Special Topics in Computational Fluid Dynamics (CFD)

Introduction – DAY 1

Kumaresh

Contents

Introduction – About this course

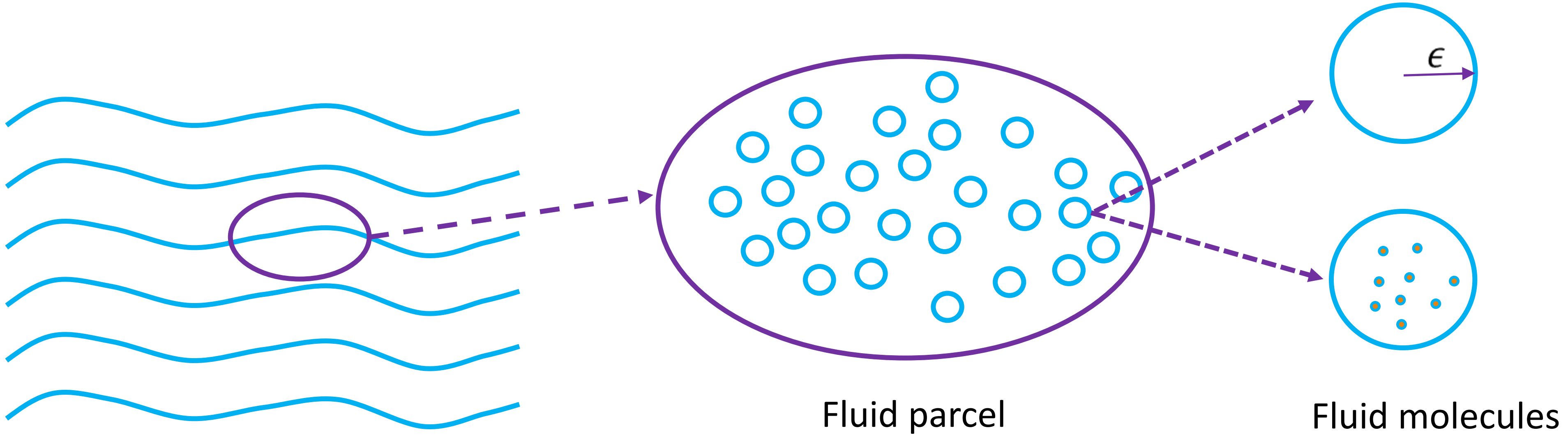
- Course duration per session: 3hrs
- Requirements:
  - Virtual box and installing OS & softwares.
  - Interest to learn CFD using OpenFOAM & Octave
  - Interest to ask questions in discussion forum (GitHub)
  - Work as a team
- Evaluation:
  - Exercises (OpenFOAM and ANSYS Fluent to some extent) (40%)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

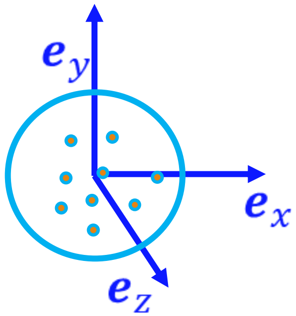
CFD fundamentals – Fluid

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

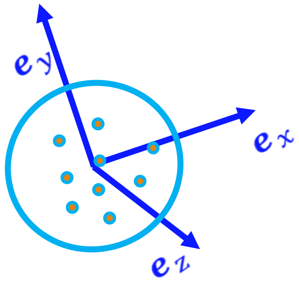


CFD fundamentals – Fluid

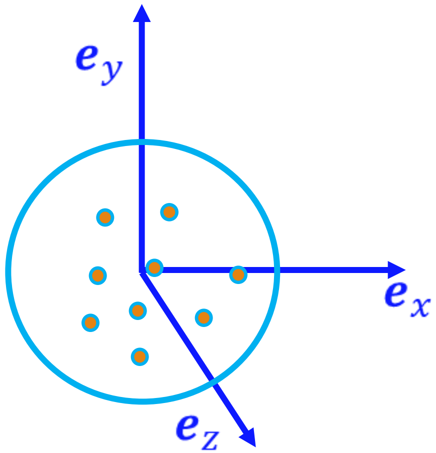
- A substance whose molecular structure offers no resistance to external forces - Ferziger, Peric



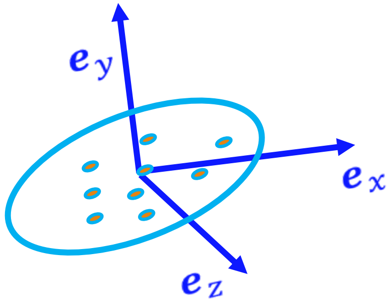
Fluid parcel



Rotation



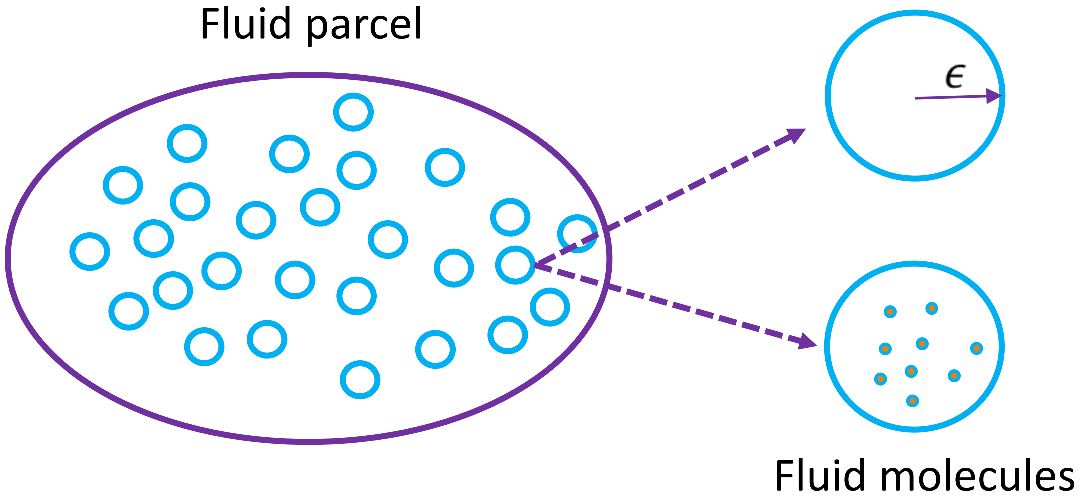
Expansion

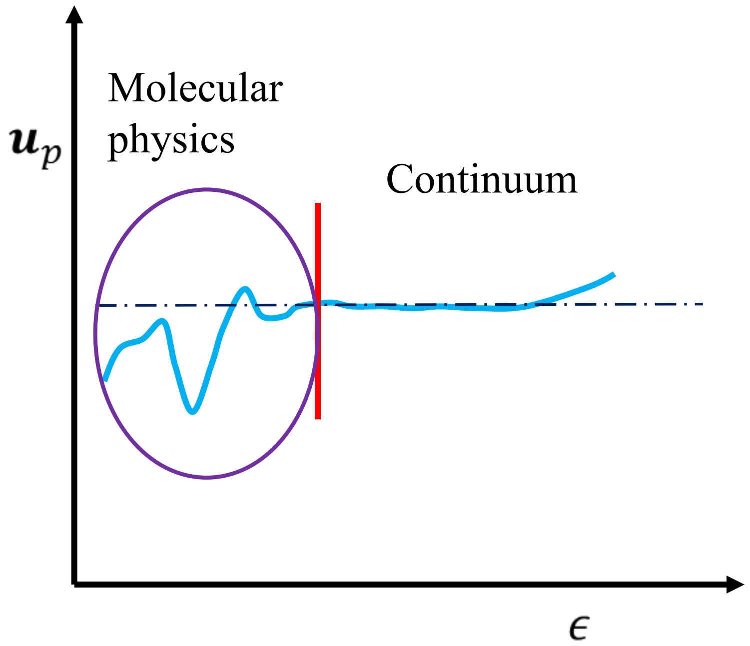


Deformation

CFD fundamentals – Fluid

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





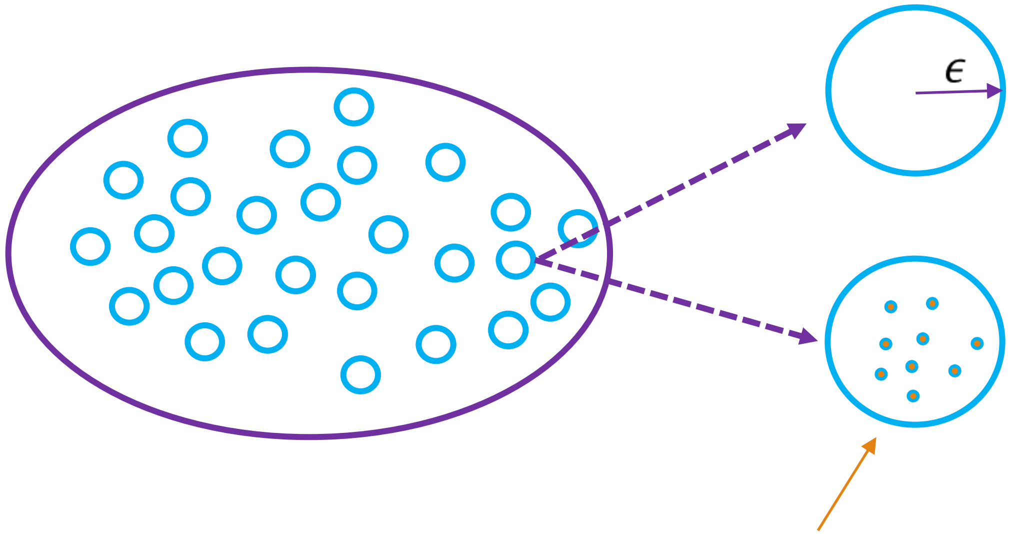
$$\boldsymbol{u}_p = \frac{\sum_{i=1}^{N_{mol}} \boldsymbol{u}_{mol}}{N_{mol}}$$

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

https://en.wikipedia.org/wiki/Knudsen_number#References

CFD fundamentals – Continuum

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

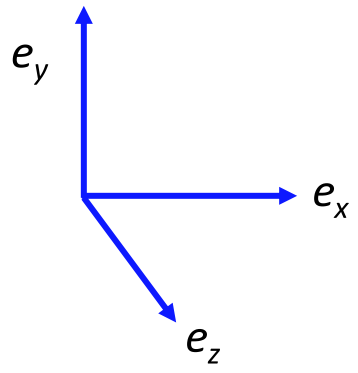


In physics, mean free path is the average distance over which a moving particle

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

$$\boxed{\frac{\partial}{\partial t}(\rho u)+\nabla\bullet(\rho \mathrm{u}u)=\nabla\bullet(\mu\,\nabla u)-\nabla p+S_u}$$

Mathematical operations



Gradient:

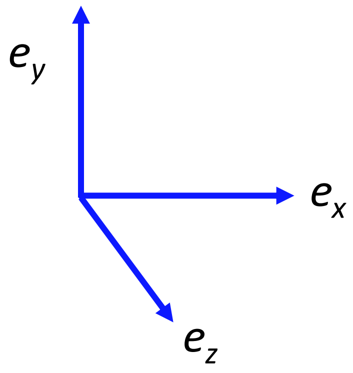
$$\nabla\rho=\left(\frac{\partial}{\partial x}\boldsymbol{e}_x+\frac{\partial}{\partial y}\boldsymbol{e}_y+\frac{\partial}{\partial z}\boldsymbol{e}_z\right)\rho=\left(\frac{\partial\rho}{\partial x}\boldsymbol{e}_x+\frac{\partial\rho}{\partial y}\boldsymbol{e}_y+\frac{\partial\rho}{\partial z}\boldsymbol{e}_z\right)$$

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

Mathematical operations

Gradient:

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



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

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

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

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.

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

Governing Equations

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

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

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

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

1 2 3 4

1 ⟶ Unsteady/Transient term

2 ⟶ Advection/Convection term

3 ⟶ Diffusion term

4 ⟶ Source term

Governing Equations

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

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

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

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

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

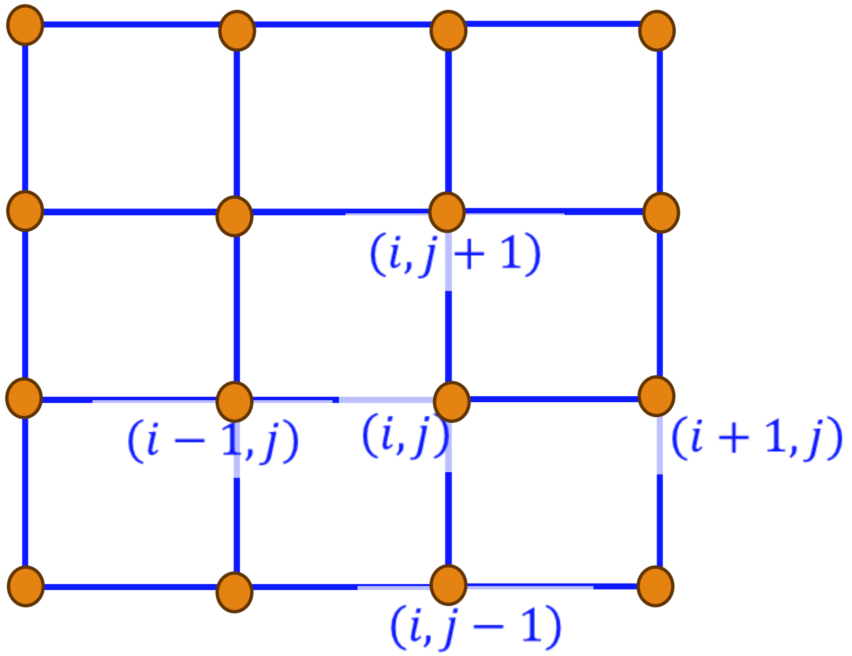
Turbulence equation: $\phi = k(\mathrm{or})\varepsilon, \Gamma = \Gamma_k(\mathrm{or})\Gamma_\varepsilon, S = S_k(\mathrm{or})S_\varepsilon$ ➔ $\boxed{\frac{\partial}{\partial t}(\rho k) + \nabla \bullet (\rho \mathrm{u} k) = \nabla \bullet (\ \Gamma_k \nabla k) + S_k}$

Ideal Gas equation: $\boxed{p = \rho RT}$

Finite Difference – Finite Volume

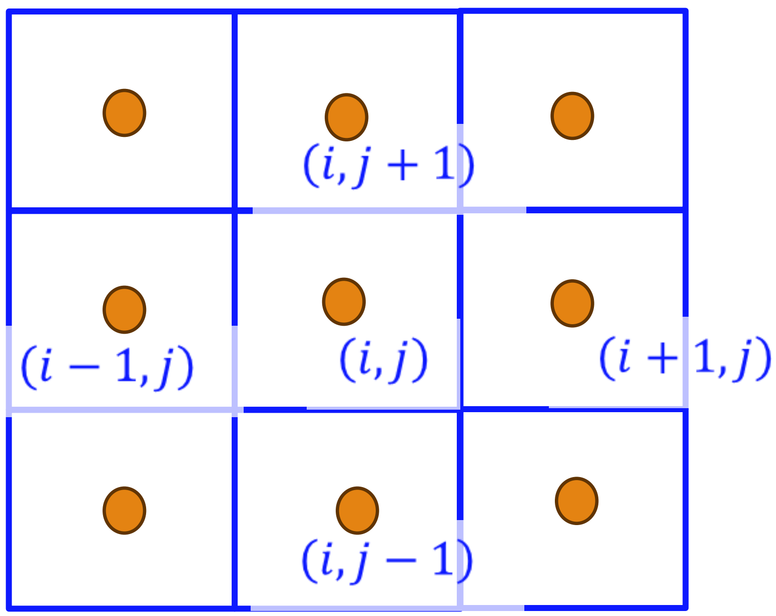
Differential form

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



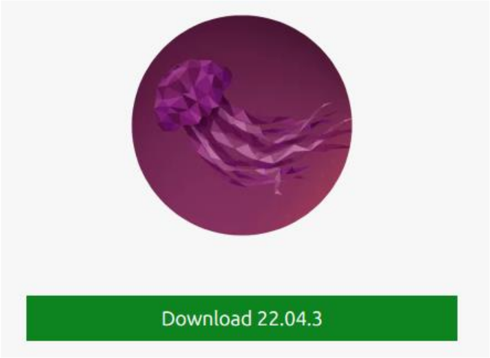
Integral form

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



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]

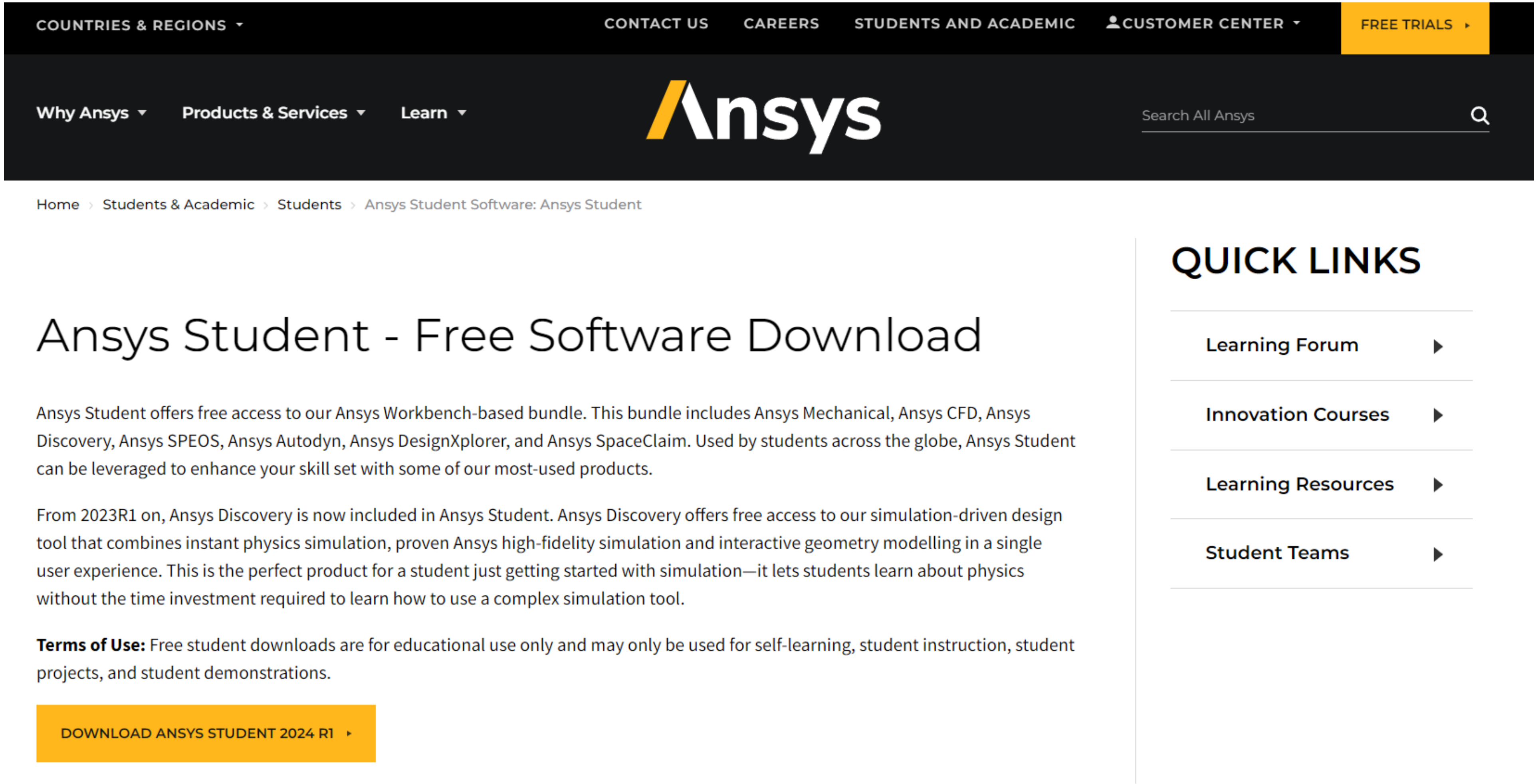


OpenFOAM

GNU Octave

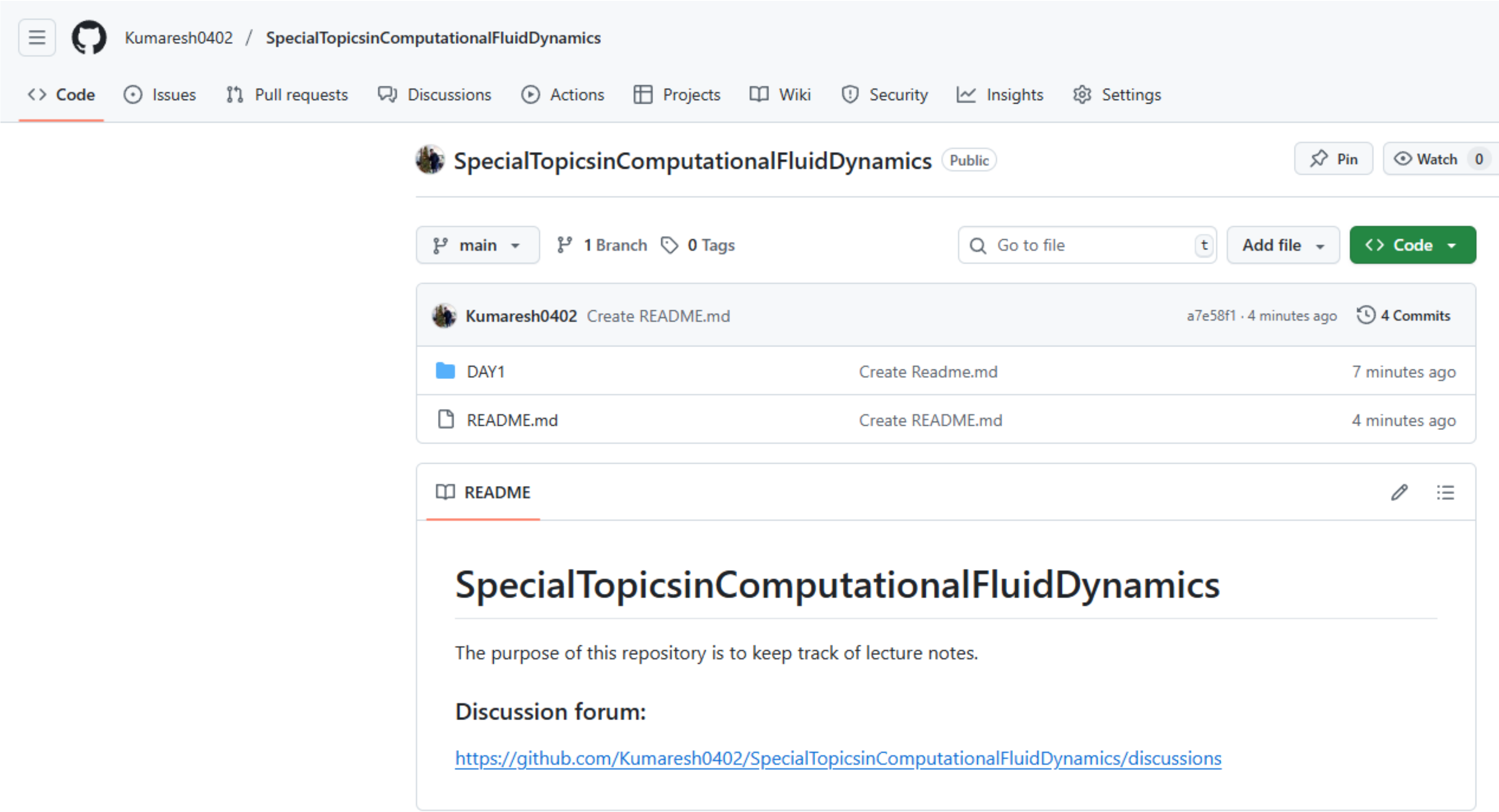
- Create a github account:
  - https://github.com/Kumaresh0402/SpecialTopicsinComputationalFluidDynamics
  - Discussion forum:
    - https://github.com/Kumaresh0402/SpecialTopicsinComputationalFluidDynamics/discussions

ANSYS Student version installation



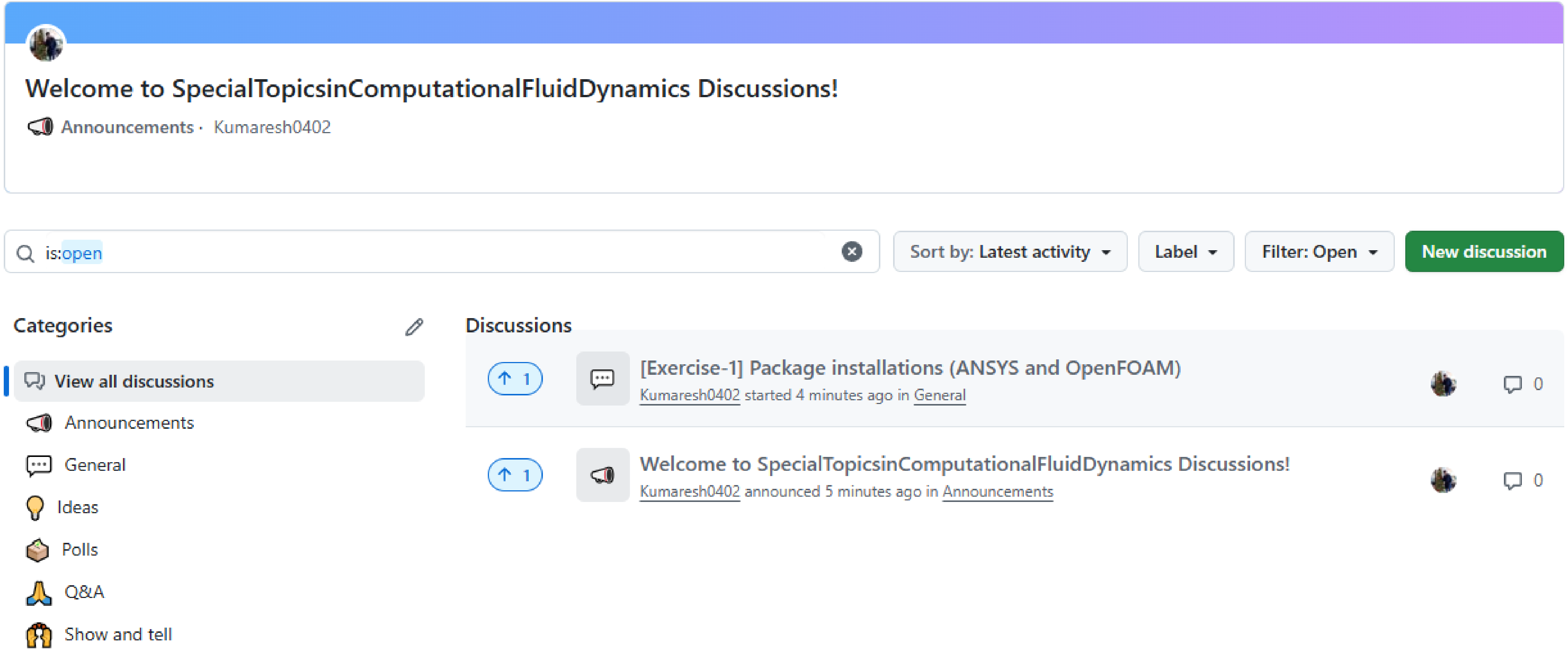
Presentation and Records

https://github.com/Kumaresh0402/SpecialTopicsinComputationalFluidDynamics



Discussions Forum

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



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.

↑ 1

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.

↑ 1