

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**

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.

What is Ansys Fluent? – Executive summary

- 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.

What is Ansys Fluent? – Executive summary

- 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.

What is Ansys Fluent? – Executive summary

- 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.

What is Ansys Fluent? – Executive summary

- 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, aero-vibro acoustics, combustion, and so on).
 - And many more ...

What is Ansys Fluent? – Executive summary

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

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

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.

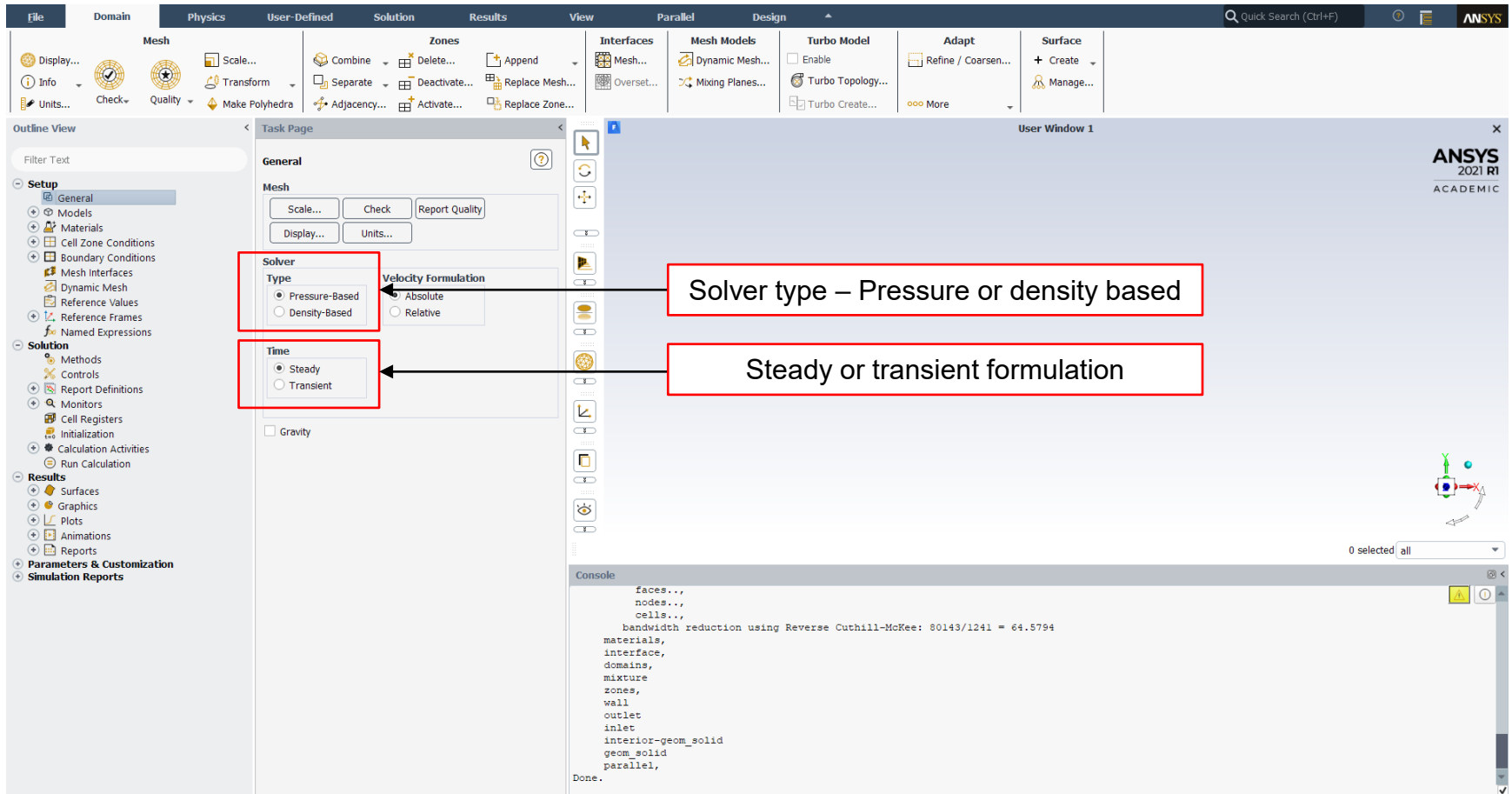


What is Ansys Fluent? – Executive summary

- 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.

What is Ansys Fluent? – Executive summary

- Where do I set the numerical method in Ansys Fluent?



What is Ansys Fluent? – Executive summary

- Where do I set the numerical method in Ansys Fluent?

The screenshot displays the Ansys Fluent 2021 R1 Academic interface. The 'Solution Methods' panel is open, showing various settings for the simulation. Three red boxes highlight specific areas, each with an annotation:

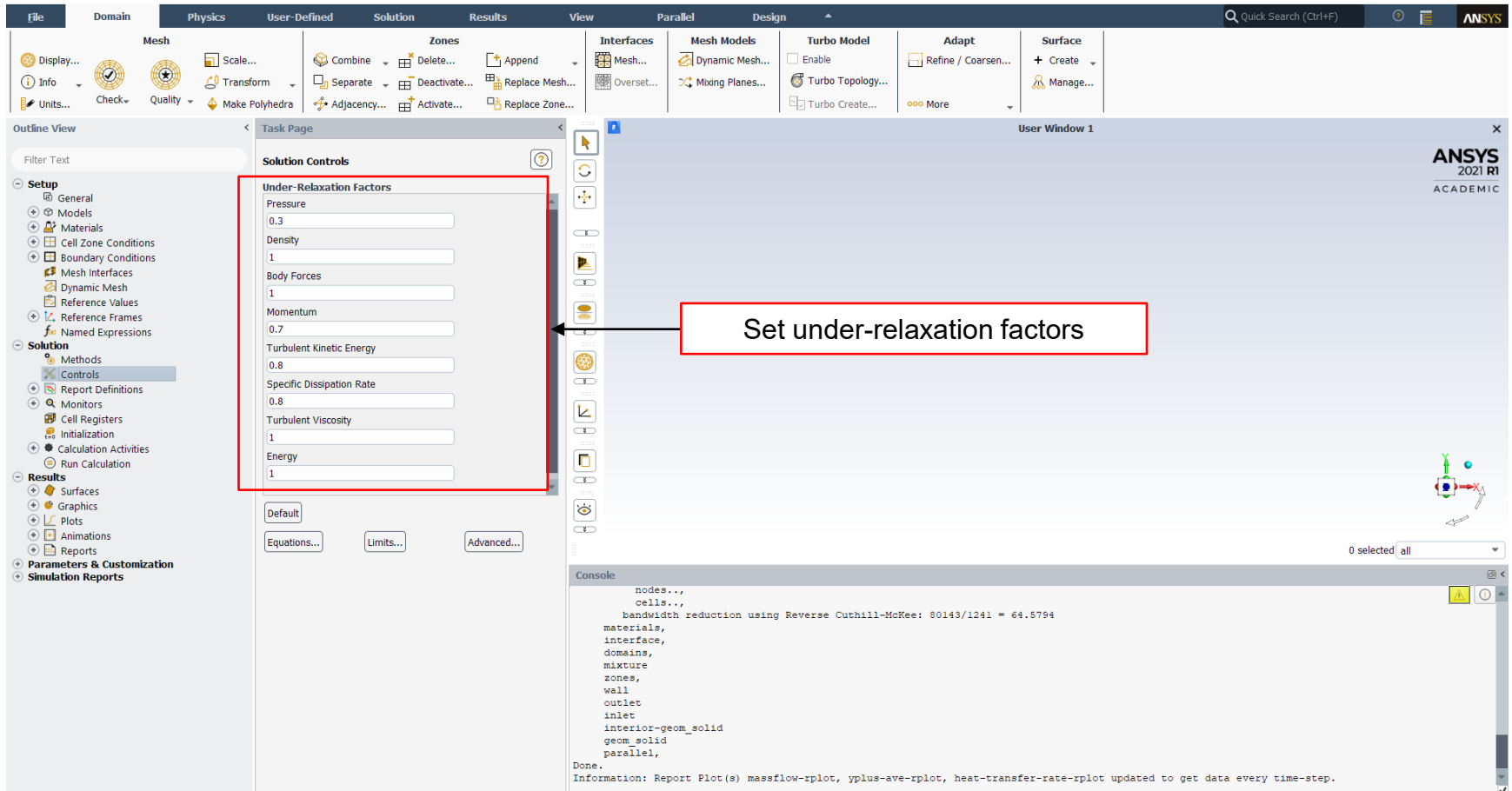
- Solver formulation – SIMPLE, PISO, Coupled, and so on**: Points to the 'Scheme' dropdown menu under 'Pressure-Velocity Coupling', which is currently set to 'SIMPLE'.
- Spatial discretization schemes – First order upwind, second order upwind, central differencing, QUICK, third order MUSCL, and so on**: Points to the 'Spatial Discretization' section, which includes dropdowns for 'Gradient' (Least Squares Cell Based), 'Pressure' (Second Order), 'Density' (Second Order Upwind), 'Momentum' (Second Order Upwind), and 'Turbulent Kinetic Energy' (Second Order Upwind).
- Time discretization schemes – First order Euler, backward Euler, bounded Euler, and so on**: Points to the 'Transient Formulation' dropdown menu, which is currently set to 'First Order Impl'.

The 'Console' window at the bottom shows the following text:

```
nodes...
cells...
bandwidth reduction using Reverse Cuthill-McKee: 80143/1241 = 64.5794
materials,
interface,
domains,
mixture
zones,
wall
outlet
inlet
interior-geom_solid
geom_solid
parallel,
Done.
Information: Report Plot(s) massflow-rplot, yplus-ave-rplot, heat-transfer-rate-rplot updated to get data every time-step.
```

What is Ansys Fluent? – Executive summary

- Where do I set the numerical method in Ansys Fluent?



What is Ansys Fluent? – Executive summary

- Where do I set the numerical method in Ansys Fluent?

