DesignBuilder Simulation Training
Natural Ventilation

Driven by

- Buoyancy Effect
- Wind

Analysis tools

- Airflow Network models
- Computational Fluid Dynamic (CFD)
- Wind Tunnel study (measurement)

Figure Source: J Good et al, SimBuild 2008
Airflow Network Model

- Developed by NIST (Walton 1989)
- Coupled to E+
- Bulk airflow movement, average temperatures
- Nodes connected by air flow elements
- Wind Pressure Coefficient ($C_p$) values need to be defined – Input/Auto
Airflow Network Model..contd

Cp determination

• Depends on build shape, wind direction, nearby buildings, vegetation, terrain etc.

• Can be only obtained from Wind tunnel test

• Default Values based on AIVC Guide* and ASHREA handbook of fundamentals

• Restricted to low rise rectangular buildings

• Wind Pressure: \( P_w = 0.5 \cdot \rho \cdot C_p \cdot v_z^2 \)
Airflow Network Model..contd

CP Values : Source: ASHRAE handbook of fundamentals, 2005 pp 16.5
AIRNET Model.. contd

Plan view of a simple airflow network showing a possible airflow pattern

Source: Lixing Gu, proceedings: Building Simulation 2007
2 Modes

DesignBuilder offers 2 Nat vent modes:
• Scheduled
• Calculated
Scheduled Nat Vent

- Exterior airflow + schedule set directly
- Infiltration is constant.
- Interior airflow through ‘mixing’
- Cooling set point temperatures
- Avoidance of heating
Tutorial 1

- Basic Scheduled natural ventilation model
- 2 zones
- Controlled Natural ventilation
- Infiltration
Tutorial 2

Same as first but with Internal Airflow

- Draw a large hole in partition
- Check change in results
Tutorial 3

Calculated Natural ventilation:

• Natural ventilation model option > “Calculated”
• Set window opening % + schedule
• Infiltration
• Check results
Tutorial 4

As previous but with variations:

• Check effects of modulation,
• Wind effects
• Internal vent airflow
CFD Introduction

• Computational Fluid Dynamics (CFD) is an analytical procedure to predict the flow of fluids (air, water etc.)

• Developed for the aerodynamics industry as a faster alternative to wind tunnel testing

• Can compute very accurately the temperature, velocity, direction of flow, pressure and energy (momentum) of a fluid particle in the solution domain
CFD – Navier Stokes Eqns

Focus: Incompressible fluids

Continuity and Momentum Equations

\[
\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_i} = 0
\]

\[
\frac{\partial u_j}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_i} = -\frac{1}{\rho} \frac{\partial p}{\partial x_j} + \frac{\mu}{\rho} \frac{\partial^2 u_j}{\partial x_i \partial x_i}
\]
Solution Methods

• Methods based on the vorticity equation
• Methods based on artificial compressibility
• Methods that use pressure iterations or pressure correction. (Fractional-Step Methods, SIMPLE, SIMPLER, SIMPLEST, PISO etc.)

Semi Implicit Method for Pressure Linked Equation Revised
What can we use CFD for?

Architectural Applications

• Thermal Comfort
• Airflow in and around buildings
• Convective and radiation heat transfer among building components – effectiveness of radiant heating and cooling systems.
• Natural Ventilation Design
• Displacement Ventilation design
• Urban air pollution and contaminants analysis
DesignBuilder CFD

- Distribution of air temperature, velocity etc
- Internal and External Analyses
- Domain and grid
- Calculations
- Boundary conditions
Internal CFD Tutorial 1

- Create single zone model.
- Add CFD surface boundaries (temp + flow)
- New CFD analysis.
- CFD calculation
- Review the results
Internal CFD Tutorial 2

- Create a radiator component
- Convert the component to an assembly
- Assign CFD boundary conditions to the assembly
- Add assembly to the assembly library.
- Place radiator assembly x 2
- Edit CFD grid.
- Add cell monitor points
- Edit calculation options.
- Comfort calculations.
External CFD Tutorial

Create model for external analysis.

- Create a new CFD analysis
- Perform the CFD calculation
- Review the results