# **Roadmap to Lecture 0**

## 1. Quick review of solution methods in CFD

## 2. What is Ansys Fluent? – Executive summary

3. Turbulence in action

- The workhorse CFD solver to be used during this course is Ansys Fluent.
- So, it is pertinent to give an executive summary of the solver.
- It is important to mention that all turbulence modeling theory that will be covered during this course is the same independently of the CFD solver used.
- Also, the standard practices are the same for every single CFD solver.
- You might find some differences between CFD solvers, but they are due to implementation details.
- It is always recommended to refer to the help system of the solver for more information on the implementation details.
- Also, different CFD solvers will add different corrections (or black magic) into their solver implementations.

- Brief overview Description and implementation:
  - First of all, Ansys Fluent is a multiphysics solver.
  - It is based on the Finite Volume Method (FVM).
  - It uses collocated unstructured meshes Cell centered formulation.
  - It can deal with hanging nodes.
  - Rhie-Chow interpolation to avoid the checkerboard instability.
  - It comes with pressure-based solvers (segregated and fully coupled).
    - Many approaches available: SIMPLE, SIMPLEC, PISO, Fractional step, coupled.
  - It also comes with density-based solvers.
  - Implicit and explicit solvers.

- Brief overview Description and implementation:
  - First and second order accuracy (and even higher) in space and time.
  - Many discretization schemes available.
    - Space: upwind, central differencing, second order upwind, high order schemes (TVD), QUICK, and so on.
    - Time: steady, first order, second order, local time-step, and so on.
  - Diffusion terms are evaluated using central differences with secondary gradient corrections (also known as non-orthogonal corrections).
  - Gradients can be approximated using Gauss method or the least squares method.
  - Efficient multigrid solver for solving the linear system of equations.
  - Can run in parallel and in GPUs.

- Ansys Fluent is capable of modeling:
  - Steady state and transient flows.
  - Laminar and turbulent flows.
  - Subsonic, transonic, and supersonic flows.
  - Heat transfer and radiation.
  - Multiphase flows Dispersed and separated flows.
  - Non-Newtonian flows.
  - Transport of non-reacting and reacting scalars (species).
  - Combustion and chemical reactions.
  - Particle tracking and interaction.

- Ansys Fluent is capable of modeling:
  - Adaptive mesh refinement.
  - Moving bodies, rigid body motion, and multiple reference frames.
  - Magnetohydrodynamics.
  - Electrical potential.
  - Acoustics.
  - Fluid structured interaction.
  - Adjoin optimization.
  - Can be coupled with external solvers to solve complex multiphysics problems (FSI, aerovibro acoustics, combustion, and so on).
  - And many more ...

• When using Ansys Fluent for CFD studies, we are solving the following equations:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$
$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) = -\nabla p + \nabla \cdot \tau + \mathbf{S}_{\mathbf{u}}$$
$$\frac{\partial (\rho e_t)}{\partial t} + \nabla \cdot (\rho e_t \mathbf{u}) = -\nabla \cdot q - \nabla \cdot (p \mathbf{u}) + \mathbf{\tau} : \nabla \mathbf{u} + \mathbf{S}_{e_t}$$

Additional equations deriving from models, such as, turbulence modeling, multiphase flows, heat transfer, radiation, chemical reactions, combustion, multi-species, thermodynamics, particle interaction, acoustics, mass transfer, and so on.

+

- These are your largest sources of uncertainty.
- Other sources of uncertainty:
  - Round-off errors (computer precision).
  - Iteration errors.
  - Discretization errors.
  - User errors.

- When using Ansys Fluent, follow these guidelines and standard practices:
  - It is highly advisable to have a good quality mesh.
  - Your final solution should be second order accurate and bounded (non-oscillatory).
  - Turbulence modelling has a significant impact on the final solution. Choose an appropriate method.
  - Choose physically realistic boundary conditions and initial conditions.
  - Monitor your solution. Not only the residuals but also integral quantities (such as forces, mass flow, heat transfer rate, and so on).
  - Understand the physics.
- Most of the times, the default settings proposed by Ansys Fluent are fine.
- Read the help system provided with Ansys Fluent, it is very complete.
- All the previous guidelines and standard practices apply to any CFD solver.



tmp Parallel Fluent@DESKTOP-NMOF58H [3d, dp, pbns, sstkw, tran	nsient, 4-processes]				– o ×
<u>F</u> ile Domain Physics User-D	efined Solution Results Vie	ew Parallel De	esign 🔺	Q Quick Search (Ctrl+ <del>F</del> )	1 The second sec
Mesh O Display O Info - Occession Check- Quality - Occession - Make Polyhedra	Zones       Combine	Interfaces         Mesh Models           Image: A state of the s	Turbo Model        □ Enable        🐨 Turbo Topology       □] Turbo Create     ০০০ More	Adapt     Surface       fine / Coarsen     + Create _       Adapt     Amage       re     -	
Outline View	< Task Page			temporary-mesh-0	×
Filter Text	Models				ANSYS
C Setup     R General     O Models     D Models     D Materials     Cell Zone Conditions     Cell Zone Conditions     G Mach Interfaces	Models Multiphase - Off Energy - On Viscous - SST k-omega Radiation - Off Heat Exchanger - Off Sparines - Off				ACADEMIC
Dynamic Mesh	Discrete Phase - Off Solidification & Melting - Off Acoustics - Off		Ad	dditional physical models	
	Acoustics - Off Structure - Off Eulerian Wall Film - Off Potential/Livion Battery - Off Battery Model - Off				
<ul> <li>Ø Cell Registers</li> <li>Initialization</li> <li>● Calculation Activities</li> <li>○ Run Calculation</li> <li>○ Results</li> </ul>	Edit				
					la p
					0 selected all 💌
		Console adjoint/ define/ display/ exit file/ >	mesh/ rep parallel/ ser plot/ sol preferences/ sol print-license-usage view	port/ cver/ lve/ fáce/ ws/	> @ • ① • •



tmp Parallel Fluent@DESKTOP-NMOF58H [3d, dp, pbns, sstkw,	transient, 4-processes]				– o ×
<u>File Domain Physics Use</u>	er-Defined Solution Results View	v Parallel De	sign 🔺		🔍 Quick Search (Ctrl+F) 🛛 🕖 📜 🖍 🕅 🖓
Mesh	Zones	Interfaces Mesh Models	Turbo Model	Adapt Surface	
Display - Scale	Combine T Delete	Mesh Ovnamic Mesh	Enable	Refine / Coarsen + Create	
			C Turbe Tenelogy		¥
1) Info	Separate      Deactivate     Replace Mesh	Mixing Planes	i urbo i opology	🚲 Manage	
Vnits Check Quality - 4 Make Polyhedr	ra   💤 Adjacency 🖽 Activate 🛛 💾 Replace Zone		Durbo Create •	More 👻	
Outline View	< Task Page	< 🛄 🔼		temporar	y-mesh-0 X
Filter Text	Solution Methods				ANSYS
⊙ Setup	Pressure-Velocity Coupling				2021 <b>R1</b>
40 General	Scheme	*			DIE DICO Counted and as an
A Models      Materials	SIMPLE	▼ -Q+	Solverion	nulation – Silvi	PLE, PISO, Coupled, and so on
Cell Zone Conditions	Spatial Discretization				$\times$ $\times$ $\times$ $\times$
📀 🖽 Boundary Conditions	Gradient				
Mesh Interfaces	Least Squares Cell Based 💌				
Dynamic Mesh	Pressure				
Keference Frames	Second Order		Spatial o	discretization s	chemes – First order upwind
Image: Second	Density		Opullar		
Solution     Mathada	Second Order Upwind 💌		second or	der upwind. ce	entral differencing, QUICK, third
X Controls	Momentum	Q			1001 I
📀 🛐 Report Definitions	Second Order Upwind 💌			order ML	JSCL, and so on
<ul> <li>Monitors</li> </ul>	Turbulent Kinetic Energy		~		
Cell Registers     Juitiplication	First Order Upwind 🔻				
	Specific Dissination Rate				
Run Calculation	Transient Formulation		Time discre	etization schen	nes – First order Fuler, backward
- Results	First Order Impli 🔻				
Graphics	Non-Iterative Time Advancement			Euler, bound	led Euler, and so on
💿 🖵 Plots	Frozen Flux Formulation				
💿 🏟 Scene	✓ Warped-Face Gradient Correction				
Animations     Benerits	✓ High Order Term Relaxation				
Parameters & Customization	Options		$\rightarrow$		
Simulation Reports	Default Report Poor Quality Elements				
				$\sim$	
					0 selected all 💌
		Console			ଡ <
		adjoint/	meeh/	report/	
		define/	parallel/	server/	
		display/	plot/	solve/	
		file/	print-license-usage	views/	
					*
		-			<u>v</u>

tmp Parallel Fluent@DESKTOP-NMOF58H [3d, dp, pbns, sstkw, tran	nsient, 4-processes]					– o ×	
<u>File</u> Domain Physics User-D	efined Solution Results Vie	w Parallel Des	ign 🔺		${f Q}$ Quick Search (Ctrl+F)	🔿 📄 🔊	
Mesh Scale ③ Display ③ Info ↓ ④ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓	Zones       Image: Combine with the second sec	Interfaces         Mesh Models           Image: An angle of the second	Turbo Model       Enable       Turbo Topology       Turbo Create	Adapt Surface Refine / Coarsen + Create - Manage			
Outline View	< Task Page	· · · · · · · · · · · · · · · · · · ·		temporary-me	sh-0	×	
Filter Text	Solution Controls					ANSYS	
Setup         Image: Constraint of the setup of the	Under-Relaxation Factors Pressure 0.3 Density 1 Body Forces 1 Momentum 0.7		Solution co	ontrols and und	er-relaxation factors	ACADEMIC	
<ul> <li>Jor Named Expressions</li> <li>Solution</li> <li>Solution</li> <li>Methods</li> <li>Controls</li> <li>Report Definitions</li> <li>A Monitors</li> <li>A Cell Registers</li> <li>Initialization</li> <li>Calculation Activities</li> <li>Cancella Registers</li> <li>Calculation Activities</li> <li>Graphics</li> <li>Jor Scene</li> <li>Animations</li> <li>Parameters &amp; Customization</li> <li>Simulation Reports</li> </ul>	U.7 Turbulent Kinetic Energy 0.8 Specific Dissipation Rate 0.8 Turbulent Viscosity 1 Energy 1 Default Equations Limits Advanced		Solution controls and under-relaxation factors				
		Console			0 se	ected all	
		adjoint/ define/ display/ exit file/ >	mesh/ rr parallel/ se plot/ se preferences/ se print-license-usage v:	report/ erver/ bolve/ uurface/ riews/			







- And after computing the solution, we need to give physical meaning to the results.
- Quantitative and qualitative postprocessing.

