
v = 0
Wall
v-cell
Active
J = NJ
j = NJ
J = NJ – 1
j = NJ – 1
J = NJ – 2
j = NJ – 2
J = NJ – 3
v = 0
Wall
v-cell
Active

Wall BC



Wall BC

Immediately adjacent to the wall we have an extremely thin viscous sub-layer followed by the buffer layer and the turbulent core. The number of mesh points required to resolve all the details in a turbulent boundary layer would be prohibitively large, and normally we employ the *'wall functions' to represent the effect of the wall boundaries*.

The implementation of wall boundary conditions in turbulent flows starts with the evaluation of

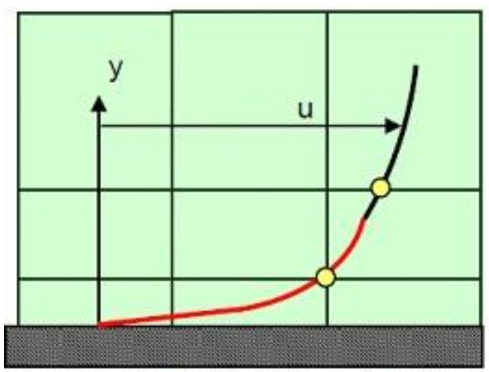
$$y^+ = \frac{\Delta y_P}{\nu}\sqrt{\frac{\tau_w}{\rho}}$$

where Δy_P is the distance of the near-wall node P to the solid surface

Wall BC

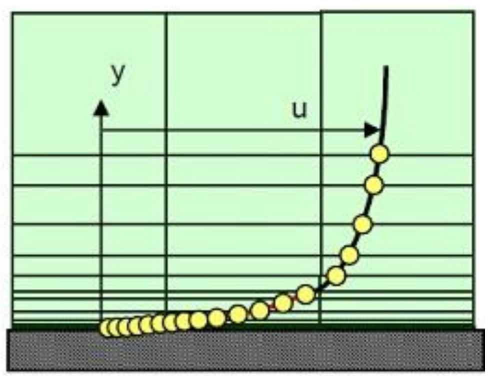
A near-wall flow is taken to be *laminar if y+ ≤ 11.63*. The wall shear stress is assumed to be entirely viscous in origin.

If *y+ > 11.63* the flow is turbulent and the wall function approach is used. The criterion places the changeover from laminar to turbulent near-wall flow in the buffer layer between the linear and log-law regions of a turbulent wall layer.

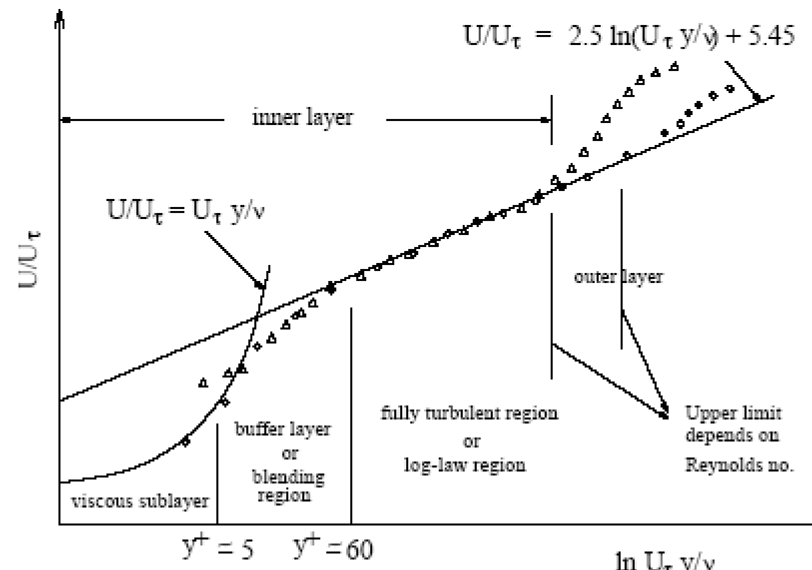


Wall functions used to resolve boundary layer

Boundary layer



Wall functions not used to resolve boundary layer



Constant pressure BC

The constant pressure condition is used in situations *where exact details of the flow distribution are unknown but the boundary values of pressure are known*.

Typical problems include external flows around objects, free surface flows, buoyancy-driven flows such as natural ventilation and fires, and also internal flows with multiple outlets.

There are several variations that can be useful in practical circumstances. Some codes apply (i) *a condition at inlet that fixes the stagnation pressure of the inlet flow just outside the domain (at i = 1) instead of the static pressure just inside the domain (at i = 2)* and/or (ii) *the extrapolation procedure at outlets for all variables including u*.

Symmetry BC

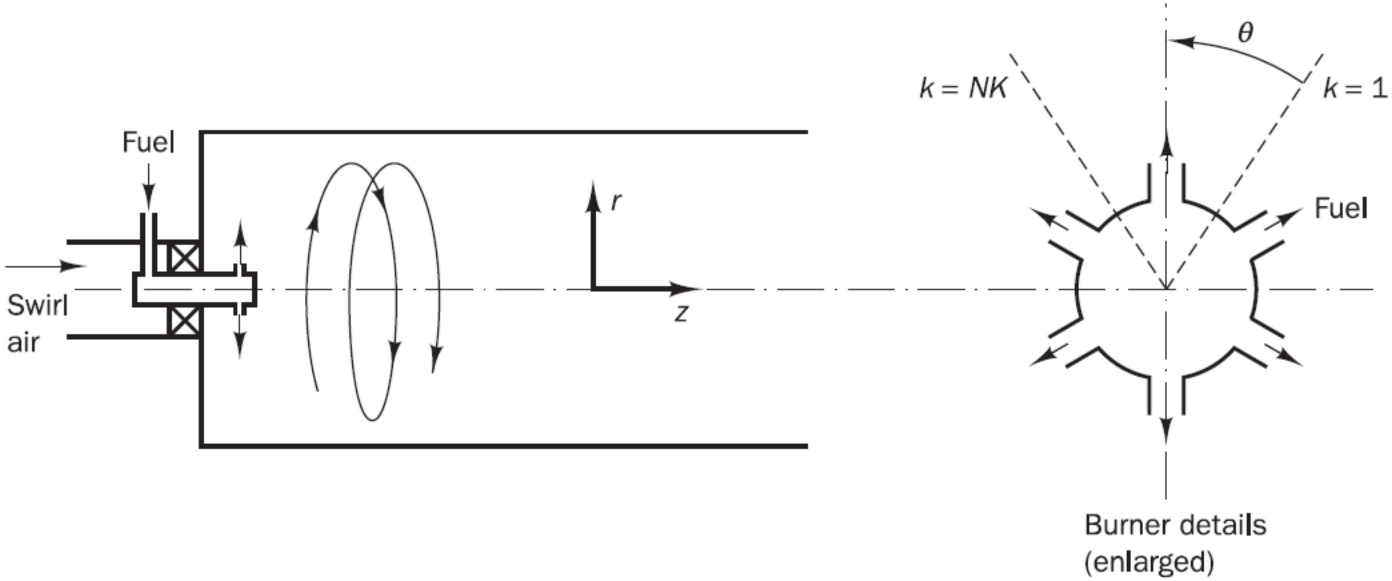
The conditions at a symmetry boundary are: (i) *no flow across the boundary* and (ii) *no scalar flux across the boundary*.

In the implementation, normal velocities are set to zero at a symmetry boundary, and the values of all other properties just outside the solution domain (say I or i = 1) are equated to their values at the nearest node just inside the domain (I or i = 2):

Periodic or cyclic BC

Periodic or cyclic boundary conditions arise due to a *different type of symmetry in a problem.*

Consider swirling flow in the cylindrical furnace shown in figure. In the burner arrangement gaseous fuel is introduced through six symmetrically placed holes and swirl air enters through the outer annulus of the burner.



Periodic or cyclic BC

This problem can be solved in cylindrical polar co-ordinates (z, r, θ) by considering a 60° angular sector as shown in the diagram, where k refers to r–z planes in the θ-direction.

The flow rotates in this direction, and under the given conditions the flow entering the first k-plane of the sector should be exactly the same as that leaving the last k-plane. This is an example of *cyclic symmetry*.

The pair of boundaries k = 1 and k = NK are called *periodic* or *cyclic boundaries*.

To apply cyclic boundary conditions we need to set the flux of all flow variables leaving the outlet cyclic boundary equal to the flux entering the inlet cyclic boundary.

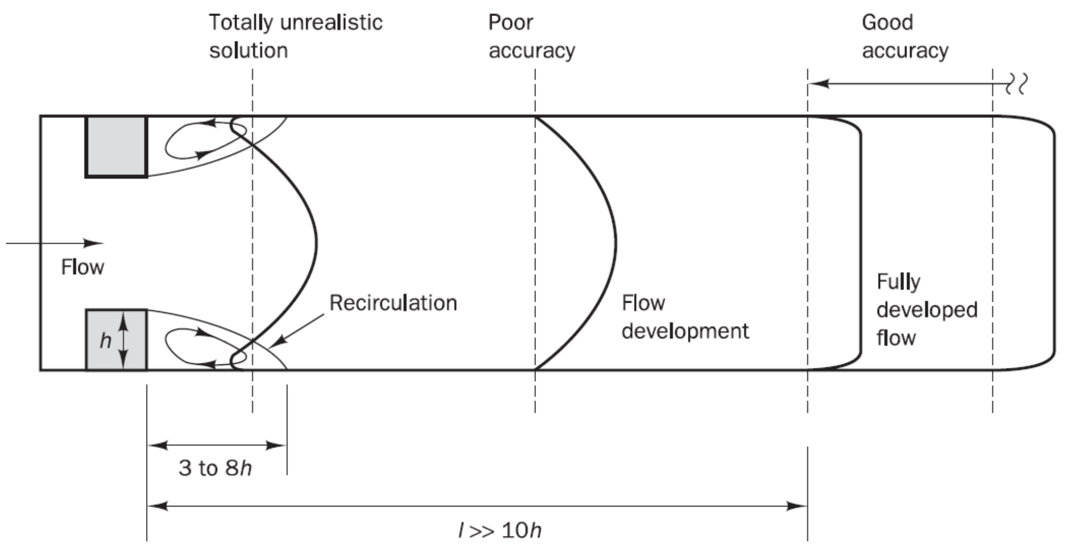
Final remarks

Flows inside a CFD solution domain are driven by the boundary conditions. In a sense the process of solving a field problem (e.g. a fluid flow) is nothing more than the extrapolation of a set of data defined on a boundary contour or surface into the domain interior.

It is of paramount importance that we supply physically realistic, well-posed boundary conditions, otherwise severe difficulties are encountered in obtaining solutions. The single, most common cause of *rapid divergence of CFD simulations* is the *inappropriate selection of boundary conditions*.

- Positioning of outlet boundaries

If outlet boundaries are placed too close to solid obstacles, it is possible that the flow has not yet reached a fully developed state (zero gradients in the flow direction), which may lead to sizeable errors. Figure below gives typical velocity profiles downstream of an obstacle, which illustrate the potential hazards.



Final remarks

If the outlet is placed close to an obstacle, it may range across a wake region with recirculation. Not only does the assumed gradient condition not hold, but there is an area of reverse flow where the fluid enters the domain whilst we had assumed an outward flow.

Of course, we cannot trust the solution if this condition arises. Somewhat further downstream there may not be reverse flow, but the zero-gradient condition does not hold since the velocity profile still changes in the flow direction.

Final remarks

It is imperative that the *outlet boundary is placed much further downstream than 10 heights downstream of the last obstacle* to give accurate results.

For high accuracy it is necessary to demonstrate that the *interior solution is unaffected by the choice of location of the outlet by means of a sensitivity study for the effect of different downstream distances*.

- Near-wall grid

The most accurate way of solving turbulent flows in a CFD code is to make use of the good empirical fits provided by the wall function approach.

To obtain the same accuracy by means of a simulation which includes points inside the (laminar) linear sub-layer, the grid spacing must be so fine as to be uneconomical.

The criterion that y+ must be greater than 11.63 sets a lower limit to the distance from the wall Δy_P of the nearest grid point.

Final remarks

The main mechanism for accuracy improvement available to us is grid refinement, but in a turbulent flow simulation we must ensure that, whilst refining the grid, the value of y+ stays greater than 11.63 and is preferably between 30 and 500.

Reference

"An introduction to computational fluid dynamics – The finite volume method", 2nd edition, H. K. Versteeg and W. Malalasekera, Pearson Education Limited, 2007.