Lecture 4:
Domains, Boundary Conditions and Sources

Introduction to ANSYS CFX
Domains

- Domains are regions of space in which the equations of fluid flow and/or heat transfer are solved.

E.g. a simulation of a copper heating coil in water requires a fluid domain and a solid domain.

- Only mesh components included in a domain are included in the simulation.

E.g. to account for rotational motion, the rotor is placed in a rotating domain.
Domain Creation – Reference Pressure

- Basic Settings: Reference Pressure
  - Represents the absolute pressure datum from which all relative pressures are measured
    \[ P_{\text{abs}} = P_{\text{reference}} + P_{\text{relative}} \]
  - Pressures specified are relative to the Reference Pressure
  - Used to avoid round-off errors which occur when the local pressure differences in a fluid are small compared with the absolute pressure level

Ex. 1: \( P_{\text{reference}} = 0 \text{ Pa} \)
- \( P_{\text{rel,min}} = 99,999 \text{ Pa} \)
- \( P_{\text{rel,max}} = 1 \text{ Pa} \)

Ex. 2: \( P_{\text{reference}} = 100,000 \text{ Pa} \)
- \( P_{\text{rel,min}} = -1 \text{ Pa} \)
- \( P_{\text{rel,max}} = 1 \text{ Pa} \)

Examples of Reference Pressure:
- \( P_{\text{reference}} = 101,325 \text{ Pa} \)
Domain Creation – Buoyancy

- When gravity acts on fluid regions with differences in density, a buoyancy force arises.
- A source term is added to the momentum equations:
  \[ SM_{buoy} = (\rho - \rho_{ref})g \]
  where \( \rho_{ref} \) is the reference density. Fluid with density other than \( \rho_{ref} \) experiences either a positive or negative buoyancy force.
  - The \( (\rho - \rho_{ref}) \) term is evaluated depending on the nature of the fluid:
    - Constant density (Boussinesq model):
      \[ (\rho - \rho_{ref}) = - \rho_{ref} \beta (T - T_{ref}) \]
    - Full Buoyancy Model:
    - Boussinesq Model:
      - Based on:
        - density differences
        - temperature differences
      - For:
        - ideal gases, real fluids, multicomponent fluids
        - constant density fluids
      - Reference Density/Temperature:
        - For single-phase models, usually use an approximate value of the average expected domain value (see next slide)
Domain Creation – Buoyancy

• Buoyancy Reference Density
  – Used to avoid round-off errors
  – Absolute Pressure includes the hydrostatic and reference pressures
    • \( P_{\text{abs}} = P_{\text{reference}} + P_{\text{relative}} + \rho_{\text{ref}} g z \)
  – Careful selection will remove the hydrostatic head from the Pressure solution
    • \( \rho_{\text{ref}} = 0 \text{ [kg m}^3\text{]} \), full hydrostatic variation appears in the Pressure field
    • \( \rho_{\text{ref}} = \) the fluid density (\( \rho \)), no hydrostatic pressure appears in the Pressure field and the only gradients are those driving the flow

For accurate small dynamic pressure changes: use the Reference Pressure to offset the operating pressure & the Buoyancy Reference Density to offset the hydrostatic pressure
• General Options panel: Domain Motion
  – Domain that is rotating about an axis
  – CFX-Solver computes the appropriate Coriolis and centrifugal momentum terms and solves a rotating frame total energy equation

• Mesh Deformation
  – Used for problems involving moving boundaries
  – Mesh motion could be imposed or arise as an implicit part of the solution
Domain Types

- The additional domain tabs/settings depend on the Domain Type selected

Heat Transfer & Turbulence are covered in their own lectures
Domain Type: Solid Models

- Example: Walls of a Heat Exchanger
  - Conjugate Heat Transfer
  - Radiation
  - Solid Motion
    - Used *only* when you need to account for advection of heat in the solid domain
    - Solid motion must be tangential to its surface everywhere
Domain Type: Porous Domains

- Used to model flows where the geometry is too complex to resolve with a grid
- Instead of including the geometric details, their effects are accounted for numerically
Domain Type: Porous Domains

• Volume Porosity: local ratio of fluid volume to total physical volume

• Loss Model
  – Isotropic or Directional Loss
  – Losses are applied using Darcy’s Law
    • Permeability and Loss Coefficients
    • Linear and Quadratic Resistance Coefficients

\[
-\frac{dp}{dx_i} = \frac{\mu}{K_{\text{perm}}} U_i + K_{\text{loss}} \frac{\rho}{2} \left| U \right| U_i \quad \frac{dp}{dx_i} = \frac{C_{R1}}{R_{1i}} U_i + \frac{C_{R2}}{R_{2i}} \left| U \right| U_i
\]

• Coefficients on basis of True Velocity or Superficial Velocity
  
  Superficial Velocity = Volume Porosity * True Velocity
1. Create a name for the fluid to be used
2. Select material to be used in the domain
3. Additional Materials are available by clicking "Materials" in the Outline tree

A Material can be created/edited by right clicking "Materials" in the Outline tree.
**Multicomponent/Multiphase Flow**

ANSYS CFX can model fluid mixtures and multiple phases

**Multicomponent**
- One flow field for the mixture
- Variations in the mixture accounted for by variable mass fractions
- Applicable when components are mixed at the molecular level

**Multiphase**
- Each fluid has its own flow field or all fluids can share a common flow field
- Applicable when fluids are mixed on a macroscopic scale with a discernible interface between the fluids.

Creating multiple fluids will allow you to specify fluid specific and fluid pair models.
Compressible Flow Modeling

- Activated by selecting a Fluid whose density is a function of pressure, e.g. ideal and real gases
- CFX can solve for subsonic, supersonic and transonic flows
- Supersonic/transonic flow
  - Set the heat transfer option to Total Energy

Click to load a real gas library
Defining Boundary Conditions

• To define boundary conditions:
  – Identify the location and type of boundary (e.g. inlet, wall, symmetry)
  – Specify values defining quantities there
    • For mass flow, momentum, energy, etc. into the domain
• Locate boundaries where flow variables have known values or can be adequately approximated
  – Poorly-defined boundary conditions can have a significant impact on your solution
Inlets are for regions where inflow is expected; however, inlets also support outflow when velocity is specified.

- Velocity specified inlets are intended for incompressible flows.
- Pressure and mass flow inlets are suitable for compressible and incompressible flows.
- The same concept applies to outlets.

Domains | Boundary Conditions | Sources
--- | --- | ---
Inlet | Velocity Specified Condition | Pressure or Mass Flow Condition
Inflow allowed | Artificial wall prevents outflow
Openings

- Opening boundaries allow both inflow and outflow
- You have to provide information on conditions, e.g. temperature, turbulence, composition, that apply to fluid flowing into the domain
- Do not use opening as an excuse for a poorly placed boundary
Symmetry

- Reduce computational effort
- No inputs are required
- Flow field as well as geometry must be symmetric:
  - Zero normal velocity at symmetry plane
  - Zero normal gradients of all variables at symmetry plane
  - Must take care to correctly define symmetry boundary locations
Specifying Well Posed Boundary Conditions

Consider a case which contains separate air & fuel supply. Three possible approaches in locating inlet boundaries are:

1. **Upstream of manifold**
   - Can use uniform profiles → natural profiles will develop in pipes
   - Requires more elements

2. **Nozzle inlet plane**
   - Requires accurate velocity profile data for the air and fuel

3. **Nozzle outlet plane**
   - Requires accurate velocity profile data and accurate profile data for the mixture fractions of air and fuel
Specifying Well Posed Boundary Conditions

- If possible, select boundary location and shape such that flow either goes in or out
- Should not observe large gradients in direction normal to boundary
  - Indicates incorrect boundary condition location

Upper pressure boundary modified to ensure that flow always enters domain

This outlet is poorly located. It should be moved further downstream
• Boundaries placed over recirculation zones
  – Poor Location: Apply an opening to allow inflow
  – Better Location: Apply an outlet with an accurate velocity/pressure profile
  – Ideal Location: Apply an outlet downstream of the recirculation zone to allow the flow to develop. This will make it easier to specify accurate flow conditions
Specifying Well Posed Boundary Conditions

• Turbulence at the Inlet
  – Nominal turbulence intensities range from 1% to 5% but depend on specific application.
  – Default turbulence intensity value of 0.037 (that is, 3.7%) is the nominal turbulence through a circular inlet and is a good estimate in the absence of experimental data.
  – Where turbulence is generated by wall friction, consider extending the domain upstream to allow the walls to generate turbulence and the flow to become developed.
Specifying Well Posed Boundary Conditions

- **External Flow**
  - A structure has height H & width W, the domain should be at least 5H high, 10W wide, with at least 2H upstream of the building & 10H downstream.
  - Verify that there are no significant pressure gradients normal to any of the boundaries of the domain. If there are, enlarge the size of domain.

Concentrate mesh in regions of high gradients.
Specifying Well Posed Boundary Conditions

• Symmetry Plane and the Coanda Effect
  – Symmetric geometry does not necessarily mean symmetric flow
  – Example: *The coanda effect*. A jet entering at the center of a symmetrical duct will tend to attach to one side above a certain Reynolds number

---

[Diagram showing the difference between No Symmetry Plane and Symmetry Plane with the Coanda effect and its restrictions.]
### Specifying Well Posed Boundary Conditions

**• When there is 1 Inlet and 1 Outlet**

<table>
<thead>
<tr>
<th>Inlet</th>
<th>Outlet</th>
<th>Result</th>
</tr>
</thead>
<tbody>
<tr>
<td>Most Robust</td>
<td>Velocity/Mass Flow</td>
<td>Static Pressure</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Inlet Total Pressure is an implicit result of the prediction.</td>
</tr>
<tr>
<td>Robust</td>
<td>Total Pressure</td>
<td>Velocity/Mass Flow</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Static Pressure at Outlet and Velocity at Inlet are part of the solution</td>
</tr>
<tr>
<td>Sensitive to Initial</td>
<td>Total Pressure</td>
<td>Static Pressure</td>
</tr>
<tr>
<td>Guess</td>
<td></td>
<td>The system Mass Flow is part of the solution</td>
</tr>
<tr>
<td>Very Unreliable</td>
<td>Static Pressure</td>
<td>Static Pressure</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Not recommended as Inlet Total Pressure level and Mass Flow are implicit result of the prediction</td>
</tr>
</tbody>
</table>
Specifying Well Posed Boundary Conditions

- At least one boundary should specify Pressure (either Total or Static)
- Outlets that vent to the atmosphere typically use a Static Pressure boundary condition
- Inlets that draw flow in from the atmosphere often use a Total Pressure boundary condition (e.g. an open window)
- Mass flow inlets produce a uniform velocity profile over the inlet
- For a mass flow outlet, the mass flow distribution, by default, is based on the upstream profile and the pressure distribution is an implicit part of the solution.
- Pressure specified boundary conditions allow a natural velocity profile to develop
Source Terms
Source Terms

- Sources add additional terms to the transport equations

\[
\frac{\partial \rho h_{tot}}{\partial t} - \frac{\partial p}{\partial t} + \nabla \cdot (\rho U h_{tot}) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot \left( \mu \nabla U + \nabla U^T - \frac{2}{3} \nabla \cdot U \delta U \right) + S_E
\]

- They provide a source (or sink) of the solved variable, e.g.
  - A source term added to the energy transport equation represents a source of heat
  - A source term added to the momentum equations represents added work e.g. a pump
  - Source terms are often used as “black-boxes”
3D, 2D and 1D Sources

- Sources can be applied at a 3D, 2D or 1D location
- 3D sources are applied on a Subdomain, a volume defined over all or part of a domain
- 2D sources are applied at boundaries (Sources tab)
- Source points act on a single mesh element, defined by Cartesian Coordinates