Roadmap to Lecture 5

- 1. Governing equations of fluid dynamics
- 2. RANS equations Reynolds averaging
- **3. The Boussinesq hypothesis**
- 4. The gradient diffusion hypothesis
- **5. Sample turbulence models**
- 6. Quick review of solution methods for the governing equations of fluid dynamics
- 7. What is Ansys Fluent? Executive summary

- The workhorse solver 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 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 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.
- By the way, the previous guidelines and standard practices apply to any CFD solver.

<u>File</u> Domain	Physics User-Defined	Solution Res	sults View	Parallel Des	ign 🔶			Q Quick Search (Ctrl+F)	🖲 📕 🔊
Mesh Display Di Info ~ Ocheck~ Qualit Outline View	👃 🖓 Transform 🖕 🖵 Sep	arate 🚽 🔲 Deactivate	Append Therface	. 🤣 Dynamic Mesh	Turbo Model Enable Turbo Topology Turbo Create	Adapt	Surface + Create + Manage User Window 1		×
Filter Text Filter Text	General Mesh Scale Display Solver Type Pressure-Base Density-Based Time Steady Transient				type – Pre eady or tra	ANSY 2021 ACADEM			
 Results ● Surfaces ● Graphics ● Poly Poly ● Animations ● Parameters & Customization ● Simulation Reports 			nn ce bann materin interfé domainn mixture zones, wall outlet inlet	or-geom_solid	g Reverse Cuthill-M	cKee: 80143/1241 = 6	54.5794	0	selected all

<u>F</u> ile	Domain		Physics	User-Defir	ned So	olution	Results	View	Parallel	Desi	gn 🔺				Q	Quick Search (C	Ctrl+F)	0	ANSYS
Implies Oisplay Implies Implies Implies Implies	Check	Mesh Quality	■ Scale. Contemporative Co	form 🖕 🕻	Separate	Zones	[™] Append [™] Replace M Replace Zo		sh 🧞 Dyn	h Models amic Mesh ing Planes	Turbo Model Enable Turbo Topology Turbo Create	Adapt Refine / Coarsen	Surface + Create -						
Outline View			<	Task Page			(?)	< 📐 🚨					User Window 1					4	× ANSYS
Filter 1 ext ● Stup ● General ● Models ● Models ● Models ● Soundary Conditions ● Boundary Conditions ● Boundary Conditions ● Boundary Conditions ● Mathing Controls ● Mathing Controls ● Controls ● Report Definitions ● A Monitors ● Calculation ● Calculation ● Results ● Ø Surfaces ● Ø Surfaces ● Ø Surfaces ● Ø Das				Scheme SIMPLE Spatial Disc Gradient	/elocity Coupl	ing			[Solver formulation – SIMPLE, PISO, Coupled, and so on								A	2021 RI ACADEMIC
				Pressure Second Order Density Second Order Upwind Momentum Second Order Upwind Turbulent Kinetic Energy			•			Spatial discretization schemes – First order upwind, second order upwind, central differencing, QUICK, third order MUSCL, and so on								d	
			Ĺ	Second Order Upwind Snerdir: Unsenation Kate Transient Formulation First Order Impli * Non-Iterative Time Advancement Frozen Flux Formulation					Time discretization schemes – First order Euler, backward Euler, bounded Euler, and so on								ırd		
📀 💽 Anir 📀 🛅 Rep					Face Gradient												0 selected	dall	•
Parameter Simulation	rs & Custon Reports	lization		High Ord	ler Term Relax	options		Console											0 <
				Default				mate inte doma mixt zone wall outl inle inte geom para Done.	rials, riface, iins, uure ss,	id		<pre>ickee: 80143/1241 = e ve-rplot, heat-trans</pre>		t updated to	o get data	every time-s	step.		

Mesh Zones Interfaces Mesh Models Turbo Model Adapt Surface Image: Sole Image: Sole<	
① Info ① Info ① ① ①	
Outline View Task Page Outline View Task Page Outline View Solution Controls Outline View Solution Controls Outline View Solution Controls Outline View Solution Controls Outline View Solution Controls Outline View Solution Controls Outline View Solution Controls Outline View Solution Controls Outline View Solution Controls Outline View Solution Controls Outline View Solution Controls Outline View Solution Controls Outline View Solution Control Outline View Solution Control	

<u>F</u> ile	Domain	Phy	ysics	User-Defined	Solution	Results	View	Parallel	Desi	jn 🔺			Q Quick Sear	ch (Ctrl+F)	0	ANSYS
i Info		(127)	Scale		Zone: Dine _▼ ⊞ [×] Delete rate _▼ ⊞ Deactiva	+ Append	- 🎛 M			Turbo Model Enable	Adapt Refine / Coarsen	Surface + Create +				
🖌 Units	Check-	Quality 👻	Make Po	lyhedra 🛛 🛷 Adja	cency 🕂 Activate.					D Turbo Create	👓 More 👻					
Outline View			<	Task Page			< No. 100 August 100 A					User Window 1				×
Filter Text				Run Calculation		(?)									Α	2021 R1
🛞 🖽 Bour 🕫 Mes	els			Check Cas Time Advancemen Type Fixed Parameters	t Method Vser-Sp										AC	ADEMIC
🖹 Refe	erence Values			Number of Time S	teps Time Ste	ep Size [s]	◀ 📄		Ite	rative or t	time march	ning pa	rameters			
	erence Frames ned Expression	s		Max Iterations/Tim	•	g Interval										
Solution Solution Met X Cont S Rep	hods trols ort Definitions			20 Profile Update Inte	1 ival	\$										
💿 🗨 Mor 😰 Cell	nitors Registers			Options												
nitia 🕄	alization Jation Activities	_		Extrapolate Va												
	Calculation	5			d Conjugate Heat Tran	isfer									ľ	•
📀 🤚 Surfi 🛞 🔮 Grap				Solution Processin	g											→× _A
📀 🗾 Plot	s			Statistics	for Time Statistics		i								~	4
🔹 💽 Anin 📀 📑 Rep														0 selecte	d Su	•
Parameter	rs & Customiz	ation			Data File Quantities		Console							0 3010000		0 <
 Simulation 	Reports			Solution Advance			console	nodes,								
					Calculate		mat int don mix zor wal out in1 int geo par Done.	cells, bandwidth reduct erials, erface, mains, ture etes, l let et erior-geom_solid m_solid callel,			ickee: 80143/1241 = 6 we-rplot, heat-trans		s updated to get data every ti	ne-step.		