# Introduction to CFD using OpenFOAM

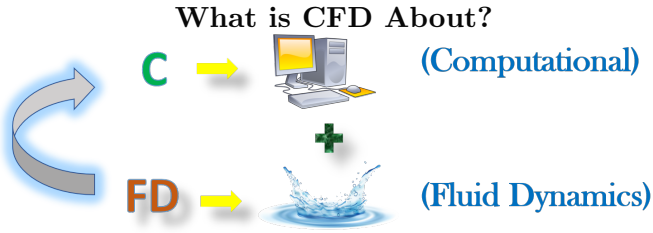

Instructor : Sumant R Morab (Ph.D Research Scholar)
Co-ordinators : (1) Prof. Janani S M (2) Divyesh Variya
Indian Institute of Technology, Bombay

# Outline

- Things to understand before starting a subject/course

- Things to understand before starting a subject/course
  1. What is the subject about?
  2. How it works?

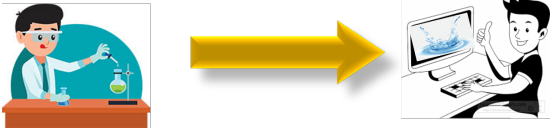
- Things to understand before starting a subject/course
  1. What is the subject about?
  2. How it works?




- Basically.... Investigate Fluid Dynamics (i.e, Fluid flow process) phenomenon using computers !

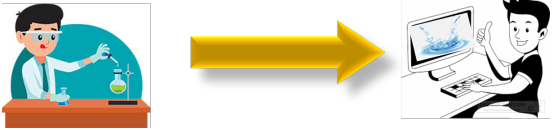
- Let's sum up.....

- Let's sum up.....



- Perform Fluid Flow experiments using computers !

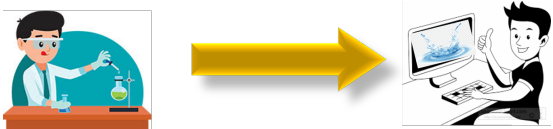
- Let's sum up.....



- Perform Fluid Flow experiments using computers !

**How does CFD work ?**

- Let's sum up.....

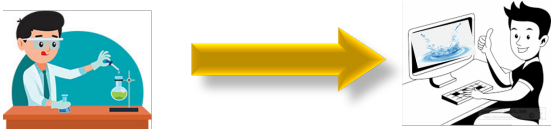


- Perform Fluid Flow experiments using computers !

**How does CFD work ?**

- Is it just applying computer programs for performing experiments ? .... No

- Let's sum up.....

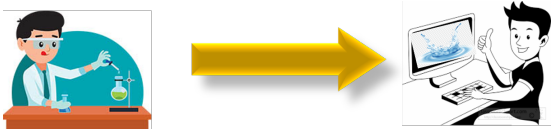


- Perform Fluid Flow experiments using computers !

**How does CFD work ?**

- Is it just applying computer programs for performing experiments ? .... No
  1. Similar to preparing experimental setup like arranging all equipments, One has to write programs, define geometry which they want to study.

- Let's sum up.....

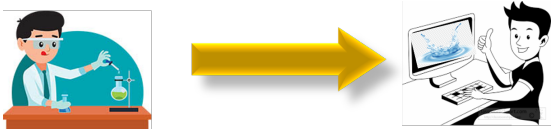


- Perform Fluid Flow experiments using computers !

**How does CFD work ?**

- Is it just applying computer programs for performing experiments ? .... No
  1. Similar to preparing experimental setup like arranging all equipments, One has to write programs, define geometry which they want to study.
  2. After getting results, calculate the require parameters of interest.

- Let's sum up.....



- Perform Fluid Flow experiments using computers !

**How does CFD work ?**

- Is it just applying computer programs for performing experiments ? .... No
  1. Similar to preparing experimental setup like arranging all equipments, One has to write programs, define geometry which they want to study.
  2. After getting results, calculate the require parameters of interest.

# CFD == Video Camera ?



What does a Video Camera do ?

# CFD == Video Camera ?

What does a Video Camera do ?

- Take pictures at different instants of time in a particular region and club them together... i.e, a motion picture !

# CFD == Video Camera ?

What does a Video Camera do ?

- Take pictures at different instants of time in a particular region and club them together... i.e, a motion picture !

What does CFD do?

# CFD == Video Camera ?

What does a Video Camera do ?

- Take pictures at different instants of time in a particular region and club them together... i.e, a motion picture !

What does CFD do?

- Calculate changes in fluid velocity, amount of energy it carries at different time-steps in a particular region of interest. They can be clubbed and played as an animation to visualise flow changes.

# CFD == Video Camera ?

What does a Video Camera do ?

- Take pictures at different instants of time in a particular region and club them together... i.e, a motion picture !

What does CFD do?

- Calculate changes in fluid velocity, amount of energy it carries at different time-steps in a particular region of interest. They can be clubbed and played as an animation to visualise flow changes.
- Only difference - CFD is a silent video camera !!

# CFD == Video Camera ?



What does a Video Camera do ?

- Take pictures at different instants of time in a particular region and club them together... i.e, a motion picture !

What does CFD do?

- Calculate changes in fluid velocity, amount of energy it carries at different time-steps in a particular region of interest. They can be clubbed and played as an animation to visualise flow changes.
- Only difference - CFD is a silent video camera !!

- A step-by-step procedure needs to be followed while investigating Fluid dynamic problem through computational lens .....

- A step-by-step procedure needs to be followed while investigating Fluid dynamic problem through computational lens .....

Development of Solver i.e, computer code – Use any language !

Application for specific problem Eg. Flow over cylinder

Analysis of Results – Make Scientific / Engineering exciting movies !

- So far.... We learnt → What is CFD

- So far.... We learnt → What is CFD
- Last but most important .....

# Why to Study CFD?

- So far.... We learnt → What is CFD
- Last but most important .....

**Why Should I learn CFD & OpenFOAM?**

- 2 Situations ....

- So far.... We learnt → What is CFD
- Last but most important .....

**Why Should I learn CFD & OpenFOAM?**

- 2 Situations ....
  1. Whenever you have a new design and want to see it's performance, you can either perform experiment on prototype/actual model.... Very costly + time-consuming..... Else, sit in front of computer and perform simulations. Get results within a quick time !

- So far.... We learnt → What is CFD
- Last but most important .....

**Why Should I learn CFD & OpenFOAM?**

- 2 Situations ....
  1. Whenever you have a new design and want to see it's performance, you can either perform experiment on prototype/actual model.... Very costly + time-consuming..... Else, sit in front of computer and perform simulations. Get results within a quick time !
  2. Some cases involve situations where experiments cannot be performed... What do you do?

- So far.... We learnt → What is CFD
- Last but most important .....

**Why Should I learn CFD & OpenFOAM?**

- 2 Situations ....
  1. Whenever you have a new design and want to see it's performance, you can either perform experiment on prototype/actual model.... Very costly + time-consuming..... Else, sit in front of computer and perform simulations. Get results within a quick time !
  2. Some cases involve situations where experiments cannot be performed... What do you do?
- Industries (ranging from food processing to aerospace) are aggressively using CFD techniques to obtain higher profits with lesser margins !... They need CFD engineers.

- So far.... We learnt → What is CFD
- Last but most important .....

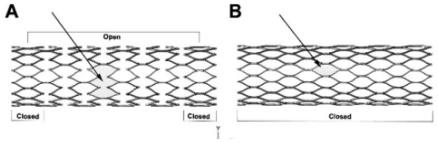
**Why Should I learn CFD & OpenFOAM?**

- 2 Situations ....
  1. Whenever you have a new design and want to see it's performance, you can either perform experiment on prototype/actual model.... Very costly + time-consuming..... Else, sit in front of computer and perform simulations. Get results within a quick time !
  2. Some cases involve situations where experiments cannot be performed... What do you do?
- Industries (ranging from food processing to aerospace) are aggressively using CFD techniques to obtain higher profits with lesser margins !... They need CFD engineers.

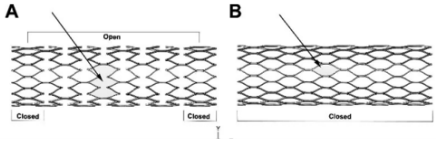
- Let us consider a biomedical application where only CFD can be used ... !
- A person has blockage in blood-carrying Carotid artery (located near neck region).... A stent needs to be put to normalise the blocked region.

- Let us consider a biomedical application where only CFD can be used ... !
- A person has blockage in blood-carrying Carotid artery (located near neck region).... A stent needs to be put to normalise the blocked region.
- Two designs are possible. Objective is to avoid re-stenosis.

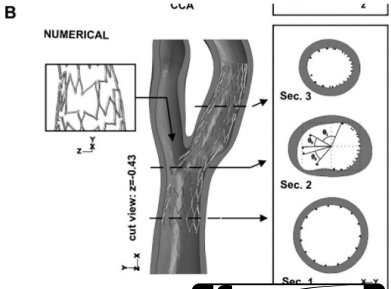
- Let us consider a biomedical application where only CFD can be used ... !
- A person has blockage in blood-carrying Carotid artery (located near neck region).... A stent needs to be put to normalise the blocked region.
- Two designs are possible. Objective is to avoid re-stenosis.



- Let us consider a biomedical application where only CFD can be used ... !
- A person has blockage in blood-carrying Carotid artery (located near neck region).... A stent needs to be put to normalise the blocked region.
- Two designs are possible. Objective is to avoid re-stenosis.




- CFD analysis can be performed on geometry extracted from MRI scan to study vital parameters and check which design performs better !



- Does CFD involve modelling only single-phase fluid flow process ? Like flow in cavity, flow over cylinder, flow inside tube etc ....

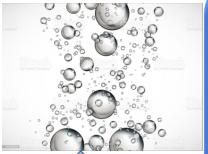
- Does CFD involve modelling only single-phase fluid flow process ? Like flow in cavity, flow over cylinder, flow inside tube etc ....
- Answer is No. Apart from single-phase fluid flow, following are covered in CFD analysis $\rightarrowtail$

- Does CFD involve modelling only single-phase fluid flow process ? Like flow in cavity, flow over cylinder, flow inside tube etc ....
- Answer is No. Apart from single-phase fluid flow, following are covered in CFD analysis $\rightarrowtail$

Multi-Phase Flows :

- Fluid flow involving 2 different phases or two different fluids
- Finds applications in many industrial processes ranging from food making to biomedical field

- Does CFD involve modelling only single-phase fluid flow process ? Like flow in cavity, flow over cylinder, flow inside tube etc ....
- Answer is No. Apart from single-phase fluid flow, following are covered in CFD analysis ⟼

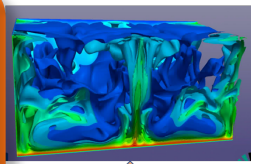


**Multi-Phase Flows :**

- Fluid flow involving 2 different phases or two different fluids
- Finds applications in many industrial processes ranging from food making to biomedical field

**Heat Transfer Study:**

- Differences in temperature may lead to fluid flow variation and variation in flow velocity effects temperature.
- This 2-way interaction finds application in geophysical flows, heat exchangers etc.



- For Development, application and analysis in CFD, there are 2 options →
  1. Develop your own solver – form the governing laws, convert them into linear algebraic equations, write code (important) and test it first before applying to actual problem.

- For Development, application and analysis in CFD, there are 2 options →
  1. Develop your own solver – form the governing laws, convert them into linear algebraic equations, write code (important) and test it first before applying to actual problem.
  2. Use solvers developed by others – Setup geometry, boundary conditions and problem parameters, run the code...

- For Development, application and analysis in CFD, there are 2 options →
  1. Develop your own solver – form the governing laws, convert them into linear algebraic equations, write code (important) and test it first before applying to actual problem.
  2. Use solvers developed by others – Setup geometry, boundary conditions and problem parameters, run the code...

OR

- For Development, application and analysis in CFD, there are 2 options →
  1. Develop your own solver – form the governing laws, convert them into linear algebraic equations, write code (important) and test it first before applying to actual problem.
  2. Use solvers developed by others – Setup geometry, boundary conditions and problem parameters, run the code...

OR

- Many Solvers are available like Fluent (ANSYS), ADINA, COMSOL etc... But these softwares need to be purchased.
- OpenFOAM is an open-source software ... So... It's free of cost ! Also, we can tweak the codes to suit our problem statement.

- Wikipedia[2] Says this

  " OpenFOAM (for "Open-source Field Operation And Manipulation") is a C++ toolbox for the development of customized numerical solvers, and pre-/post-processing utilities for the solution of continuum mechanics problems, most prominently including computational fluid dynamics (CFD) "

- Wikipedia[2] Says this

  " OpenFOAM (for "Open-source Field Operation And Manipulation") is a C++ toolbox for the development of customized numerical solvers, and pre-/post-processing utilities for the solution of continuum mechanics problems, most prominently including computational fluid dynamics (CFD) "

- In simple words, it's a book which contains several programs which we can directly copy and use or edit it to suit our needs.

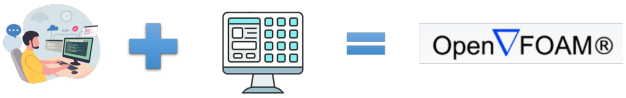
- Wikipedia[2] Says this
  " OpenFOAM (for "Open-source Field Operation And Manipulation") is a C++ toolbox for the development of customized numerical solvers, and pre-/post-processing utilities for the solution of continuum mechanics problems, most prominently including computational fluid dynamics (CFD) "
- In simple words, it's a book which contains several programs which we can directly copy and use or edit it to suit our needs.

- Wikipedia[2] Says this
  " OpenFOAM (for "Open-source Field Operation And Manipulation") is a C++ toolbox for the development of customized numerical solvers, and pre-/post-processing utilities for the solution of continuum mechanics problems, most prominently including computational fluid dynamics (CFD) "
- In simple words, it's a book which contains several programs which we can directly copy and use or edit it to suit our needs.

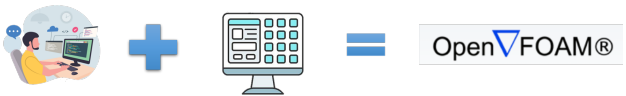

- Developers are spread across globe including India. A user can also contribute towards development in special cases !

- To perform CFD investigation, there are many tools – like develop your own solver, use commercial software with beautiful GUI etc....

- To perform CFD investigation, there are many tools – like develop your own solver, use commercial software with beautiful GUI etc....



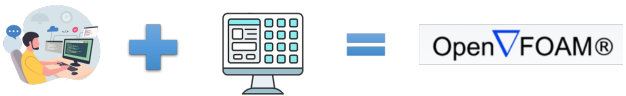
- To perform CFD investigation, there are many tools – like develop your own solver, use commercial software with beautiful GUI etc....




- What if both capabilities are present at single location... and that too free !

# Why OpenFOAM?

- To perform CFD investigation, there are many tools – like develop your own solver, use commercial software with beautiful GUI etc....




- What if both capabilities are present at single location... and that too free !
- That's why OpenFOAM is the best option.
- Following phenomenon can be studied through OpenFOAM (not restricted though...)

- To perform CFD investigation, there are many tools – like develop your own solver, use commercial software with beautiful GUI etc....


- What if both capabilities are present at single location... and that too free !
- That's why OpenFOAM is the best option.
- Following phenomenon can be studied through OpenFOAM (not restricted though...)

SINGLE-PHASE MULTI-PHASE HEAT-TRANSFER

FLUID-STRUCTURE-INTERACTION

# Summary of Lecture

So, what did we learn so far.....

1. What is CFD – Analogy with Video Camera

# Summary of Lecture

So, what did we learn so far.....

1. What is CFD – Analogy with Video Camera
2. Stages involved in a CFD Study

# Summary of Lecture

So, what did we learn so far.....

1. What is CFD – Analogy with Video Camera
2. Stages involved in a CFD Study
3. Why to Study CFD? A Biomedical Case Study

# Summary of Lecture

So, what did we learn so far.....

1. What is CFD – Analogy with Video Camera
2. Stages involved in a CFD Study
3. Why to Study CFD? A Biomedical Case Study
4. OpenFOAM as a CFD tool – It's capabilities

# Summary of Lecture

So, what did we learn so far.....

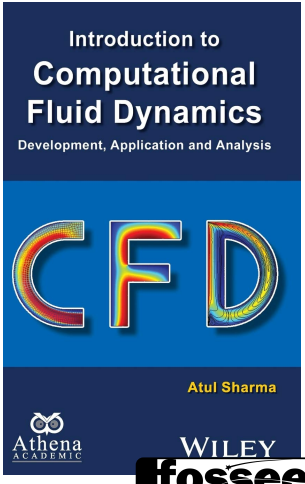
1. What is CFD – Analogy with Video Camera
2. Stages involved in a CFD Study
3. Why to Study CFD? A Biomedical Case Study
4. OpenFOAM as a CFD tool – It's capabilities
5. Why OpenFOAM? It's advantages

So, what did we learn so far.....

1. What is CFD – Analogy with Video Camera
2. Stages involved in a CFD Study
3. Why to Study CFD? A Biomedical Case Study
4. OpenFOAM as a CFD tool – It's capabilities
5. Why OpenFOAM? It's advantages

Next Class, we shall see a brief overview of Development, Application and Analysis which is crux of CFD !

1. Sharma, A. (2016). Introduction to computational fluid dynamics: development, application and analysis. John Wiley & Sons.
2. https://en.wikipedia.org/wiki/OpenFOAM
3. https://www.openfoam.com/
4. Conti, M., Loo, D. V., Auricchio, F. et al. (2011). Impact of carotid stent cell design on vessel scaffolding: a case study comparing experimental investigation and numerical simulations. Journal of Endovascular Therapy, 18(3), 397-406.



Thank you for listening!

Sumant Morab