

Special Topics in Computational Fluid Dynamics (CFD)

Introduction – DAY 1

Kumaresh

Contents

- Introduction
- CFD fundamentals
- Mathematical operations
- Basic Governing Equations
- Installations
- Exercise – 1

Introduction – About this course

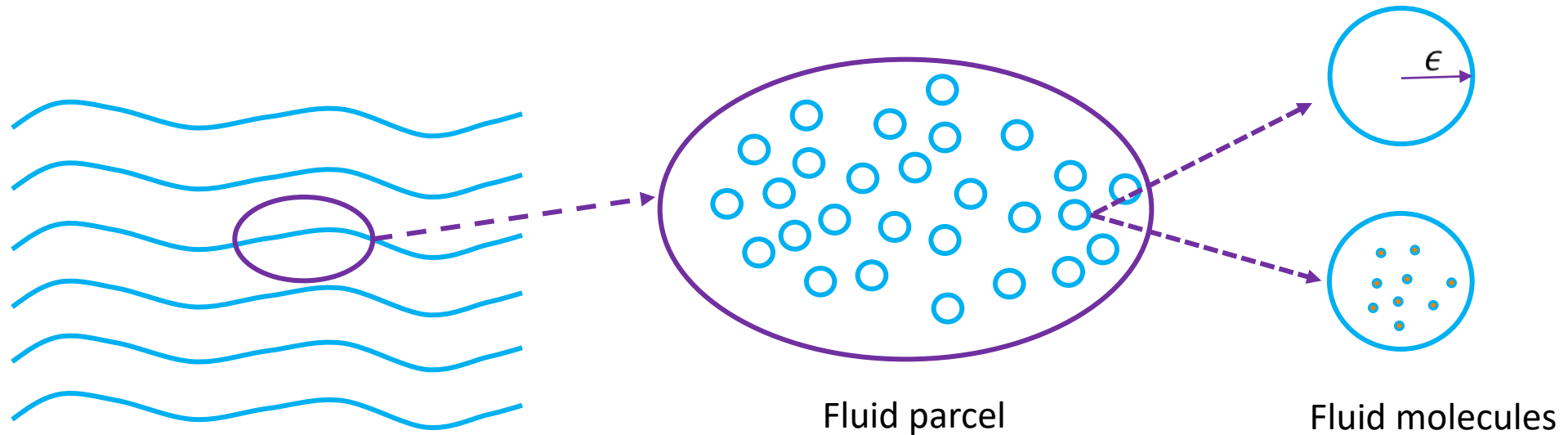
- Course duration per session: 3hrs
- Requirements:
 - Virtual box and installing OS & softwares.
 - Interest to learn CFD using OpenFOAM & Octave
 - Interest to ask questions in discussion forum (GitHub)
 - Work as a team
- Evaluation:
 - Exercises (OpenFOAM and ANSYS Fluent to some extent) (40%)
 - Project Reports (30%)
 - 1 seminar (10%)
 - Attendance (20%)

References

- Ferziger and Peric; Computational Methods for Fluid Dynamics.
- S. Patankar; Numerical Heat Transfer and Fluid Flow.
- Tannehill et al.; Computational Fluid Mechanics and Heat Transfer.
- Versteeg, Malalasekera; An Introduction to Computational Fluid Dynamics.
- C.J. Greenshields, H.G. Weller; Notes on CFD: General Principles (OpenFOAM)

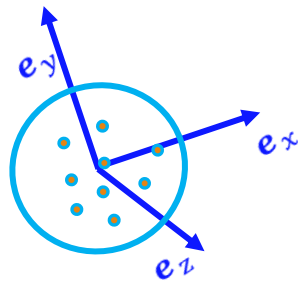
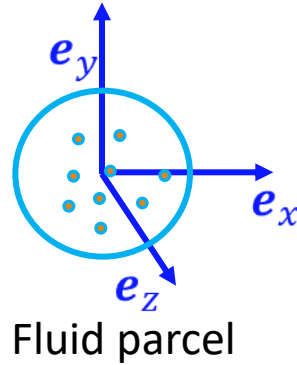
CFD fundamentals – Fluid

- A substance whose molecular structure offers no resistance to external forces - Ferziger, Peric

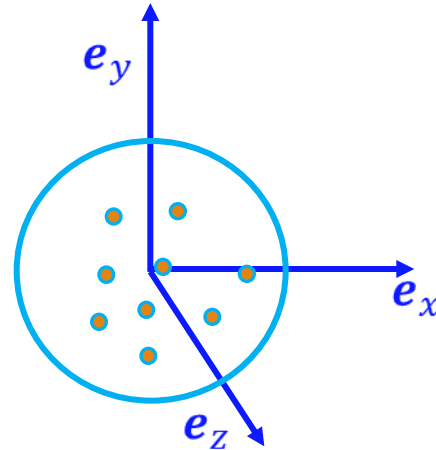


CFD fundamentals – Fluid

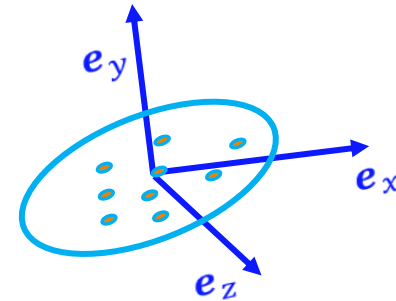
- A substance whose molecular structure offers no resistance to external forces - Ferziger, Peric



Rotation



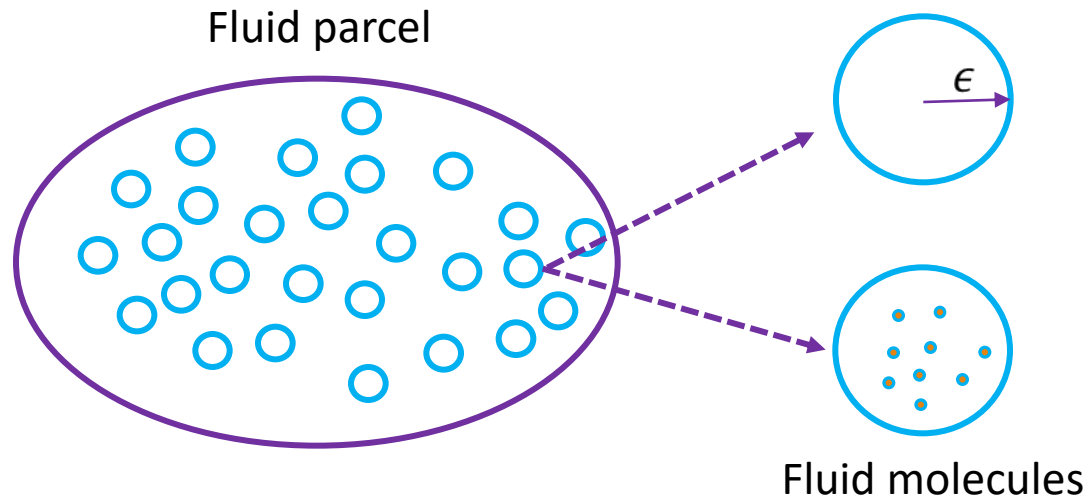
Expansion



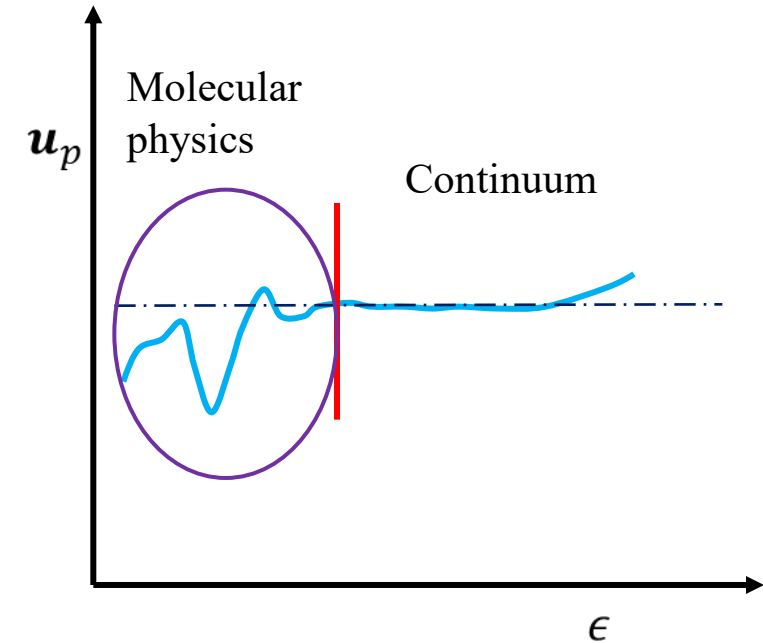
Deformation

CFD fundamentals – Fluid

- A substance whose molecular structure offers no resistance to external forces - Ferziger, Peric



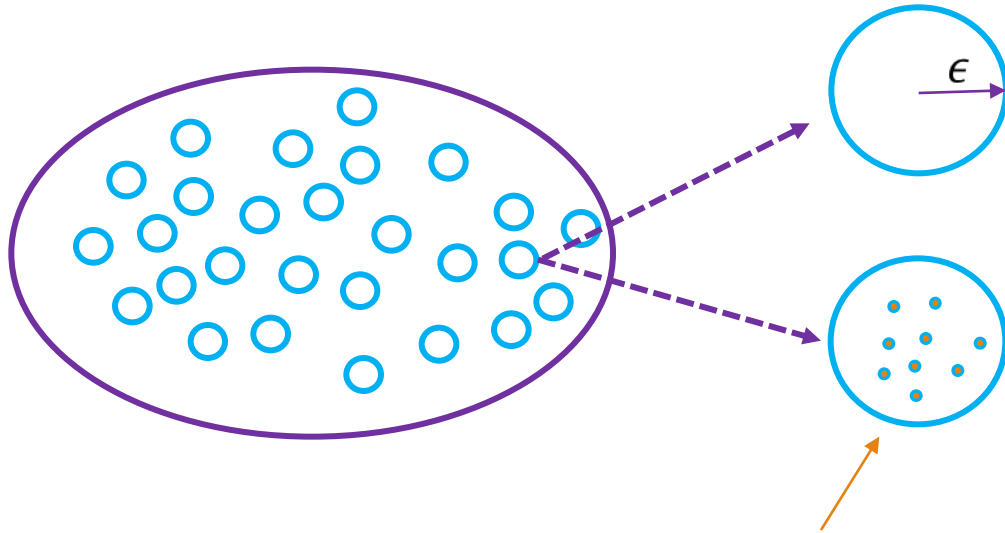
$$\mathbf{u}_p = \frac{\sum_{i=1}^{N_{mol}} \mathbf{u}_{mol}}{N_{mol}}$$



Fluid velocity: $\mathbf{u}(\mathbf{x}, t)$

CFD fundamentals – Continuum

Knudsen number: $Kn = \frac{\lambda}{L} = \frac{\text{molecular mean free path length}}{\text{physical length}}$



In physics, mean free path is the average distance over which a moving particle

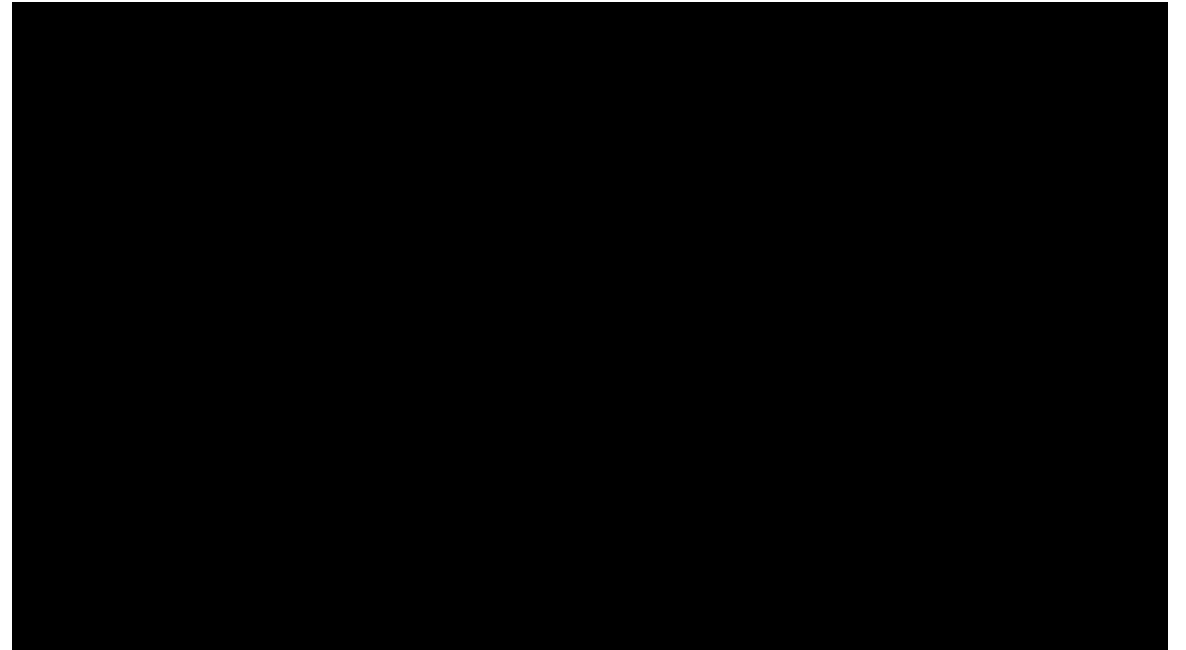
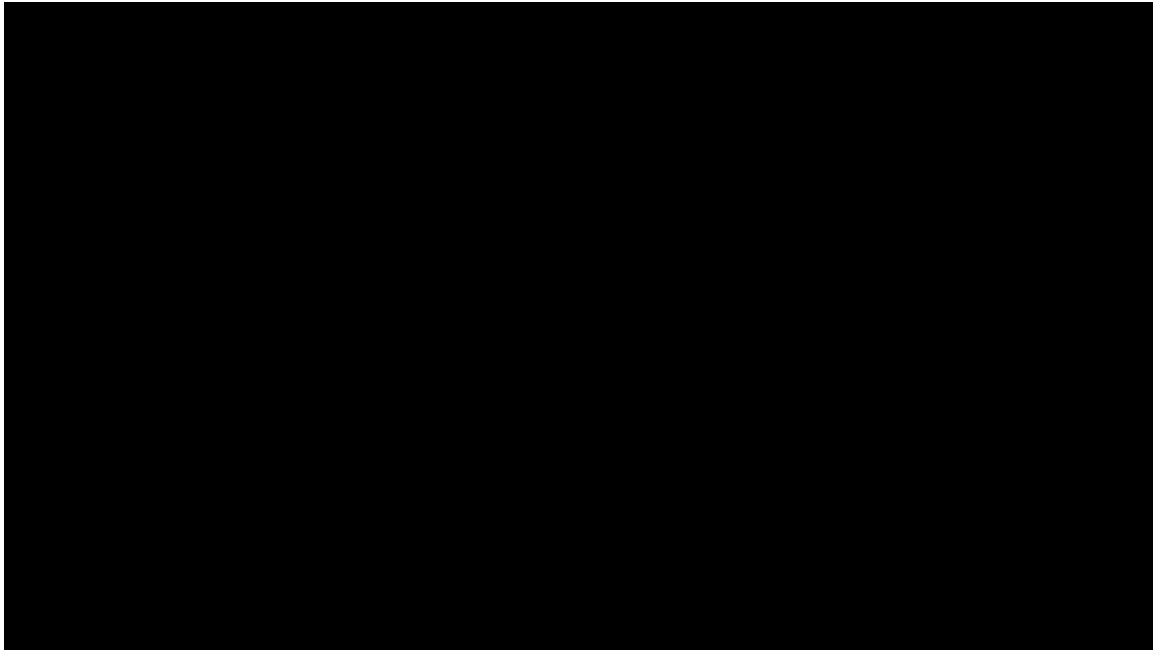
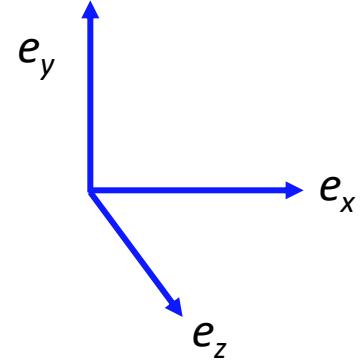
$Kn < 0.01$	Continuum flow
$0.01 < Kn < 0.1$	Slip flow
$0.1 < Kn < 10$	Transitional flow
$Kn > 10$	Free molecular flow

$$\frac{\partial}{\partial t}(\rho u) + \nabla \cdot (\rho u u) = \nabla \cdot (\mu \nabla u) - \nabla p + S_u$$

Mathematical operations

Gradient:

$$\nabla \rho = \left(\frac{\partial}{\partial x} \mathbf{e}_x + \frac{\partial}{\partial y} \mathbf{e}_y + \frac{\partial}{\partial z} \mathbf{e}_z \right) \rho = \left(\frac{\partial \rho}{\partial x} \mathbf{e}_x + \frac{\partial \rho}{\partial y} \mathbf{e}_y + \frac{\partial \rho}{\partial z} \mathbf{e}_z \right)$$



$$\frac{\partial}{\partial t}(\rho u) + \nabla \cdot (\rho u u) = \nabla \cdot (\mu \nabla u) - \nabla p + S_u$$

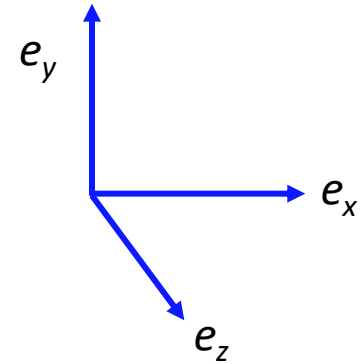
Mathematical operations

Gradient:

$$\nabla \rho = \left(\frac{\partial}{\partial x} \mathbf{e}_x + \frac{\partial}{\partial y} \mathbf{e}_y + \frac{\partial}{\partial z} \mathbf{e}_z \right) \rho = \left(\frac{\partial \rho}{\partial x} \mathbf{e}_x + \frac{\partial \rho}{\partial y} \mathbf{e}_y + \frac{\partial \rho}{\partial z} \mathbf{e}_z \right)$$

$$\nabla \mathbf{u} = \begin{bmatrix} \partial u / \partial x & \partial v / \partial x & \partial w / \partial x \\ \partial u / \partial y & \partial v / \partial y & \partial w / \partial y \\ \partial u / \partial z & \partial v / \partial z & \partial w / \partial z \end{bmatrix}$$

$$\frac{\partial \rho}{\partial x} = \frac{d\rho}{dx} (\text{in 1D})$$



$$\frac{\partial}{\partial t}(\rho u) + \nabla \cdot (\rho u u) = \nabla \cdot (\mu \nabla u) - \nabla p + S_u$$

Mathematical operations

Divergence:

- In vector calculus, divergence is a vector operator that operates on a vector field, producing a scalar field giving the **quantity of the vector field's source at each point**.
- More technically, the divergence represents the volume density of the outward flux of a vector field from an infinitesimal volume around a given point.
- As an example, consider air as it is heated or cooled. **The velocity of the air at each point defines a vector field**. While air is heated in a region, it expands in all directions, and thus the velocity field points outward from that region. **The divergence of the velocity field in that region would thus have a positive value**. While the air is cooled and thus contracting, the divergence of the velocity has a negative value.

$$\nabla \cdot \mathbf{u} = \left(\frac{\partial}{\partial x} \mathbf{e}_x + \frac{\partial}{\partial y} \mathbf{e}_y + \frac{\partial}{\partial z} \mathbf{e}_z \right) (u \mathbf{e}_x + v \mathbf{e}_y + w \mathbf{e}_z) = \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right)$$

Governing Equations

- The general equation can be written in the form as:

$$\frac{\partial}{\partial t}(\rho\phi) + \nabla \cdot (\rho u \phi) = \nabla \cdot (\Gamma \nabla \phi) + S$$

$$\frac{\partial}{\partial t}(\rho\phi) + \frac{\partial}{\partial x_j}(\rho u_j \phi) = \frac{\partial}{\partial x_j} \left(\Gamma \frac{\partial \phi}{\partial x_j} \right) + S$$

$$\frac{\partial}{\partial t} \iiint \rho \phi dV + \iint \rho \phi (u \cdot dA) = \iint \Gamma \nabla \phi \cdot dA + \iiint S dV$$

1

2

3

4

1 → Unsteady/Transient term

3 → Diffusion term

2 → Advection/Convection term

4 → Source term

Governing Equations

$$\frac{\partial}{\partial t}(\rho\phi) + \nabla \cdot (\rho u \phi) = \nabla \cdot (\Gamma \nabla \phi) + S$$

Continuity equation: $\phi = 1, S = 0 \rightarrow \frac{\partial}{\partial t}(\rho) + \nabla \cdot (\rho u) = 0$

Momentum equation: $\phi = u, \Gamma = \mu, S = S_u \rightarrow \frac{\partial}{\partial t}(\rho u) + \nabla \cdot (\rho u u) = \nabla \cdot (\mu \nabla u) - \nabla p + S_u$

Energy equation: $\phi = h, \Gamma = k / C_p, S = S_h \rightarrow \frac{\partial}{\partial t}(\rho h) + \nabla \cdot (\rho u h) = \nabla \cdot \left(\frac{k}{C_p} \nabla h \right) + S_h$

Species equation: $\phi = h, \Gamma = \Gamma_l, S = S_m \rightarrow \frac{\partial}{\partial t}(\rho m_l) + \nabla \cdot (\rho u m_l) = \nabla \cdot (\Gamma_l \nabla m_l) + S_m$

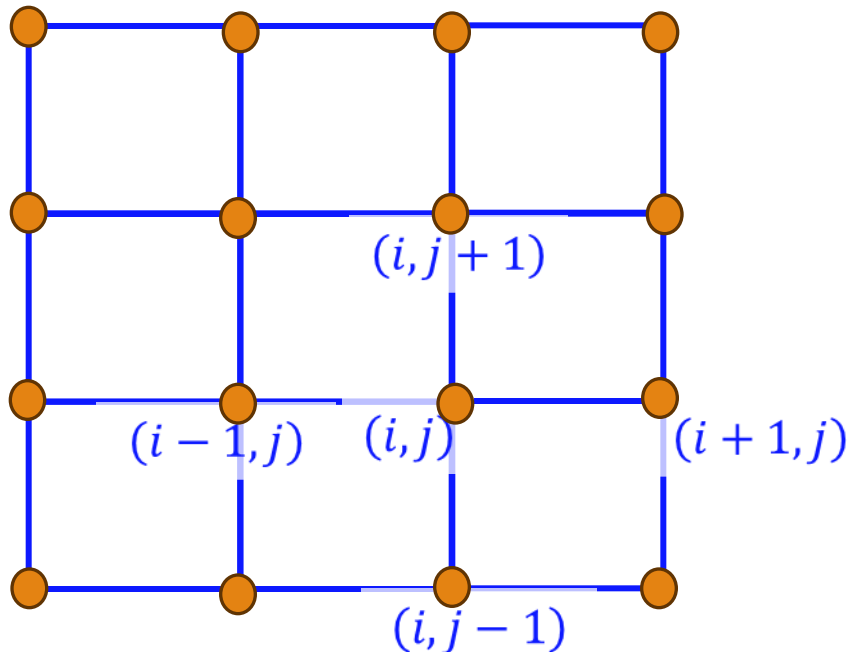
Turbulence equation: $\phi = k(\text{or})\varepsilon, \Gamma = \Gamma_k(\text{or})\Gamma_\varepsilon, S = S_k(\text{or})S_\varepsilon \rightarrow \frac{\partial}{\partial t}(\rho k) + \nabla \cdot (\rho u k) = \nabla \cdot (\Gamma_k \nabla k) + S_k$

Ideal Gas equation: $p = \rho R T$

Finite Difference – Finite Volume

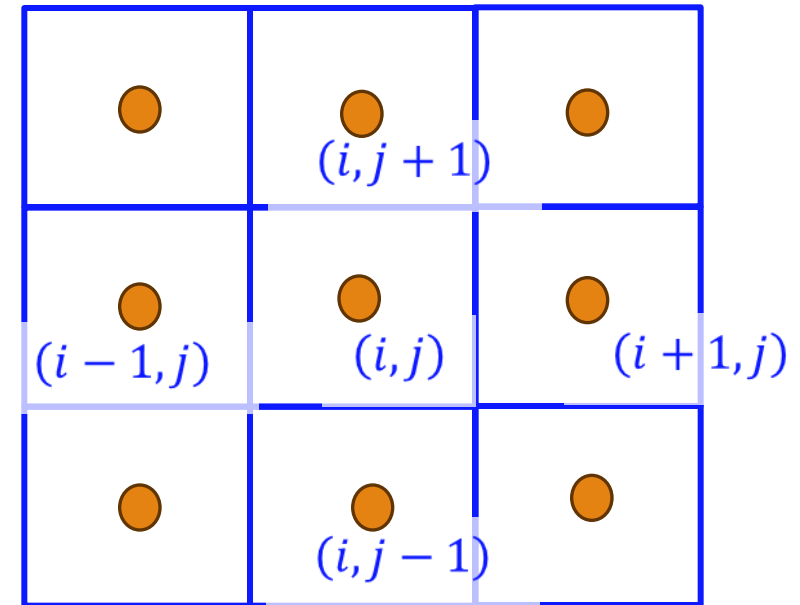
Differential form

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$



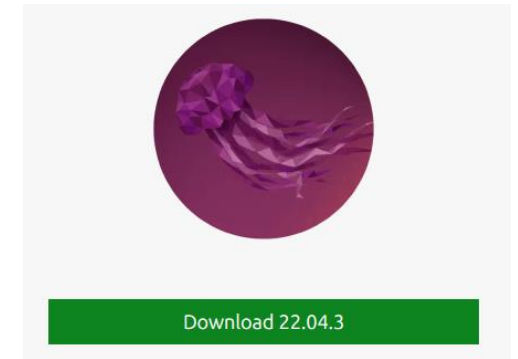
Integral form

$$\frac{\partial}{\partial t} \int_V \rho dV + \oint_S \rho \mathbf{u} \cdot d\mathbf{S} = 0$$



Installations

- Preconfiguration packages:
 - <https://onedrive.live.com/?redeem=aHR0cHM6Ly8xZHJ2Lm1zL2YvcyFBcVQyWUVCOTctMVJnUDhNdHNNUHFvT0dzcTRkZGc%5FZT1Jb2NYdjA&id=51EDEF7D4060F6A4%2116268&cid=51EDEF7D4060F6A4>
- List
 - Virtual Box [to create virtual machines]
 - Operating System: Ubuntu 24.04.1 [Install OpenFOAMv2412 & Octave]
 - AnyDesk [For remote access]
- Create a github account:
 - <https://github.com/Kumaresh0402/SpecialTopicsinComputationalFluidDynamics>
 - Discussion forum:
 - <https://github.com/Kumaresh0402/SpecialTopicsinComputationalFluidDynamics/discussions>



Open▽FOAM



ANSYS Student version installation

[Home](#) > [Students & Academic](#) > [Students](#) > [Ansys Student Software: Ansys Student](#)

Ansys Student - Free Software Download

Ansys Student offers free access to our Ansys Workbench-based bundle. This bundle includes Ansys Mechanical, Ansys CFD, Ansys Discovery, Ansys SPEOS, Ansys Autodyn, Ansys DesignXplorer, and Ansys SpaceClaim. Used by students across the globe, Ansys Student can be leveraged to enhance your skill set with some of our most-used products.

From 2023R1 on, Ansys Discovery is now included in Ansys Student. Ansys Discovery offers free access to our simulation-driven design tool that combines instant physics simulation, proven Ansys high-fidelity simulation and interactive geometry modelling in a single user experience. This is the perfect product for a student just getting started with simulation—it lets students learn about physics without the time investment required to learn how to use a complex simulation tool.

Terms of Use: Free student downloads are for educational use only and may only be used for self-learning, student instruction, student projects, and student demonstrations.

[DOWNLOAD ANSYS STUDENT 2024 R1 ▸](#)

QUICK LINKS

[Learning Forum](#) ▸



[Innovation Courses](#) ▸

[Learning Resources](#) ▸


[Student Teams](#) ▸

Presentation and Records


<https://github.com/Kumaresh0402/SpecialTopicsinComputationalFluidDynamics>



 Kumaresh0402 / SpecialTopicsinComputationalFluidDynamics


[Code](#) [Issues](#) [Pull requests](#) [Discussions](#) [Actions](#) [Projects](#) [Wiki](#) [Security](#) [Insights](#) [Settings](#)

 **SpecialTopicsinComputationalFluidDynamics** Public Pin Watch 0

main 1 Branch 0 Tags Add file Code

 **Kumaresh0402** Create README.md a7e58f1 · 4 minutes ago 4 Commits

 DAY1	Create Readme.md	7 minutes ago
 README.md	Create README.md	4 minutes ago

 **README** Edit More

SpecialTopicsinComputationalFluidDynamics


The purpose of this repository is to keep track of lecture notes.

Discussion forum:


<https://github.com/Kumaresh0402/SpecialTopicsinComputationalFluidDynamics/discussions>


Discussions Forum

<https://github.com/Kumaresh0402/SpecialTopicsinComputationalFluidDynamics/discussions>










Welcome to SpecialTopicsinComputationalFluidDynamics Discussions!




 Announcements · Kumaresh0402




 Sort by: Latest activity ▾ Label ▾ Filter: Open ▾ New discussion

Categories

-  View all discussions
-  Announcements
-  General
-  Ideas
-  Polls
-  Q&A
-  Show and tell

Discussions

 1  [Exercise-1] Package installations (ANSYS and OpenFOAM)
Kumaresh0402 started 4 minutes ago in General  0

 1  Welcome to SpecialTopicsinComputationalFluidDynamics Discussions!
Kumaresh0402 announced 5 minutes ago in Announcements  0

Introduce yourself here please !

<https://github.com/Kumaresh0402/SpecialTopicsinComputationalFluidDynamics/discussions/1>

Welcome to SpecialTopicsinComputationalFluidDynamics Discussions! #1

Edit

Kumaresh0402 announced in Announcements



Kumaresh0402 6 minutes ago

Maintainer

...



Welcome!

We're using Discussions as a place to connect with other members of our community. We hope that you:

- Ask questions you're wondering about.
- Share ideas.
- Engage with other community members.
- Welcome others and are open-minded. Remember that this is a community we build together 🤝.

To get started, comment below with an introduction of yourself and tell us about what you do with this community.



Category



Announcements

Labels



None yet

1 participant



Notifications



Unsubscribe

You're receiving notifications because you authored the thread.

Exercise – 1

<https://github.com/Kumaresh0402/SpecialTopicsinComputationalFluidDynamics/discussions/2>

[Exercise-1] Package installations (ANSYS and OpenFOAM) #2

Kumaresh0402 started this conversation in General



Kumaresh0402

5 minutes ago

Maintainer

...

ANSYS Fluent Package: <https://www.ansys.com/products/fluids/ansys-fluent/ansys-fluent-trial>

Please install OpenFOAM and GNU Octave following the documentation in <https://1drv.ms/f/s!AqT2YEB97-1RgP8MtsMPqoOGsq4ddg?e=locXv0>

Follow instructions in installation_steps.pdf

Once after you finish installing, try installing GNU Octave on either Windows or Ubuntu.

Post your queries and make a comment here as part of your Exercise-1.

