Roadmap to Lecture 0

1. ~~Quick review of solution methods in CFD~~
2. **What is Ansys Fluent? – Executive summary**
3. Turbulence in action

What is Ansys Fluent? – Executive summary

- 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.

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:

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

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

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

$$+$$

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.

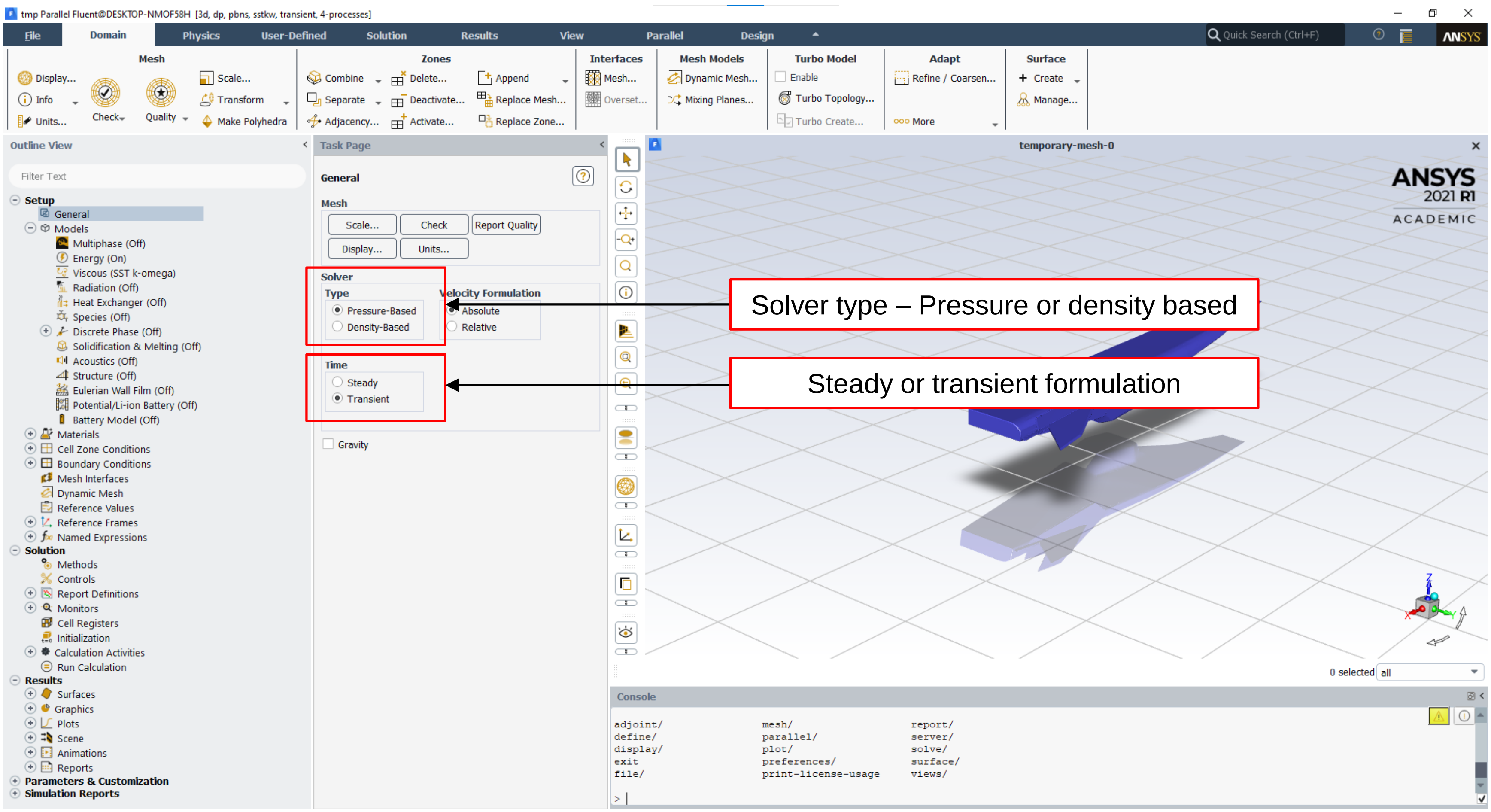
- These are your largest sources of uncertainty.
- Other sources of uncertainty:
  - Round-off errors (computer precision).
  - Iteration errors.
  - Discretization errors.
  - User errors.

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.
- All the previous guidelines and standard practices apply to any CFD solver.

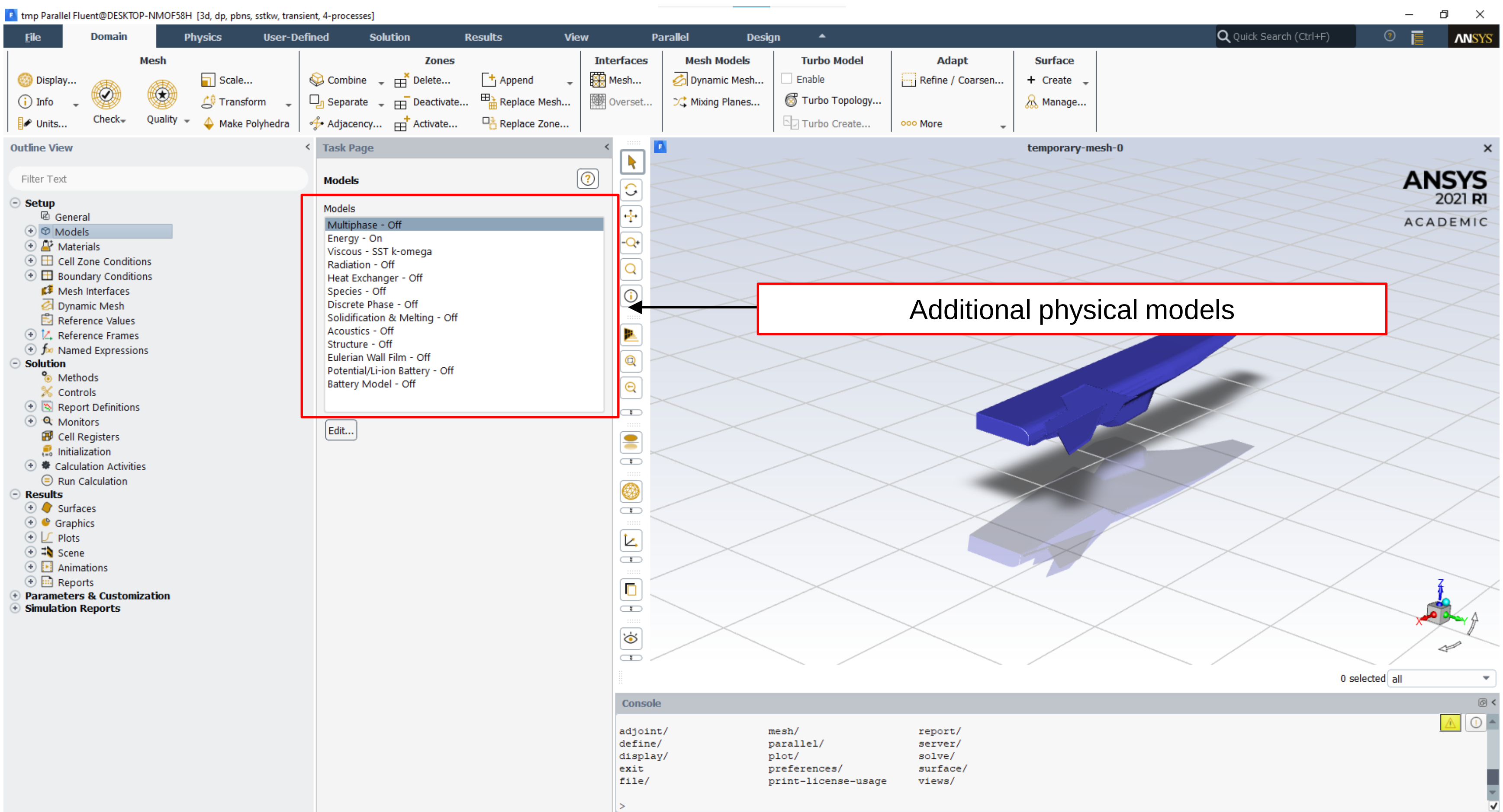
What is Ansys Fluent? – Executive summary

- Quick overview of Ansys Fluent GUI.



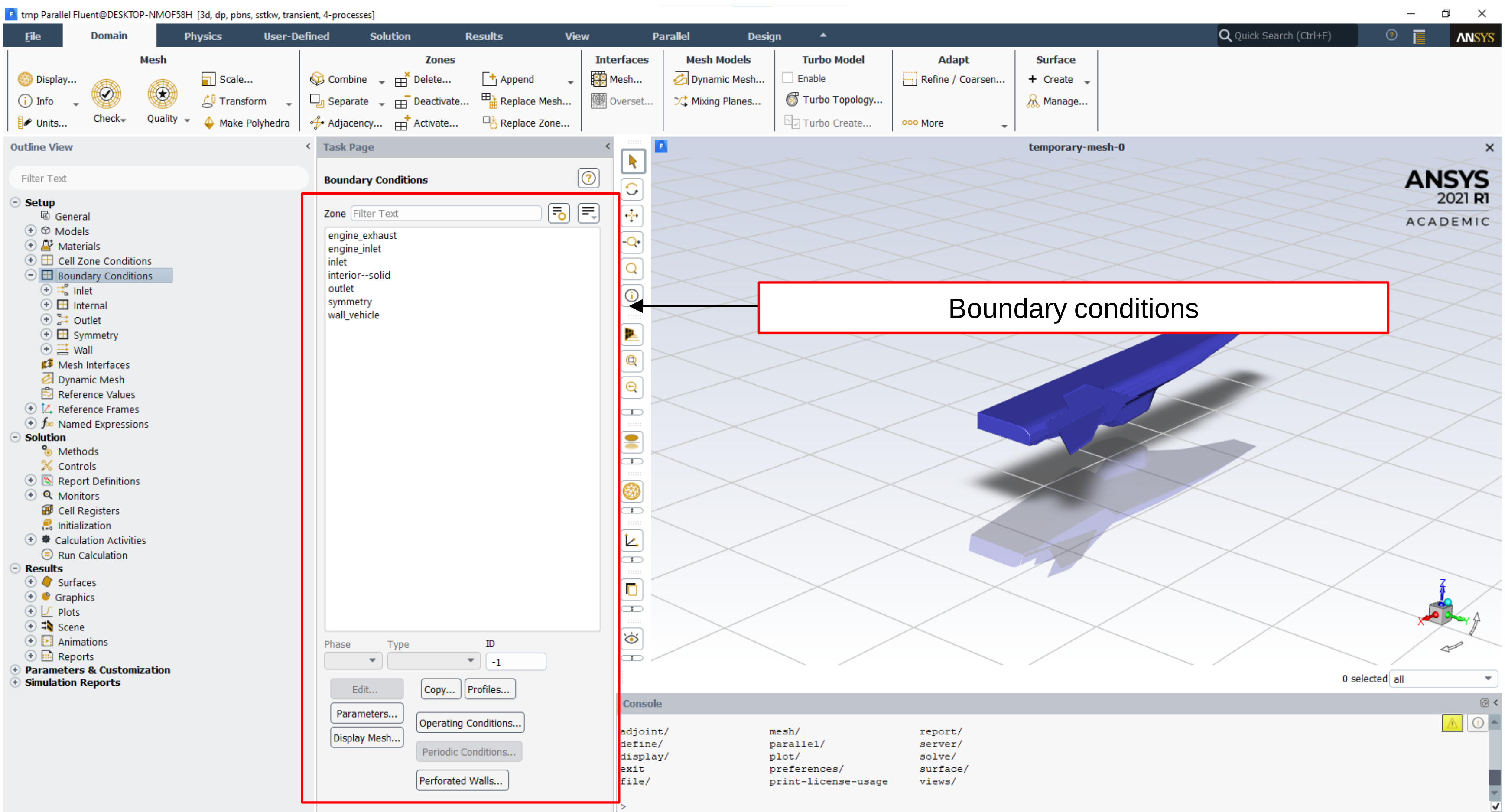
What is Ansys Fluent? – Executive summary

- Quick overview of Ansys Fluent GUI.



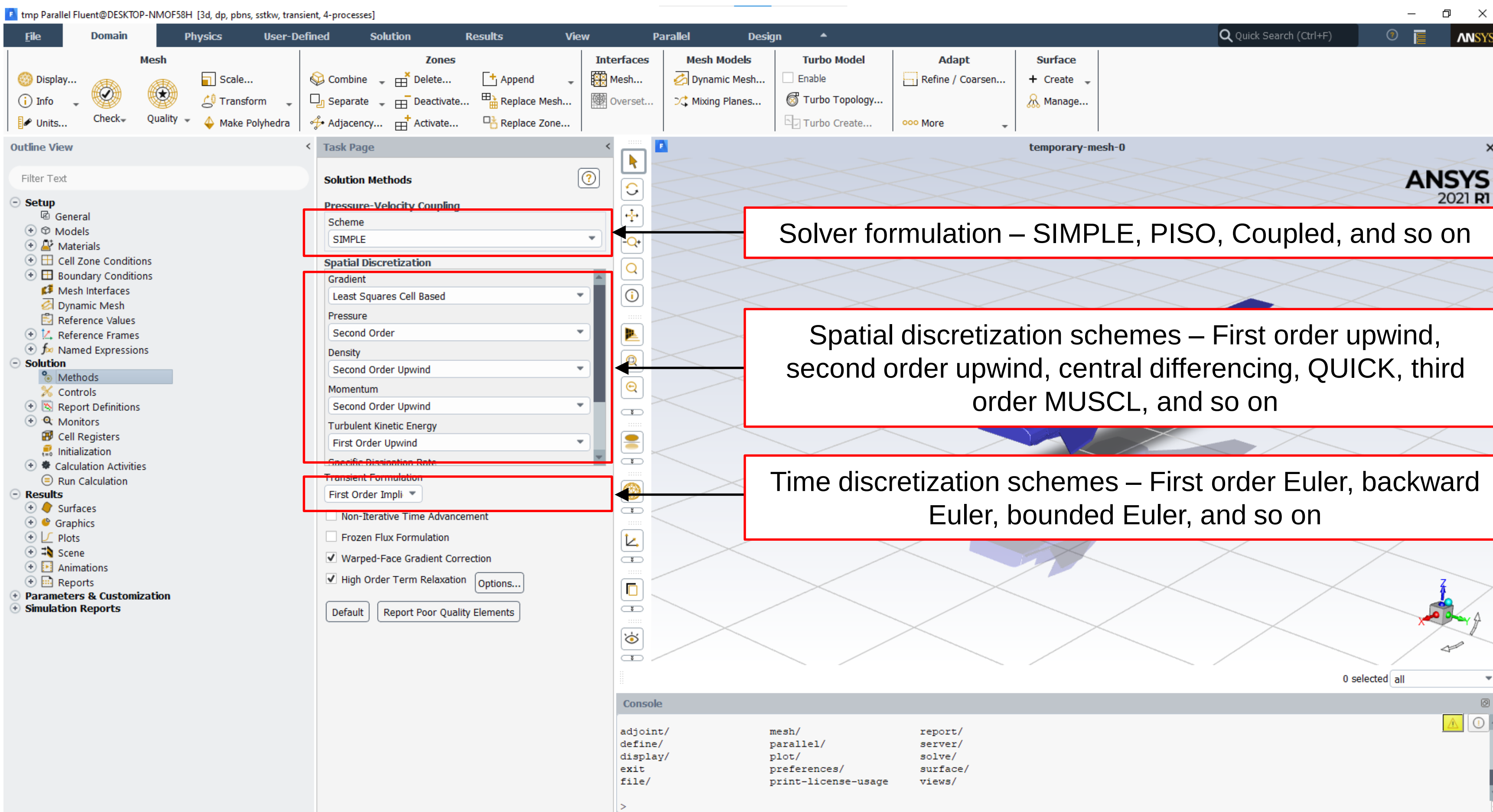
What is Ansys Fluent? – Executive summary

- Quick overview of Ansys Fluent GUI.



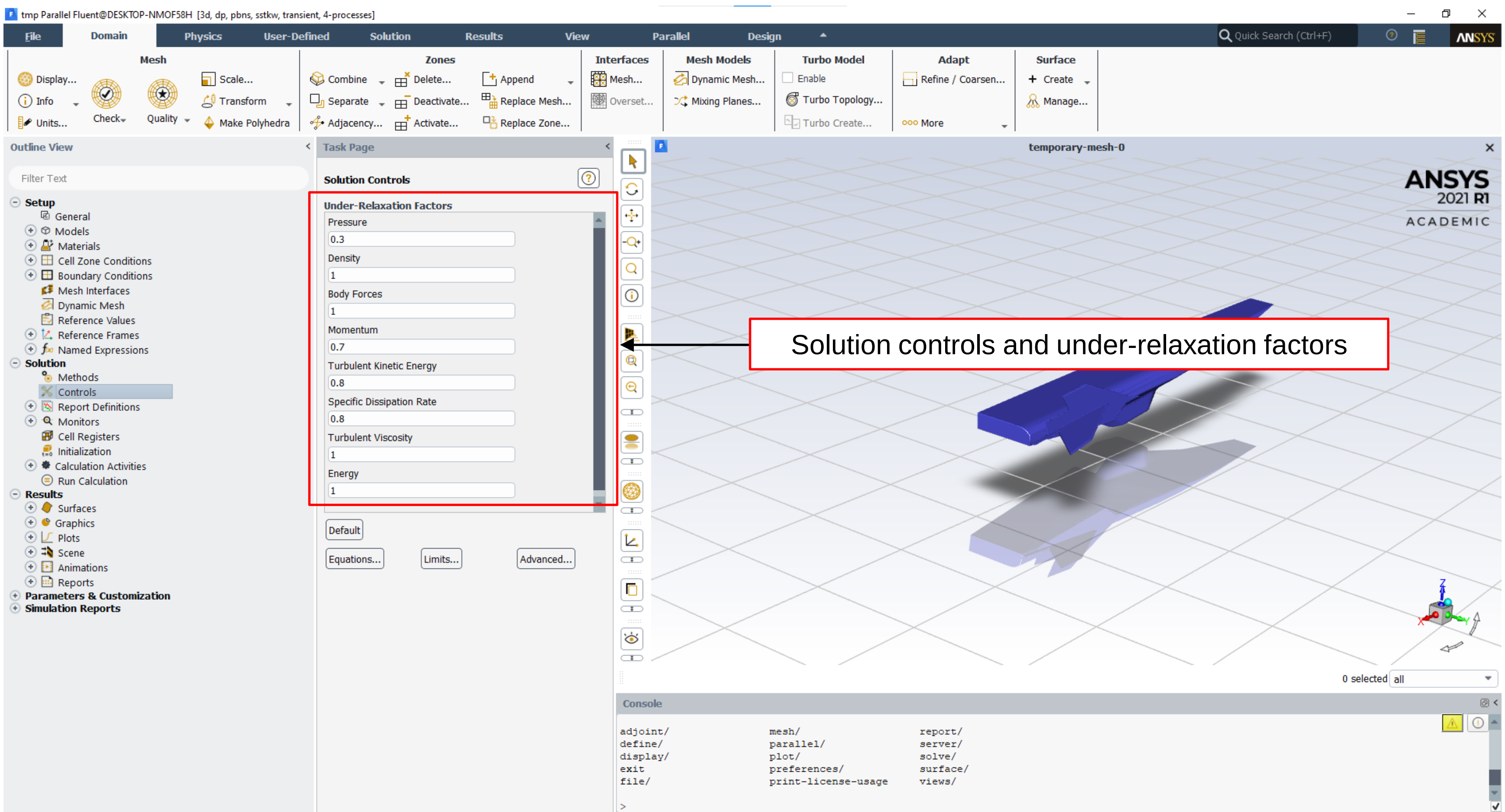
What is Ansys Fluent? – Executive summary

- Quick overview of Ansys Fluent GUI.



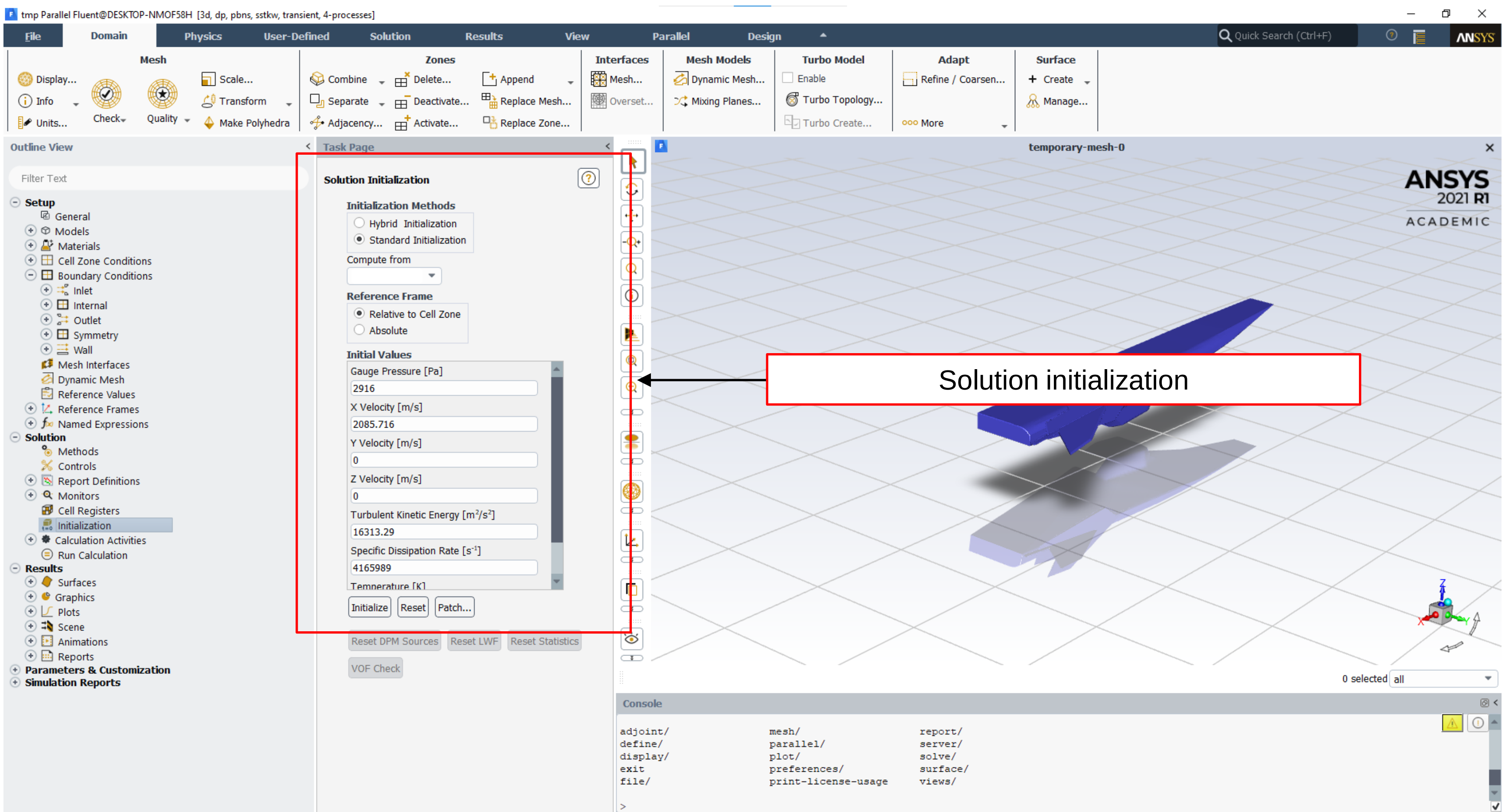
What is Ansys Fluent? – Executive summary

- Quick overview of Ansys Fluent GUI.



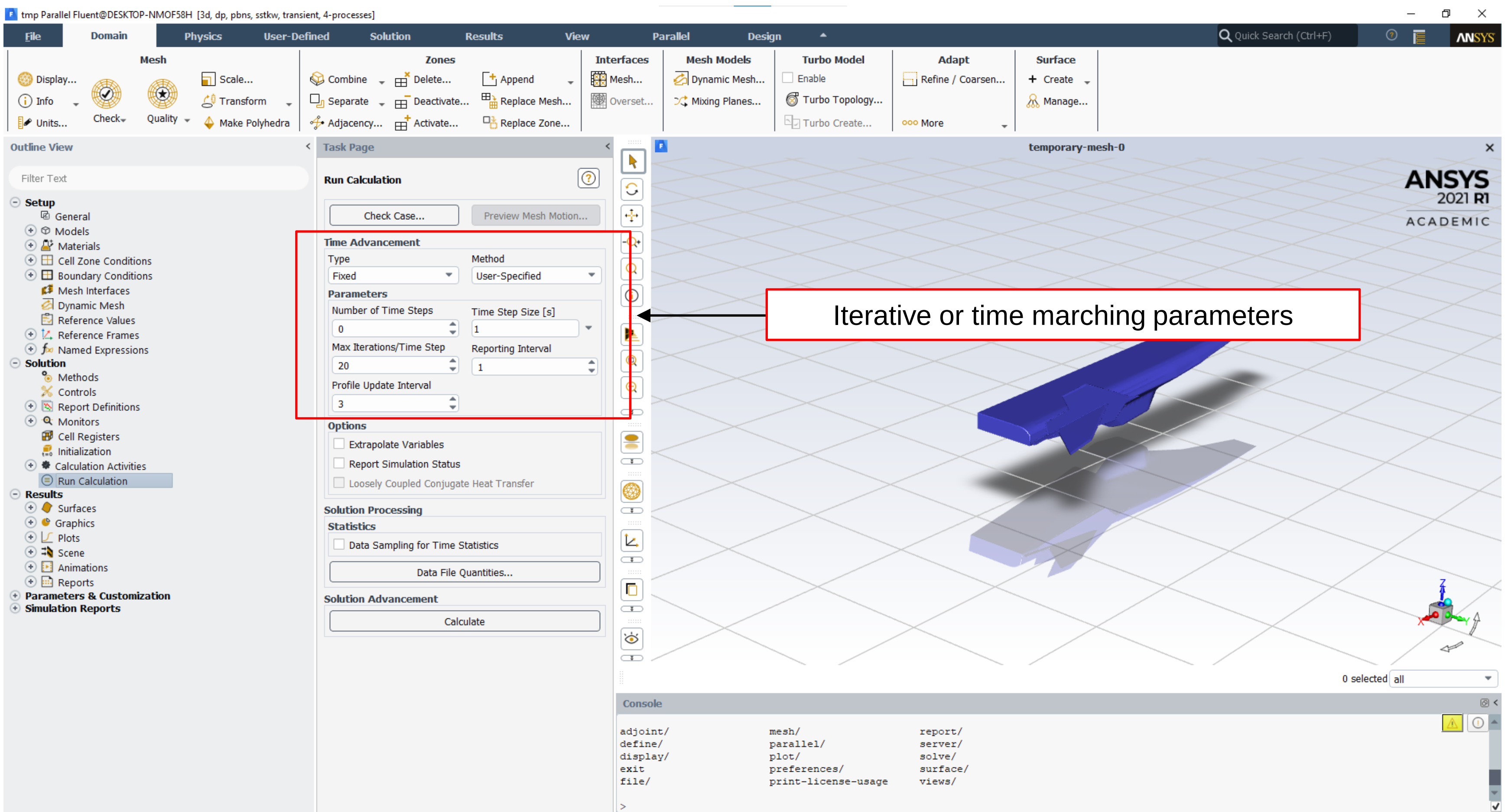
What is Ansys Fluent? – Executive summary

- Quick overview of Ansys Fluent GUI.



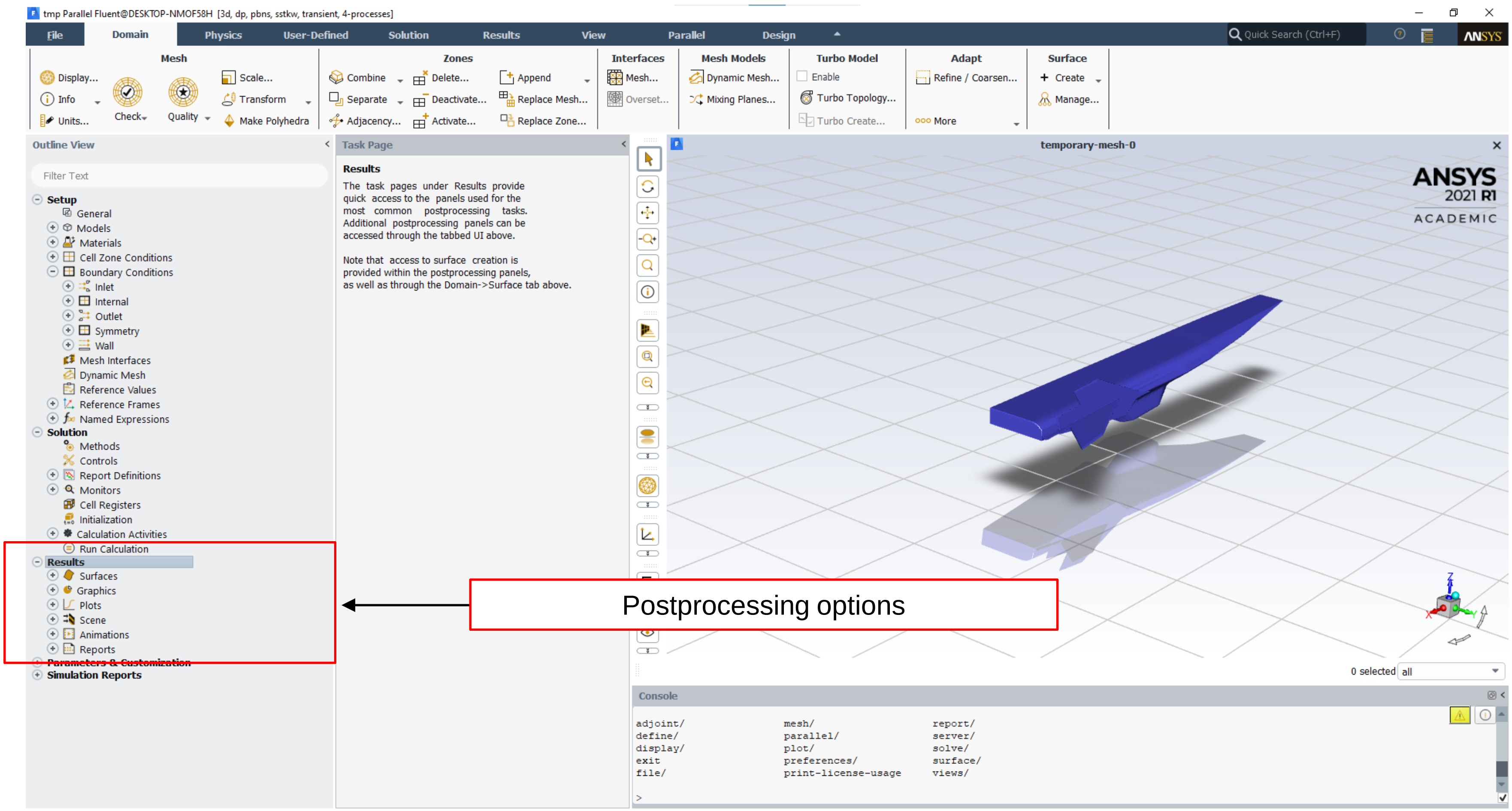
What is Ansys Fluent? – Executive summary

- Quick overview of Ansys Fluent GUI.



What is Ansys Fluent? – Executive summary

- Quick overview of Ansys Fluent GUI.



What is Ansys Fluent? – Executive summary

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