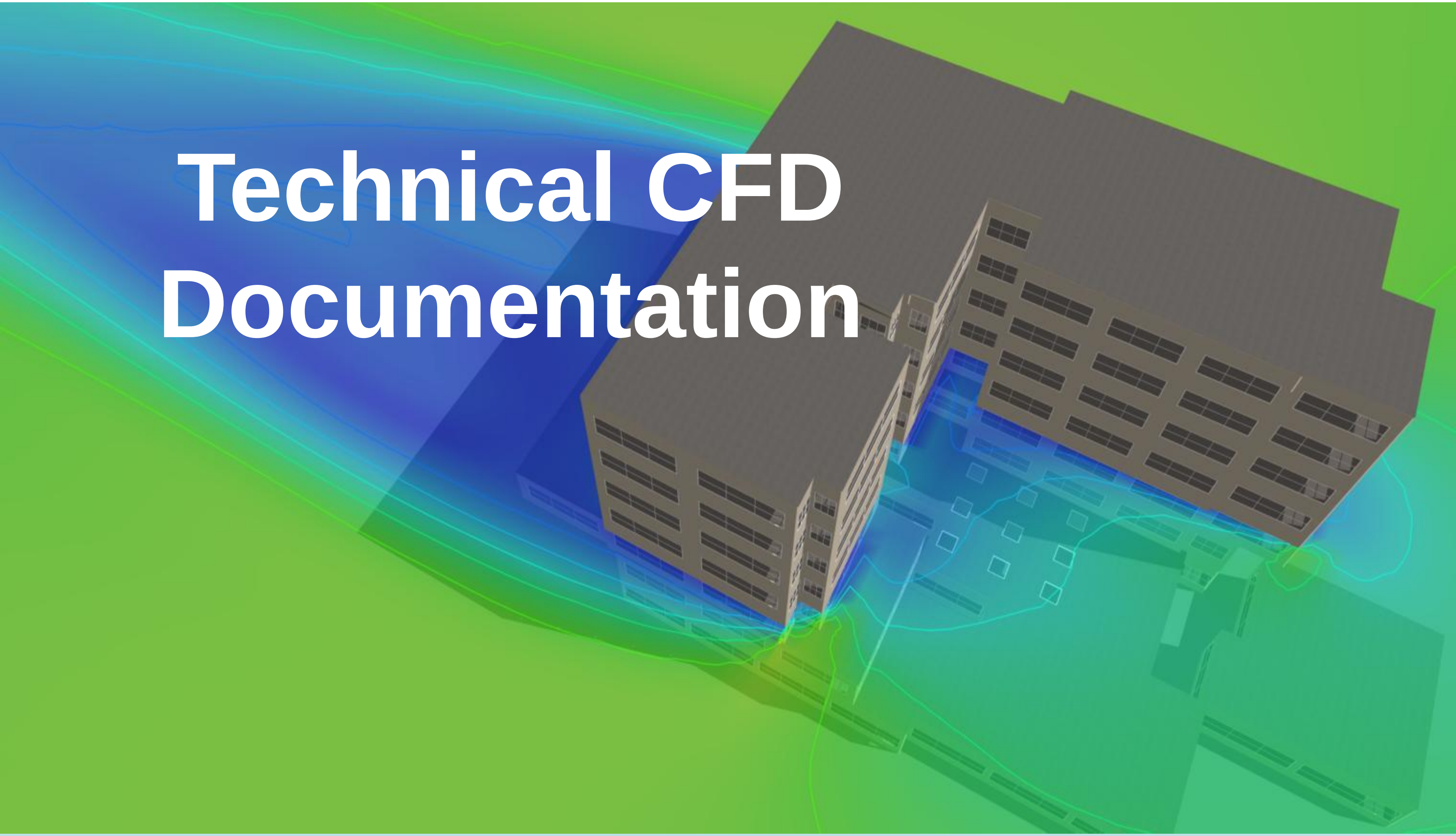
DesignBuilder Software

Technical CFD Documentation



DesignBuilder Software

What is CFD?

- Computational Fluid Dynamics (CFD) is the term used to describe a family of numerical methods used to calculate the temperature, velocity and other fluid properties throughout a region of space.
- CFD applied to buildings provides the designer with information on probable air velocities, pressures and temperatures occurring in and around building spaces with specified boundary conditions including climate, internal heat gains and HVAC systems.

How does CFD work?

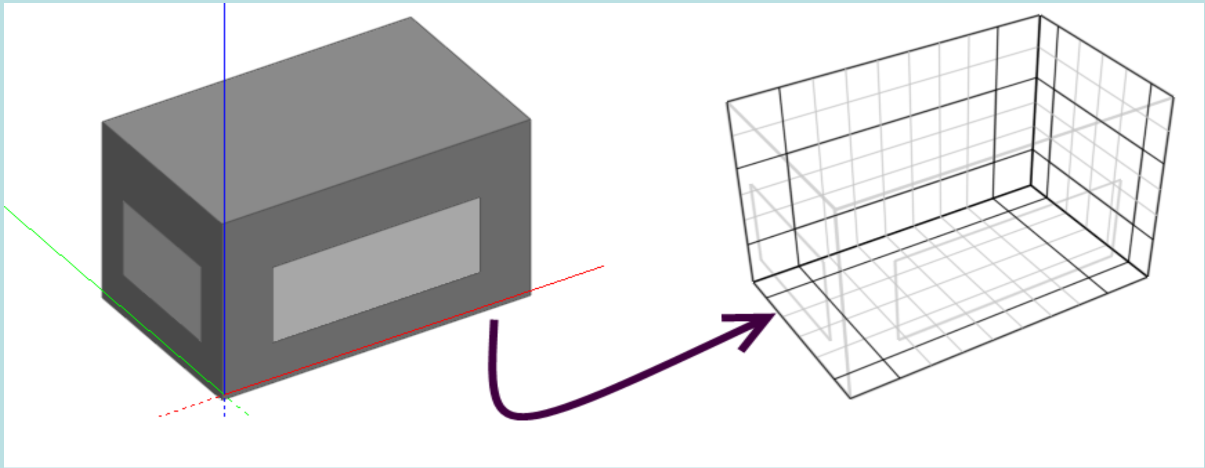
- DesignBuilder CFD is based on a method known as the finite volume (FV) method
- The method involves the solution of a set of partial differential equations (PDEs) describing the transport of momentum, energy and turbulence quantities

$$\underbrace{\frac{\partial}{\partial t}(\rho\phi)}_{\text{Transient}} + \underbrace{\mathrm{div}(\rho u\phi)}_{\text{Convection}} = \underbrace{\mathrm{div}(\Gamma\ \mathrm{grad}\ \phi)}_{\text{Diffusion}} + \underbrace{S}_{\text{Source}}$$

Equation	Variable
Momentum	u, v and w velocities
Energy	temperature
Turbulence KE	turbulence kinetic energy
Turbulence Diss	dissipation rate of turbulence kinetic energy

How does CFD work?

- Partial differential equations are converted into a set of simultaneous algebraic equations
- Building space under analysis is divided into a set of non-overlapping adjoining rectilinear cells (finite volume grid)



- Algebraic equations are set up for each grid cell and the whole set of equations solved using a numerical method

How does CFD work?

What is a partial differential equation?

- A PDE is a type of equation that is used to describe the variation of a dependent variable (such as temperature or velocity) with a number of independent variables (such as time and distance)
- The PDEs are made up of a number of terms incorporating the dependent variable and a multiplier or coefficient
- The PDE coefficients themselves can contain the same dependent variables that they are associated with and consequently the equations cannot be solved using analytical methods

Momentum equation:

$$\frac{\partial}{\partial t}(\rho\, u_i) + \underbrace{\frac{\partial}{\partial x_j}(\rho\, u_j\, u_i)}_{\text{Convection}} = -\frac{\partial \Pi}{\partial x_i} + \frac{\partial}{\partial x_j}\left\{(\mu + \mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right\} - g_i$$

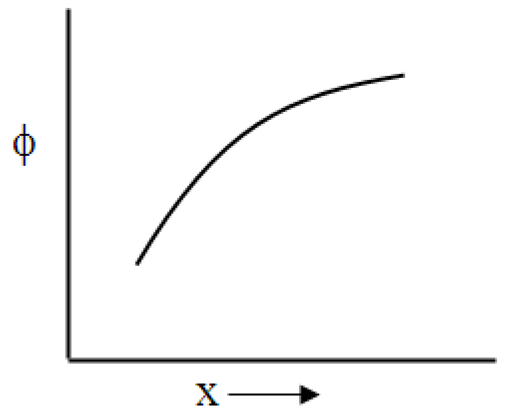
convection term coefficient incorporates velocity

How does CFD work?

The solution – a numerical approach

- The variation of the PDE dependent variable is continuous and may be visualised in the form of a curve

$$\frac{\partial}{\partial x_j}(\rho u_j \phi_i) = \frac{\partial}{\partial x_j}\left(\Gamma \frac{\partial \phi_i}{\partial x_j}\right)$$



- The continuous nature of these non-linear variations can be approximated by a number of linear relationships, i.e. the curve can be represented as a series of straight lines
- The idea behind this is that equations representing straight lines are easily solved

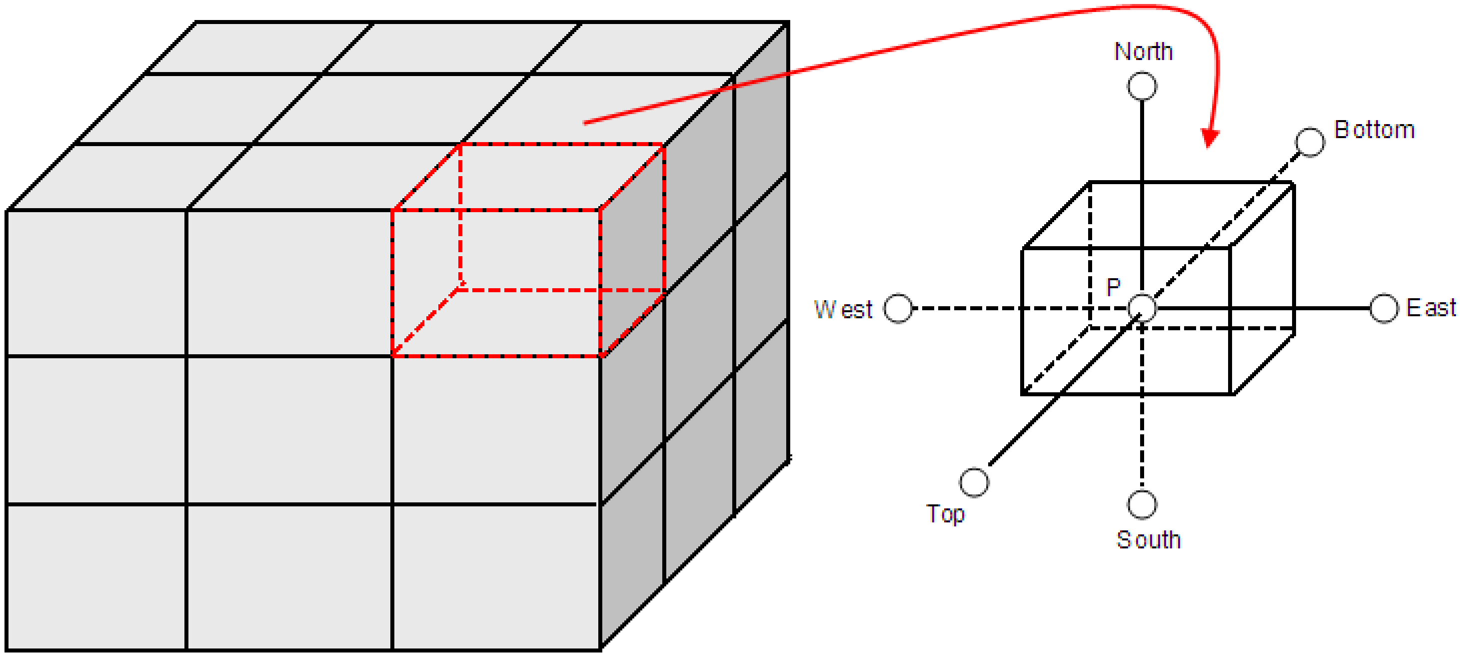
How does CFD work?

The solution – a numerical approach (cont'd)

- This process is known as "discretisation"
- To start the process, we divide the building space into a number of non-overlapping volumes or "control" volumes collectively known as the finite volume grid
- Each control volume surrounds a grid point at which the dependent variable is evaluated

How does CFD work?

The solution – a numerical approach (cont'd)



Finite Volume (FV) grid

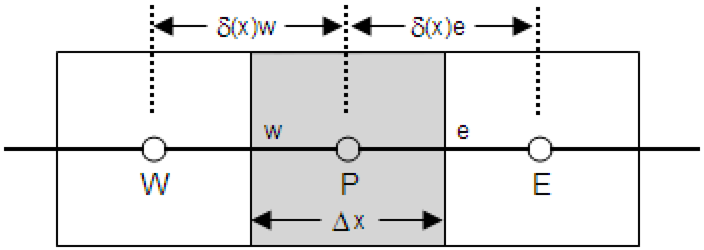
FV grid control volume with neighbouring points

DesignBuilder Software

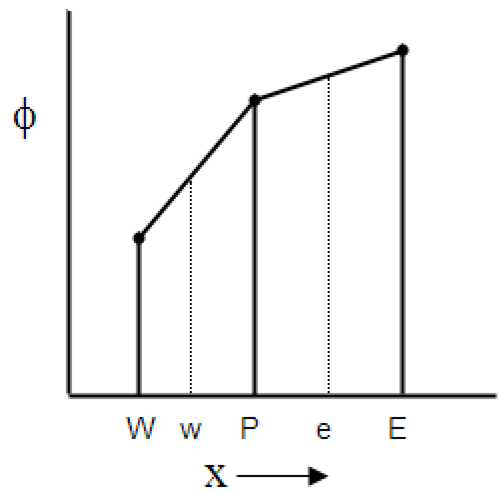
How does CFD work?

The solution – a numerical approach (cont'd)

- The PDEs can then be discretised across these control volumes using linear profiles to represent the variation of the dependent variables between the grid points and their neighbours



Finite volume grid in one dimension



Discretisation over control volume:

$$(\rho u\phi)_e - (\rho u\phi)_w = \left(\Gamma\frac{\partial\phi_i}{\partial x_j}\right)_e - \left(\Gamma\frac{\partial\phi_i}{\partial x_j}\right)_w$$

Re-write equation using linear profile shown at top right:

$$\tfrac{1}{2}(\rho u)_e(\phi_E+\phi_P) - \tfrac{1}{2}(\rho u)_w(\phi_P+\phi_W) = \frac{\Gamma_e(\phi_E-\phi_P)}{(\delta x)_e} - \frac{\Gamma_w(\phi_P-\phi_W)}{(\delta x)_w}$$

How does CFD work?

The solution – a numerical approach (cont'd)

- These discretised equations are then re-arranged to take the form of a set of simple algebraic equations that can be solved using basic numerical methods:

$$a_P\phi_P = a_E\phi_E + a_W\phi_W + a_N\phi_N + a_S\phi_S + a_T\phi_T + a_B\phi_B$$

$$a_P = a_E + a_W + a_N + a_S + a_T + a_B$$

where a_E, a_W, a_N, a_S, a_T and a_B are the combined diffusion-convection coefficients:

$$a_E = \frac{\Gamma_e}{\delta x} - \frac{\rho u_e}{2}$$

How does CFD work?

The iterative nature of the solution and convergence

- The numerical methods used to solve the equation set are iterative whereby the equations are repeatedly re-constructed and solved until there is no change in the dependent variables
- The algebraic equations are constructed in such a way that if the coefficients were constant, a converged solution would be guaranteed using the Gauss-Siedel method (simplest numerical method for solving simultaneous equations)
- The dependent variable coefficients contain the dependent variables themselves and are therefore not constant
- In practice, if the coefficients are of similar magnitude throughout and change gradually, a converged solution can normally be achieved

How does CFD work?

Outer and inner iterations

- Because the coefficients are changing, the iterative procedure uses an inner iterative procedure to solve the dependent variable equations within an outer iterative procedure to update the dependent variable coefficients
- At each outer iteration, only tentative values of the dependent variables are realised and consequently only a few inner iterations are required

How does CFD work?

Relaxation factors

- In order to prevent the equation coefficients from changing too quickly, the change in dependent variables from one outer iteration to another can be slowed by ‘relaxing’ them
- The traditional method of relaxation is the relaxation factor which combines a proportion of the dependent variable from the previous outer iteration with a proportion from the current iteration
- A relaxation value of 1.0 uses 100% of the current dependent variable value whereas a relaxation factor of 0.5 would combine 50% of the previous iteration value with 50% of the current value.

How does CFD work?

Dynamic solution and false time steps

- DesignBuilder CFD is steady-state, i.e. it calculates a ‘snap-shot’ in time
- In a dynamic CFD solution, the transient term acts as a very effective inertial relaxation factor
- The equation set in DesignBuilder CFD is actually constructed in a fully dynamic form
- The time steps in the transient terms are replaced with ‘false time steps’
- False time steps are generally more effective than relaxation factors

How does CFD work?

Convergence and termination residuals

- The solution may be considered converged when there is no perceptible change in the dependent variables from one outer iteration to another
- In cases where heavy under-relaxation or very small false time steps are employed, changes in dependent variables may not be obvious
- An attractive feature of the control volume formulation is that once convergence is achieved, integral conservation of quantities such as mass, momentum and energy is exactly satisfied for each cell, any group of cells and of course for the whole domain

How does CFD work?

Convergence and termination residuals (cont'd)

- A more meaningful indication of convergence is to consider the degree to which conservation of mass, momentum and energy is satisfied for the calculation domain
- A residual may be calculated by considering the overall balance of a particular quantity using the algebraic equation for the appropriate dependent variable:

$$R = \Sigma a_{nb}\phi_{nb} + b - a_P\phi_P$$

- Maximum residuals for each dependent variable are calculated and compared with a supplied termination residual to determine whether or not convergence has been achieved

How does CFD work?

Discretisation scheme – Upwind, Hybrid and Power-law

- The formulation of the combined convection-diffusion coefficient whereby the interface convection coefficient is determined from a simple average of the grid point velocity and the neighbouring grid point velocity is found to result in very unstable solutions
- As previously noted, the algebraic equations are constructed to comply with a set of rules that would guarantee convergence for an equation set involving constant coefficients
- One of the rules requires that coefficients must not become negative
- The simple average convection coefficient formulation can lead to a negative coefficient and consequently a non-convergent solution

How does CFD work?

Discretisation scheme – the Upwind scheme

- A remedy for this difficulty is to ensure that the convected value of an interface property is equal to the value of the property on the upwind side of the face

$$a_E = \frac{\Gamma_e}{\delta x} - \frac{\rho u_e}{2}$$

$$\Downarrow$$

$$a_E = \frac{\Gamma_e}{\delta x} + \max\left(-\frac{\rho u_e}{2}, 0\right)$$

- The rationale behind this measure is that convection (unlike diffusion) may be considered a one-way process in that properties upstream of a point can affect properties downstream but not the other way round

DesignBuilder Software

How does CFD work?

Discretisation scheme – Hybrid and Power-law schemes

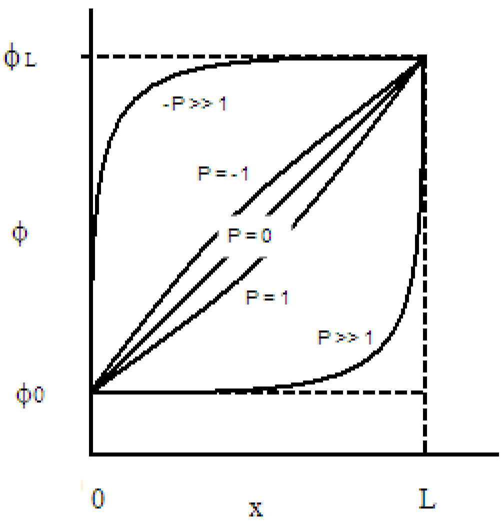
- If we take our basic PDE, we can obtain an exact analytical solution by assuming that the convection and diffusion coefficients are constant

$$\frac{d}{dx_j}(\rho u_j \phi_i) = \frac{d}{dx_j}\left(\Gamma \frac{d\phi_i}{dx_j}\right)$$

$$\downarrow$$

$$\frac{\phi_P - \phi_P}{\phi_L - \phi_0} = \frac{\exp(P_x/L) - 1}{\exp(P) - 1}$$

$$\text{Peclet number, } P = \frac{\rho u L}{\Gamma}$$



How does CFD work?

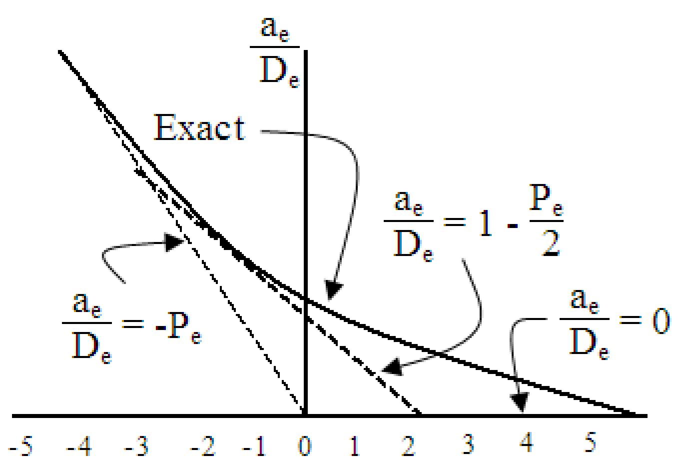
Discretisation scheme – Hybrid and Power-law schemes (cont’d)

- The ratio of the strengths of convection and diffusion may be measured using the Peclet number, P
- For high absolute values of P, the value of the dependent variable at the interface (i.e. x = L/2) can be seen to be nearly equal to the value at the upwind boundary which is the assumption made by the upwind scheme
- The dependent variable gradient at high absolute values of P is nearly zero at the interface (i.e. the line is horizontal)
- The upwind scheme always calculates diffusion assuming a linear relationship between the dependent variable and distance and therefore overestimates diffusion at large absolute values of P

How does CFD work?

Discretisation scheme – Hybrid scheme

- The hybrid scheme attempts to remedy this over-estimation of diffusion by representing the combined convection-diffusion coefficients with three straight line relationships for different ranges of Peclet number



For Pe < -2,

$$\frac{a_e}{D_e} = -P_e$$

For –2 =< Pe =< 2,

$$\frac{a_e}{D_e} = 1 - \frac{P_e}{2}$$

For Pe > 2

$$\frac{a_e}{D_e} = 0$$

Where

$$D = \frac{\Gamma_e}{\delta x}$$

How does CFD work?

Discretisation scheme – Hybrid scheme (cont'd)

- The simple average convection coefficient formulation can then be replaced with a formula combining the three straight lines:

$$a_E = \frac{\Gamma_e}{\delta x} - \frac{\rho u_e}{2}$$

$$a_E = \max\left(-\rho u_e \, , \frac{\Gamma_e}{\delta x} - \frac{\rho u_e}{2}, 0 \right)$$

How does CFD work?

Discretisation scheme – power-law scheme

- The departure of the hybrid scheme from the exact solution is quite marked when the absolute value of the Peclet number is equal to 2
- A better approximation to the exact curve is provided by the power-law scheme

For Pe < -10,

$$\frac{a_e}{D_e} = -P_e$$

For –10 =< Pe =< 0,

$$\frac{a_e}{D_e} = (1 + 0.1P_e)^5 - P_e$$

For 0 =< Pe =< 10,

$$\frac{a_e}{D_e} = (1 +- 0.1P_e)^5$$

For Pe > 10

$$\frac{a_e}{D_e} = 0$$

How does CFD work?

Discretisation scheme – power-law scheme

- The simple average convection coefficient formulation can then be replaced with a formula incorporating the power-law relationship:

$$a_E = \frac{\Gamma_e}{\delta x} - \frac{\rho u_e}{2}$$

$$\Downarrow$$

$$a_E = D_e \max\left(0, \left(1 - \frac{0.1\,|F_e|}{D_e}\right)^5\right) + \max(0, -F_e)$$

where, $F = \frac{\rho u_e}{2}$ $D = \frac{\Gamma_e}{\delta x}$

Geometric Considerations

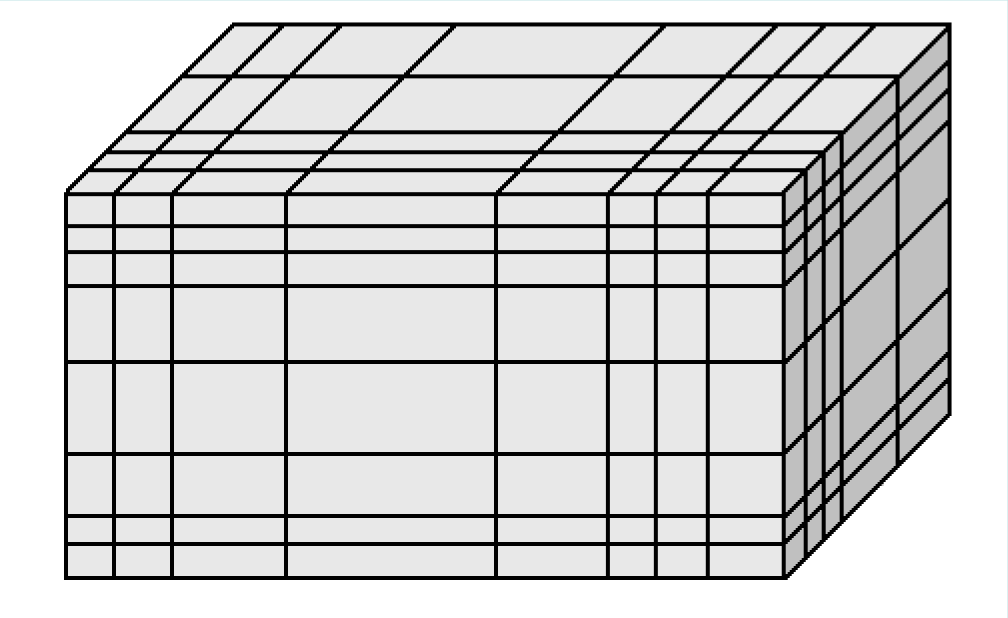
Finite Volume Grid

- The space across which the calculations are to be conducted is first divided into a number of non-overlapping adjoining cells which are collectively known as the finite volume grid.
- The grid is automatically generated from key coordinates obtained from the model geometry along each major grid axes throughout the calculation domain.
- These key coordinates, extended from the X, Y and Z-axes across the width, depth and height of the domain respectively are called ‘grid lines’.
- The distances between grid lines along each axis are called ‘regions’ and these regions are initially spaced using a supplied default grid spacing.

Geometric Considerations

Finite Volume Grid

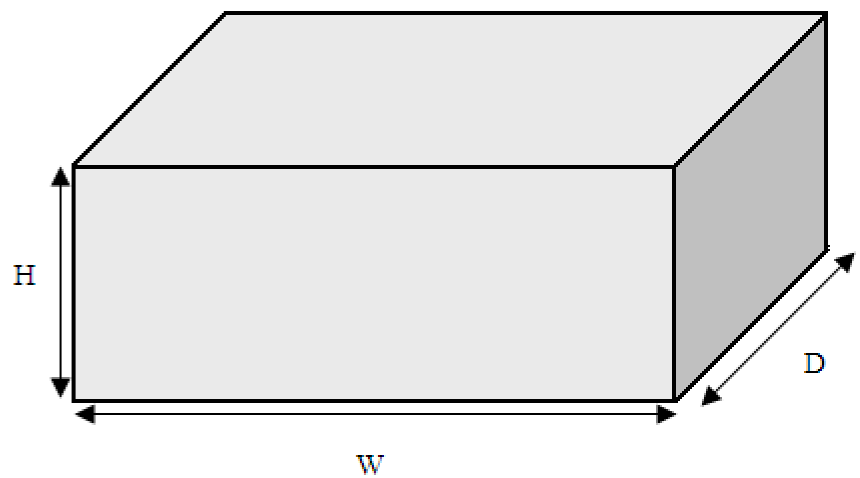
- The grid used by DesignBuilder CFD is a non-uniform rectilinear Cartesian grid, which means that the grid lines are parallel with the major axes and the spacing between the grid lines enables non-uniformity.



Geometric Considerations

Cell Aspect Ratio

- The cell aspect ratio is the ratio of the maximum cell dimension to the minimum cell dimension



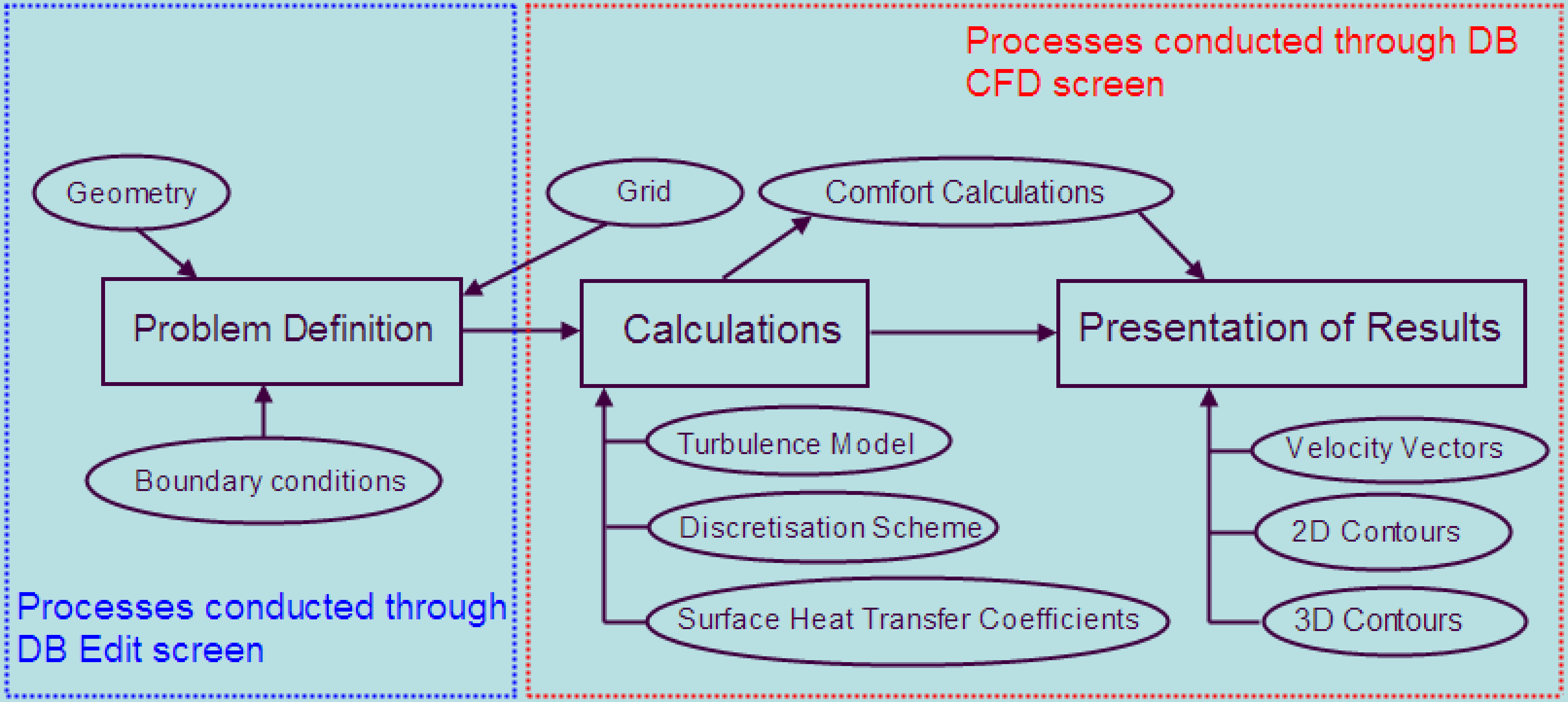
$$\text{cell aspect ratio} = \frac{\max(H,W,D)}{\min(H,W,D)}$$

Geometric Considerations

Cell Aspect Ratio (cont'd)

- If the cell aspect ratio is high, the equation coefficients can become widely different in magnitude which will result in large changes occurring in some variables but not in others which in turn can lead to an oscillating unstable solution

CFD Workflow



<u>Problem Definition</u>

- Define geometry using DesignBuilder geometric modeller

CFD Workflow

Problem Definition (cont'd)

- Select required domain (building, building block or zone)
- Set default surface and window temperature boundary conditions (not required for external flows)
- Add surface boundary conditions including temperature patches, HVAC supplies, extracts, etc. (not required for external flows)
- Add CFD component blocks and/or assemblies representing occupants, radiators, fan-coil units, etc. (not required for external flows)
- Finite volume (FV) grid automatically generated from geometry during CFD project creation
- Edit FV grid as required

CFD Workflow

Calculations

- Turbulence model:
  - Constant effective viscosity
  - k-ε
- Discretisation scheme:
  - Upwind
  - Hybrid
  - Power-law
- Outer iterations
- Isothermal

CFD Workflow

Calculations (cont'd)

- Specification of surface heat transfer coefficients:
  - Calculated (wall functions)
  - User supplied
- Initial conditions
  - X-axis velocity component
  - Y-axis velocity component
  - Z-axis velocity component
  - Temperature
- Cell monitor
  - Cell location
  - Monitored variable

CFD Workflow

Calculations (cont'd)

- Residual display
- Dependent variable control settings
  - Inner iterations
  - False time step
  - Relaxation factor
  - Termination residual

CFD Workflow

Presentation of Results

- Velocity vectors
  - Scale factor
  - Maximum vector length
- Contours
  - Velocity
  - Pressure
  - Temperature
  - PMV
  - PPD

CFD Workflow

Presentation of Results (cont'd)

- Filled Contours
  - Velocity
  - Pressure
  - Temperature
  - PMV
  - PPD
- 3-D Contours
  - Velocity
  - Pressure
  - Temperature
  - PMV
  - PPD