One Dimensional Subchannel Analysis on PWR (Pressurised Water Reactor)

- In this divide the fuel bundles into a no. of subchannels and solves the equations governing two-phase flow in each subchannel.
- It is a method for simulating two phase flow in a fuel bundle.
  Simulating two-phase flow in a fuel bundle is the process of using computer models to predict the behaviour of water and steam (2 phase) in nuclear reactor.
- So, we are using this to predict the risk of accidents, such as fuel rod failures and steam explosions.
- Design fuel bundles that are more resident to accidents.
- Optimise the operating conditions of nuclear power plants.
- It is used to find out coolant temperature and flow rate in a subchannel.
- Predict the performance of fuel bundles.
- So, info can be used to improve the safety and efficiency of nuclear power plant.

Some equations which govern 1D subchannel Analysis:

1. Mass Balance (Law of conservation of Mass) – Nusselt correlation
2. Conservation of Energy
3. Momentum Conservation

---------- personal ----------
Learn about the equations governing two phase flow.
Learn about the cell-centred approach.
Then need to implement the method in computer code.
Then comparing the results of our code with the results of other methods.
If successful then developed a new tool for simulating two phase flow in a fuel bundle.
----------_____----------

CELL CENTERED
Simplified from of Finite volume method approach (1 direction)

- It is a powerful method for solving partial differential equation.
- Example: Law of conservation of mass
- In the context of one method subchannel analysis, the cell centred approach is a method for solving the equation governing two phase flow.
  - The equations are solved at the centre of each cell in the domain.
  - It means that the values of the variables such as velocity, pressure & temperature are represented at the centre of each cell.
  - Approach is simple and efficient.
  - But not accurate as other methods such as finite volume method.

Now can we relate :

- 1 Dimensional subchannel analysis can be related to cell centred approach by dividing the subchannel into a no. of cells.
- The conversation of mass, momentum, energy equations are then solved for each cell. (So more accurate as it accounts for the effects of turbulence and non-uniformity in the coolant flow.
- So, divide the subchannel into no of cells.
- Solve the conservation of mass, momentum, energy equations of each cell.
- Use the solution from each cell to calculate overall behaviour of the subchannel.

One Dimension Subchannel Analysis:

1. Mass conservation:

- Rate of mass entering the control volume = Rate of mass leaving the control volume.
- It is used to calculate the void fraction of the gas phase in control volume.

Conservation of mass:

$$\frac{dm_i}{dz} = -\sum_{j=1}^{N_j}(w_{ij})$$

(file:///C:/Users/HP/Downloads/Paper.pdf)

----- Not valid for unsteady case ----write transient conservation equation

- Steady state eq is right

- To derive the conservation of mass or continuity equation, consider a control volume within the boundary layer, as shown in Fig. 4.5 , and assume that steady-state conditions prevail. There are no gradients in the z direction (perpendicular to the plane of the sketch), and the fluid is incompressible. Then the rates of mass flow into and out of the control volume, respectively, in the x direction are:

$$\rho u\, dy \quad \text{and} \quad \rho\left(u + \frac{\partial u}{\partial x}dx\right)dy$$

Thus, the net mass flow into the element in the x direction is

$$-\rho\frac{\partial u}{\partial x}dx\,dy$$

Similarly, the net mass flow into the control volume in the y direction is

$$-\rho\frac{\partial v}{\partial y}dx\,dy$$

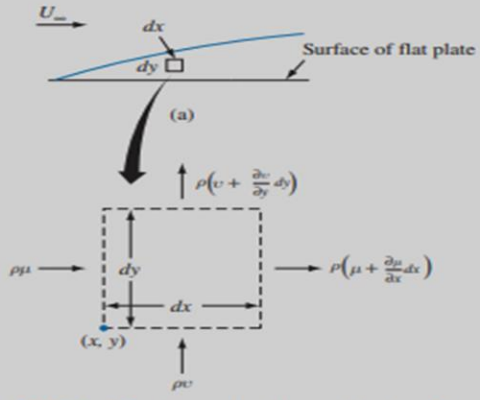


FIGURE 4.5 Control volume ($dx\,dy \cdot 1$) for conservation of mass in an incompressible boundary layer in flow over a flat plate.

Page 258

(http://160592857366.free.fr/joe/ebooks/Mechanical%20Engineering%20Books%20Collection/HEAT%20TRANSFER/Ptinciples%20of%20Heat%20Transfer.pdf)

- IN own words represent it .

"Nusselt Correlation: " -

Nusselt Correlation relates the heat transfer coefficient to the Reynolds number and Prandtl number:
It is used to characterize the heat transfer between a fluid and solid surface:

For turbulent flow heat transfer with $Re_D \geq 10^4$, the Nusselt number correlation developed by Manglik and Bergles [60] is expressed as

$$\overline{Nu}_D = 0.023 Re_D^{0.8} Pr^{0.4} \left[1 + \frac{0.769}{y}\right] \left[\frac{\pi + 2 - (2\delta/D)}{\pi - (4\delta/D)}\right]^{0.2} \times \left[\frac{\pi}{\pi - (4\delta/D)}\right]^{0.8} \phi \qquad (6.84)$$

Page: 421
(http://160592857366.free.fr/joe/ebooks/Mechanical%20Engineering%20Books%20Collection/HEAT%20TRANSFER/Ptinciples%20of%20Heat%20Transfer.pdf)

2. Conservation of Energy:

$$\frac{dh_i}{dz} = \frac{1}{m_i}[q_i P_{he} + \sum_{j=1}^{N_j} (W_{ij}(h_j - h_i) - k_{ij}(T_i - T_j) + w_{ij}(h_i - h^*))]$$

Page 5 (Paper harish sir.pdf)
--- Unsteady case in two fluid model (work on it)

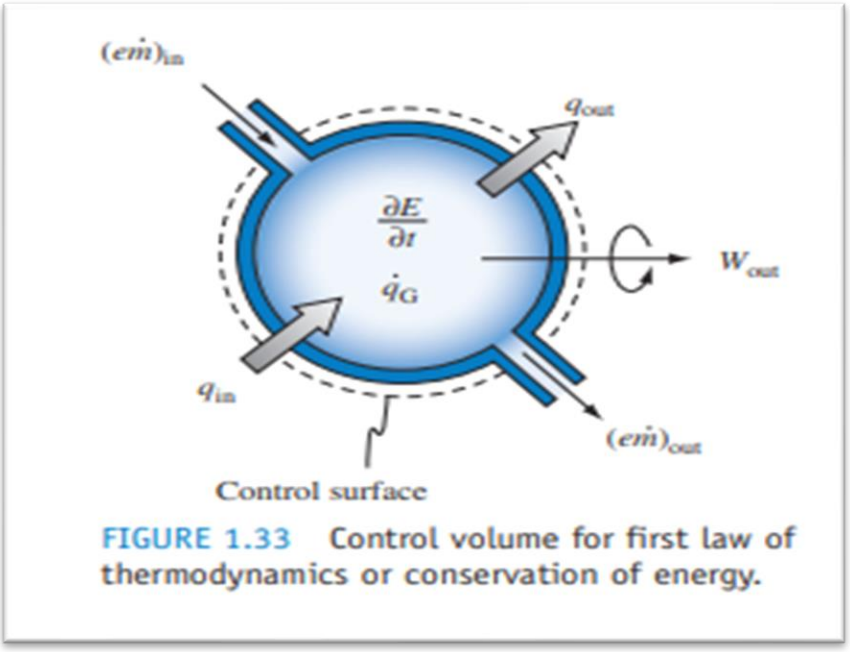


FIGURE 1.33 Control volume for first law of thermodynamics or conservation of energy.

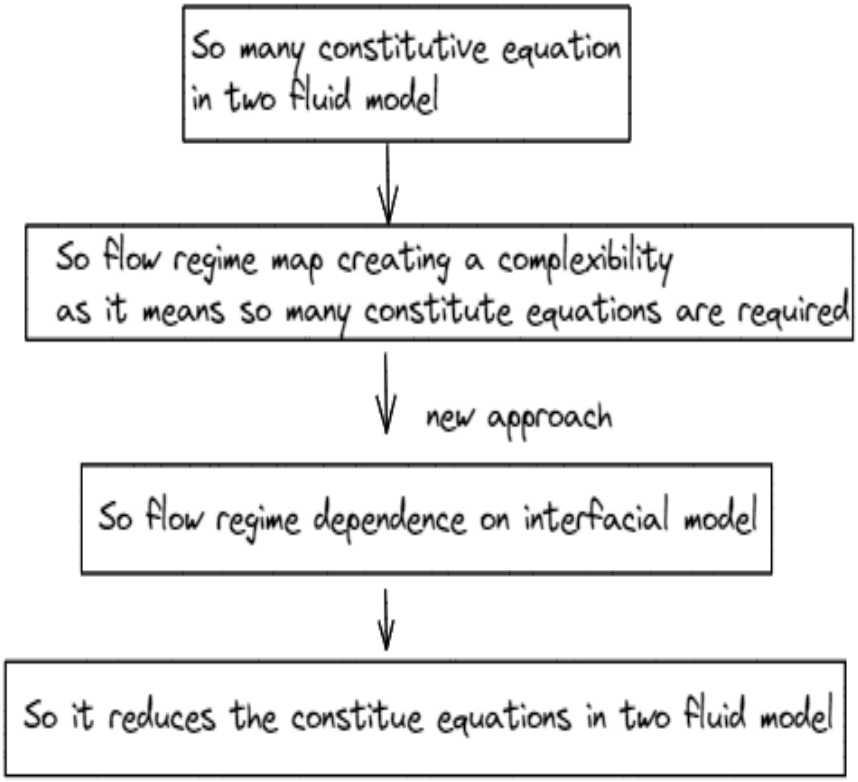
Observe that the inflow and outflow rate terms are surface phenomena and are therefore proportional to the surface area. The internal energy generation term is encountered when another form of energy (such as chemical, electrical, or nuclear energy) is converted to thermal energy within the control volume. The generation term is therefore a volumetric phenomenon, and its rate is proportional to the volume within the control

surface. Energy storage is also a volumetric phenomenon associated with the internal energy of the mass in the control volume, but the process of energy generation is quite different from that of energy storage, even though both contribute to the rate of energy storage.

Physical model understanding:

* It is a macroscopic model
* Treats two phases as - interpenetrating fluids.
* Based on conservation of Mass/momentum/energy.
* There are so many equations coupled through interfacial transform.

So it predicts the behaviour of two phase flows.



Physical Model:

- The two fluid model is a macroscopic model of two-phase flow that treats the two phases as interpenetrating fluids.
- The model is based on conservation of mass, momentum and energy.
- It accounts for the interaction between the two phases through interfacial transfer terms.
- Interfacial transfer terms account for the exchange of mass, momentum and energy between two phases.
- So, it predicts the behaviour of two-phase flows in a variety of applications including nuclear reactors, chemical processes and power plants.

- Here, there are validated constitute relations:
  - Validated constitute relations are the equations that describes the interaction between the two phases.
  - These equations based on experimental data.
- Flow regime map is a graphical representation which describes the complexibility of two phase model based on their physical properties. It can be used to predict the pressure drop, heat transfer and other properties of the system.
  - So, it proves that we need large number of constitute relations because each flow regime has its own set of microscopic properties.
- So, to reduce the complexibility, here discuss a simpler approach to the flow regime that is dependence of the interfacial model. (Interfacial model describes the transfer of mass, momentum and energy between the phases of a two-phase system.
  - So, this approach reduces the number of required constitute equations by using a single set of equations for all flow regimes.
  - Interfacial exchange models incorporated with THERMIT-2 two phase model.

Equations of Physical model:

Here we are converting all these equations in one dimensional so alpha=0;

a. Conservation of Liquid Mass:

$$\frac{\partial}{\partial t}[(1-\alpha)\rho_l] + \nabla.\left[(1-\alpha)\rho_l\vec{V_l}\right] = \Gamma_l - W_{tl}$$

As it is one Dimensional so put alpha = 0, we get

$$\frac{\partial}{\partial t}(\rho_l) + \nabla.\left[\rho_l\vec{V_l}\right] = 0$$

b. Conservation of Liquid Energy:

$$\frac{\partial}{\partial t}[(1-\alpha)\rho_l e_l] + \nabla.\left[(1-\alpha)\rho_l e_l\vec{V_l}\right] + P\nabla.\left[(1-\alpha)\vec{V_l}\right] - P\frac{\partial\alpha}{\partial t} = Q_{wl} + Q_{il} - Q_{tl}$$

As it is one Dimensional so put alpha = 0, we get

$$\frac{\partial}{\partial t}[\rho_l e_l] + \nabla.\left[\rho_l e_l\vec{V_l}\right] = Q_{wl}$$

c. Conservation of Liquid Momentum:

$$(1-\alpha)\rho_l\frac{\partial\vec{V_l}}{\partial t} + (1-\alpha)\rho_l\vec{V_l}.\nabla\vec{V_l} + (1-\alpha)\nabla P = -\vec{F}_{wl} - \vec{F}_{il} + (1-\alpha)\rho_l\vec{g} - \vec{F}_{tl}$$

As it is one Dimensional so put alpha = 0, we get

$$\rho_l\frac{\partial\vec{V_l}}{\partial t} + \rho_l\vec{V_l}.\nabla\vec{V_l} + \nabla P = -\vec{F}_{wl} + \rho_l\vec{g}$$

SIMPLE (Semi-Implicit and Method for Pressure Linked Equations) ALGORITHM IN CFD (Computational Fluid Dynamics)

Semi-Implicit – explicit + implicit

- Explicit (convective terms) – efficient ---------- unknown quantity
- Implicit (diffusive terms) – stable ---------- known quantity

So semi-implicit method is both efficient and stable. (Left side explicit and right)

The fundamental governing equations of fluid mechanics are based on three laws of conservation, referred to as the
- law of **conservation of mass**,
- the **law of conservation of momentum** and
- the **law of conservation of energy**.
- The **continuity equation** represents the **law of conservation of mass**,
- The **Navier-Stokes equations** represent **the law of conservation of momentum**, and
- the **energy equation** represents **the law of conservation of energy**.

- In SIMPLE, the continuity and Navier-Stokes equations are required to be discretized and solved in a semi-implicit way.

- Discretized means the continuity and Navier-Stokes equations are converted from **continuous equations to discrete equations.**

- Continuous equations are difficult to solve directly, but discrete equations can be solved using a variety of numerical methods.

  - In the first step, the **continuity equation is solved for the velocity field.** This is done using a **pressure-implicit method**, which means that **the pressure is treated as a known quantity**.

  - The velocity field is then used to solve the Navier-Stokes equations for the pressure field. This is done using an explicit method, which means that the pressure is treated as an unknown quantity.
  - That's why method is known as semi-implicit.

  - The **continuity equation is solved implicitly**, while the **Navier-Stokes equations are solved explicitly**.
  - This means that the continuity equation is solved using a method that takes into **account the effects of the pressure field**,
  - Navier-Stokes equations are solved using a method that **doesn't consider the effects of the pressure field**.

Equations:

- The continuity equation is based on **conservation of mass**

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

As it is one dimensional so on x axis the equation is:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} = 0$$

where ρ **is fluid density**, u **is fluid velocity** in the **x-direction**, v **is fluid velocity** in the **y direction**, w **is fluid velocity** in **the z-direction** and t is time.

- The **Navier-Stokes** equations based on **conservation of momentum for the x-direction**

- $$\frac{\partial(\rho u)}{\partial t} + \nabla.\left(\rho u \vec{U}\right) = -\frac{\partial(\rho)}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x$$

As it is only in one dimensional .

- $$\frac{\partial(\rho u)}{\partial t} + \nabla.\left(\rho u \vec{U}\right) = -\frac{\partial(\rho)}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \rho f_x$$

pf x it is a external force in x direction

The discretization of the Navier-Stokes equations in one dimension using the semi-implicit method for the pressure-linked equations.

- **We use Finite Difference Method for discretizing the Navier Stokes equations**
- Backward difference and central difference are examples of finite difference methods for discretizing equations, particularly partial differential equations (PDEs).
- We assume that they are incompressible Navier-Stokes equations.

a. Continuity Equation:

The continuity equation in one dimension is:

- The continuity equation is based on **conservation of mass**
- The law of conservation of mass states that mass cannot be created or destroyed.
- This means that the total mass of a fluid must remain constant over time.
- The continuity equation is a fundamental equation in fluid dynamics.

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho v)}{\partial x} = 0$$

ρ is the density of the fluid
v is the velocity of the fluid
t is the time
x is the spatial coordinate

- $\frac{\partial \rho}{\partial t}$ represents **the rate of change of density with respect to time**.
- $\frac{\partial (\rho v)}{\partial x}$ represents the rate of change of the product of density and velocity with respect to space.

- Assuming a uniform grid with grid spacing Δx and time step Δt, we can write:

- Now the **discretised** form of the **continuity equation** is:

$$\left[\left(\frac{\rho_i^{n+1} - \rho_i^n}{\Delta t} \right) + \left(\frac{\rho u_{i+1}^n - \rho u_{i-1}^n}{\Delta X} \right) \right] = 0$$

Change in density over time **Change in density due to advection**

ρ_i^{n+1}: represents the density at position i and time step n+1.
ρ_i^n: represents the density at position i and time step n.
Δt: represents the time step size, which is the time interval between n and n+1.
It indicates the discretization of time, or how much time passes between consecutive time steps.
ρu_{i+1}^n: represents the product of density and velocity at position i+1 and time step n.
It denotes the density multiplied by the velocity at the neighboring position (i+1) and the current time step.

$\rho\, u_{i-1}^{n}$: represents the product of density and velocity at position i-1 and time step n.

ΔX: This term represents the spatial step size, which is the distance between adjacent positions in the discrete grid. It indicates the discretization of space, or how much distance separates adjacent grid points.

Git hub code for continuity equation

b. Momentum Equation:

- It represents a form of the Navier-Stokes equation, which is a fundamental equation governing the motion of fluid.
- This specific equation is a simplified form that describes the **conservation of momentum in the x-direction for an incompressible fluid.**

The momentum equation in one dimension is:

$$\rho \left[\frac{\partial(u)}{\partial t} + \frac{u\partial(u)}{\partial x} \right] = - \frac{\partial(P)}{\partial x} + \frac{\mu \partial^2 u}{\partial x^2} + \rho g$$

ρ=density of the fluid
P= Pressure
u=velocity component along x axis,
μ= dynamic viscosity of fluid
t= time

$\frac{\partial(u)}{\partial t}$ **It represents unsteady acceleration term**

$\frac{u\partial(u)}{\partial x}$ **It represents convective acceleration term**

$\frac{\partial(p)}{\partial x}$ **It represents pressure Gradient term**

$\frac{\mu \partial^2 u}{\partial x^2}$ **It represents viscous Diffusive term**

ρg **It represents gravitational force**

Discrete equations

- The discrete equations for fluid dynamics are a set of equations that are used to approximate **the behaviour of fluids using numerical methods**.

- The equations are based on the fundamental laws of physics that **govern fluid flow**, such as the **conservation of mass, momentum, and energy**.

- Discretization involves approximating the continuous variables (such as velocity, pressure, or temperature) at each point in **space and time** using discrete values.

- **The discretized equation of Navier stokes equation of momentum is:**
  - By solving this discretized equation numerically, we can simulate and analyse the behaviour of fluid flow in one dimension in the x-direction.
  - Here we assume that the gravitational force is null.
  - Superscript – temporal (time)
  - Subscript – space

Forward in time and central in space

- $$\rho\left[\left(\frac{u_i^{n+1}-u_i^n}{\Delta t}\right)+u_i^n\left(\frac{u_{i+1}^n-u_{i-1}^n}{2\Delta X}\right)\right]=-\left(\frac{P_{i+1}^n-P_{i-1}^n}{2\Delta X}\right)+\mu\left[\frac{u_{i+1}^n-2u_i^n+u_{i-1}^n}{\Delta X^2}\right]$$

Advection term **Convective term** **Pressure term** **Viscous/Diffusive term**

- where u^n represents the velocity at time step n, and
- P^n represents the pressure at time step n

Git hub link for momentum equation

c. Energy Equation:

- It represents a form of the Navier-Stokes equation, which is a fundamental equation governing the motion of fluid.
- The rate of change of energy in a system is equal to the sum of the work done by the pressure, the heat flux and any other sources of energy.

The Energy equation in one dimension is:

$$\rho c_p \left[\frac{\partial(T)}{\partial t} + \frac{u\partial(T)}{\partial x}\right] = -P\frac{\partial(u)}{\partial x} + \frac{\partial}{\partial x}\left(k * \frac{\partial(T)}{\partial x}\right) + q$$

1 2 3 4 5

1. **Rate of change of E**
2. **Rate of change of E due to convection**
3. **work done by pressure**
4. **heat flux**
5. **Any other heat source**

- The first term $\rho c_p \frac{\partial(T)}{\partial t}$ represents the time rate of change of the internal energy of the material.
- The second term $\rho c_p \frac{u\partial(T)}{\partial x}$ represents the convective heat transfer, where **u** is the velocity of the fluid and $\frac{\partial(T)}{\partial x}$ represents the spatial temperature gradient.
- The third term $-P\frac{\partial(u)}{\partial x}$ represents the **work done by the pressure gradient** on the fluid.
- The fourth term $\frac{\partial}{\partial x}\left(k * \frac{\partial(T)}{\partial x}\right)$ represents **the heat transfer due to conduction**, **where k is the thermal conductivity of the material.**
- The last **term q represents any additional heat sourc**e or sink in the system.

- **ρ: Density of the material**
- **Cp: Specific heat capacity of the material**
- **T: Temperature of the material**
- **t: Time**
- **u: Velocity of the fluid (in the x-direction)**
- **x: Spatial coordinate (in the x-direction)**
- **P: Pressure gradient in the x-direction**
- **k: Thermal conductivity of the material**
- **q: Heat source term**

Discrete equation of Energy

- The discrete equations for fluid dynamics are a set of equations that are used to approximate **the behaviour of fluids using numerical methods**.

- The equations are based on the fundamental laws of physics that **govern fluid flow**, such as the **conservation of mass, momentum, and energy**.

- Discretization involves approximating the continuous variables (such as velocity, pressure, or temperature) at each point in **space and time** using discrete values.

- **The discretized equation of Navier stokes equation of Energy is:**
  - By solving this discretized equation numerically, we can simulate and analyse the behaviour of fluid flow in one dimension in the x-direction.
  - Here we assume that the work done by pressure and external force is null.
  - Superscript – temporal (time)
  - Subscript – space

- $$\rho_i^{\eta+1} c_p \left[\left(\frac{T_i^{n+1} - T_i^n}{\Delta t} \right) + u_i^n \left(\frac{T_i^n - T_i^{n-1}}{\Delta X} \right) \right] = K * \left(\frac{T_{i+1}^n - 2T_i^n + T_{i-1}^n}{\Delta X^2} \right)$$

-

ρ_i : Density of the material at grid point i
η : Coefficient for time stepping (such as SIMPLE method)
c_p : Specific heat capacity of the material
T_i^n : Temperature at grid point i and time step n
Δt : Time step size
u_i^n : Velocity of the fluid at grid point i and time step n (in the x-direction)
ΔX : Grid spacing in the x-direction
K : Thermal conductivity of the material
$T_{i+1}^n, T_i^n, T_{i-1}^n$: Temperatures at neighboring grid points i+1, i, and i-1 at time step n

Git hub link for energy equation