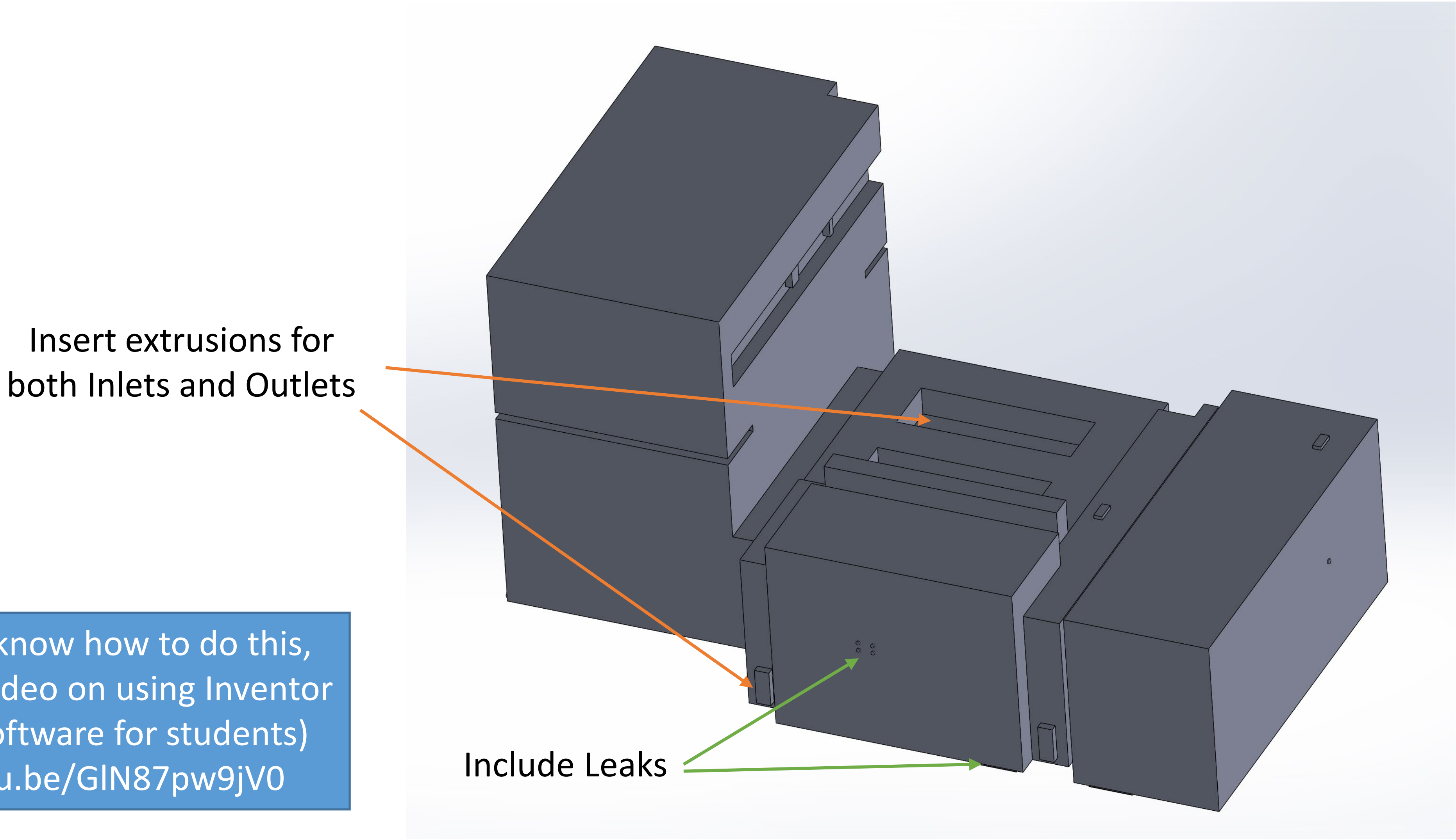
How To Model Your Lab's Airflow

(Total Simulation Time ~2 hours)

Create a 3D CAD model of lab internal volume



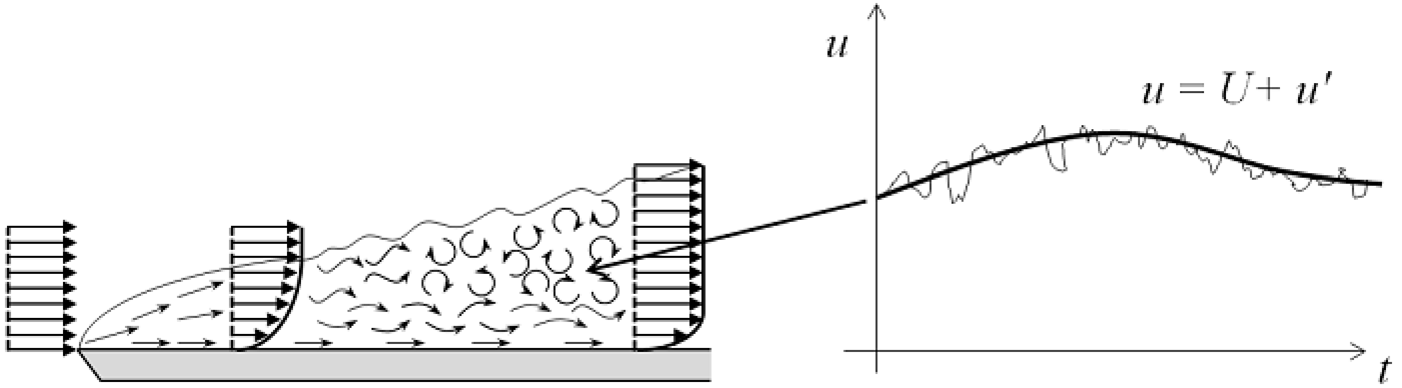
If you don't know how to do this, follow this video on using Inventor (Free CAD Software for students) https://youtu.be/GlN87pw9jV0

Verify airflow from each outlet/inlet

- Inlet flow rate dependent on return conductance.
- Include any high flow rate devices, such as HEPA flow hoods, laminar flow hoods, fans, chillers.

CFD Simulation assumptions used

- Isothermal ideal air
  - **(For low airflow rooms with high heat loads this is not valid)**
  - Segregated flow (Due to low speed flows)
- Steady state flow assumed
- Reynolds-Averaged Navier-Stokes
  - K-epsilon turbulence model



CFD Software Options

Shown Here

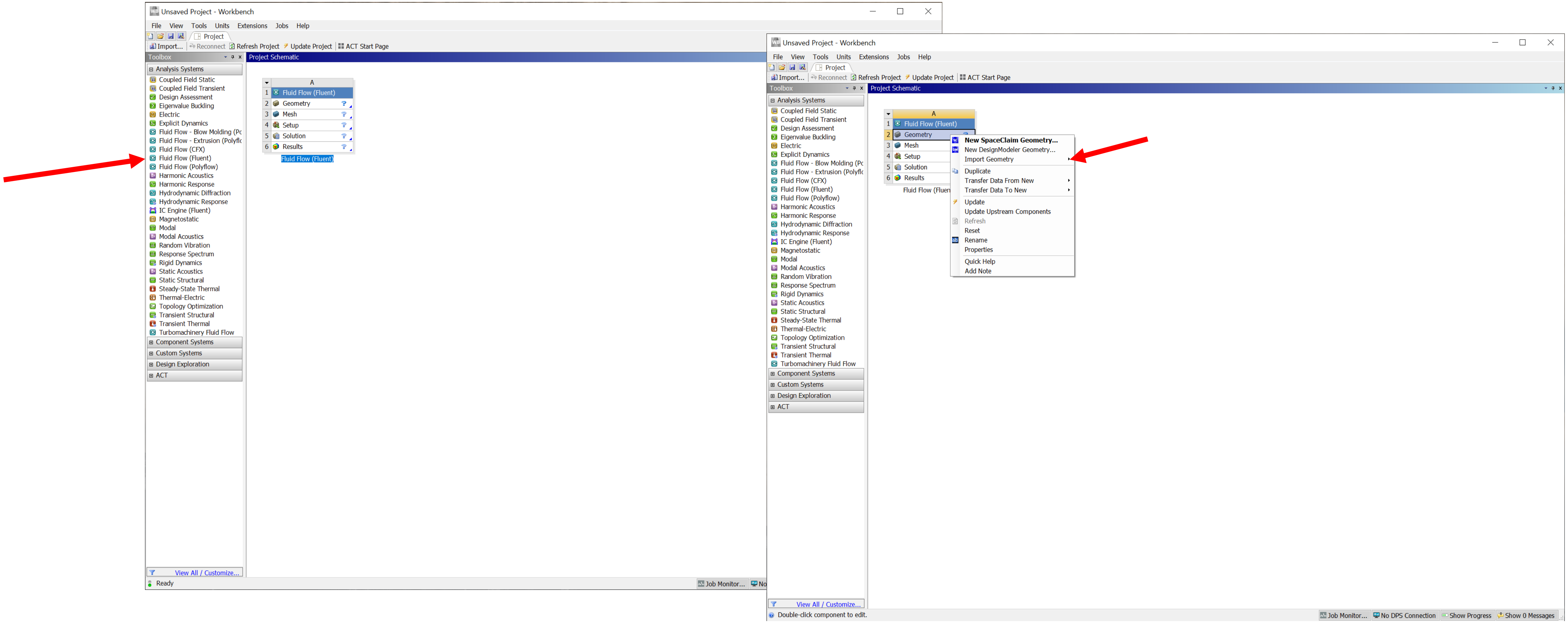
- Ansys Fluent – Free Student Version Available

https://www.ansys.com/academic/free-student-products

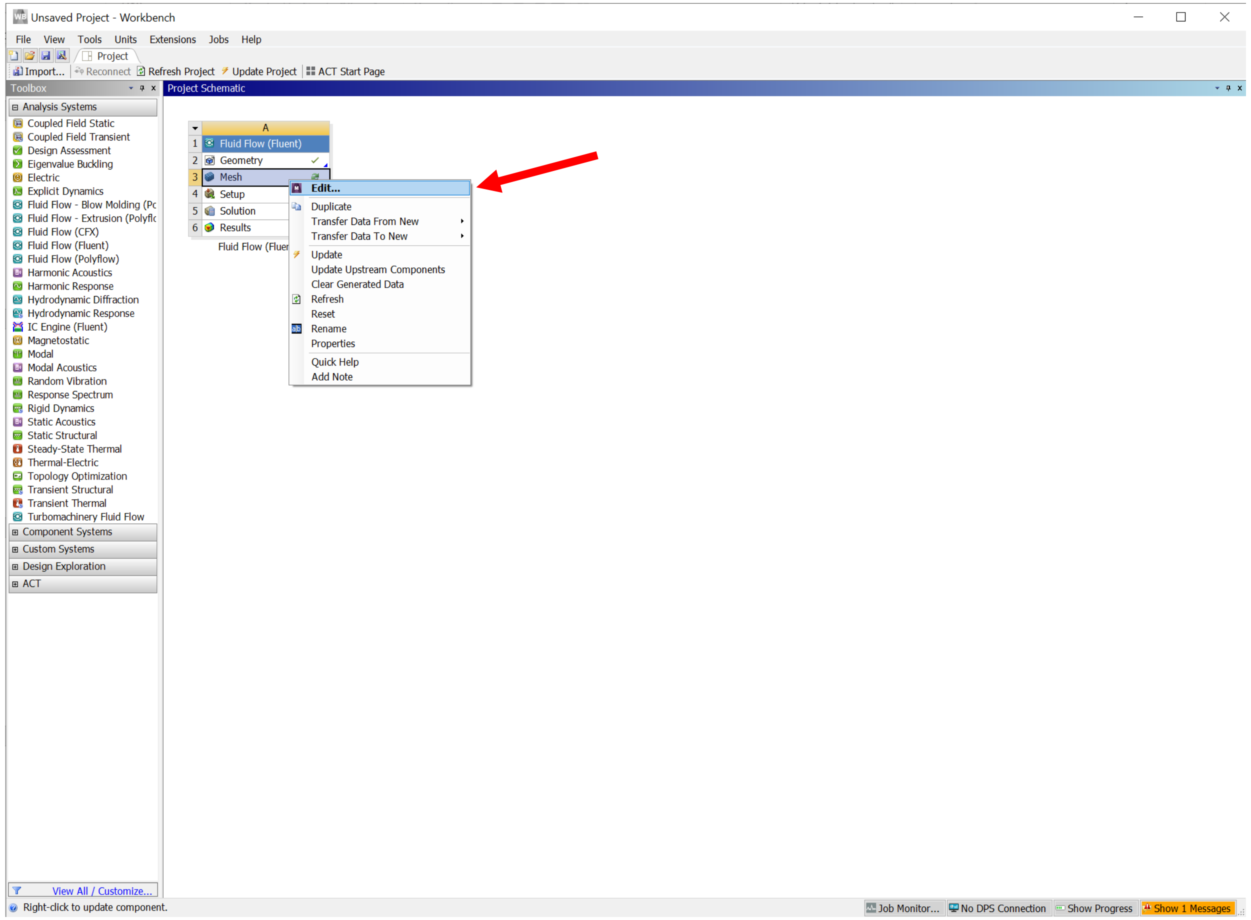
Other Options:

- Siemens Star CCM+
- COMSOL Multiphysics

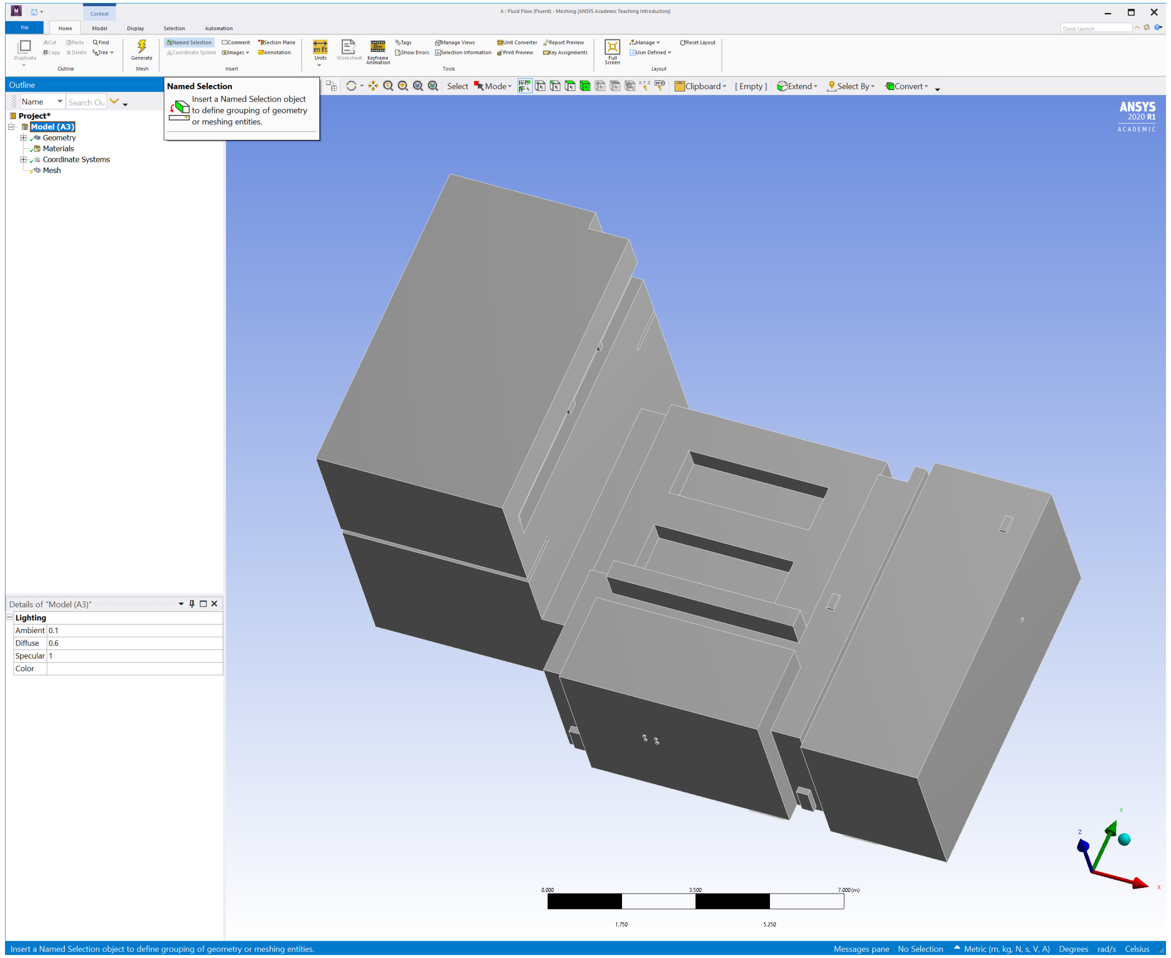
Click Fluid Flow (Fluent), Import Geometry

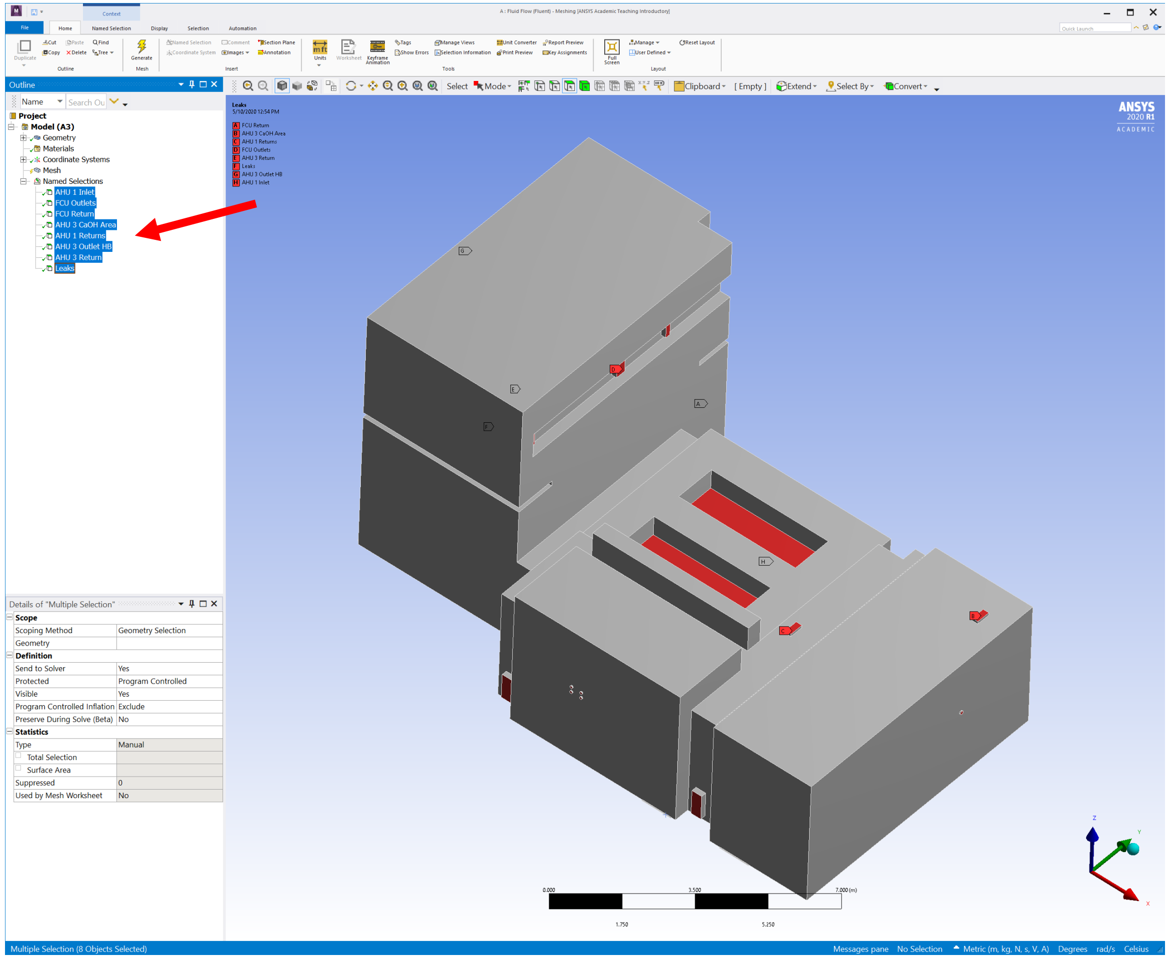


Open Mesh Editor

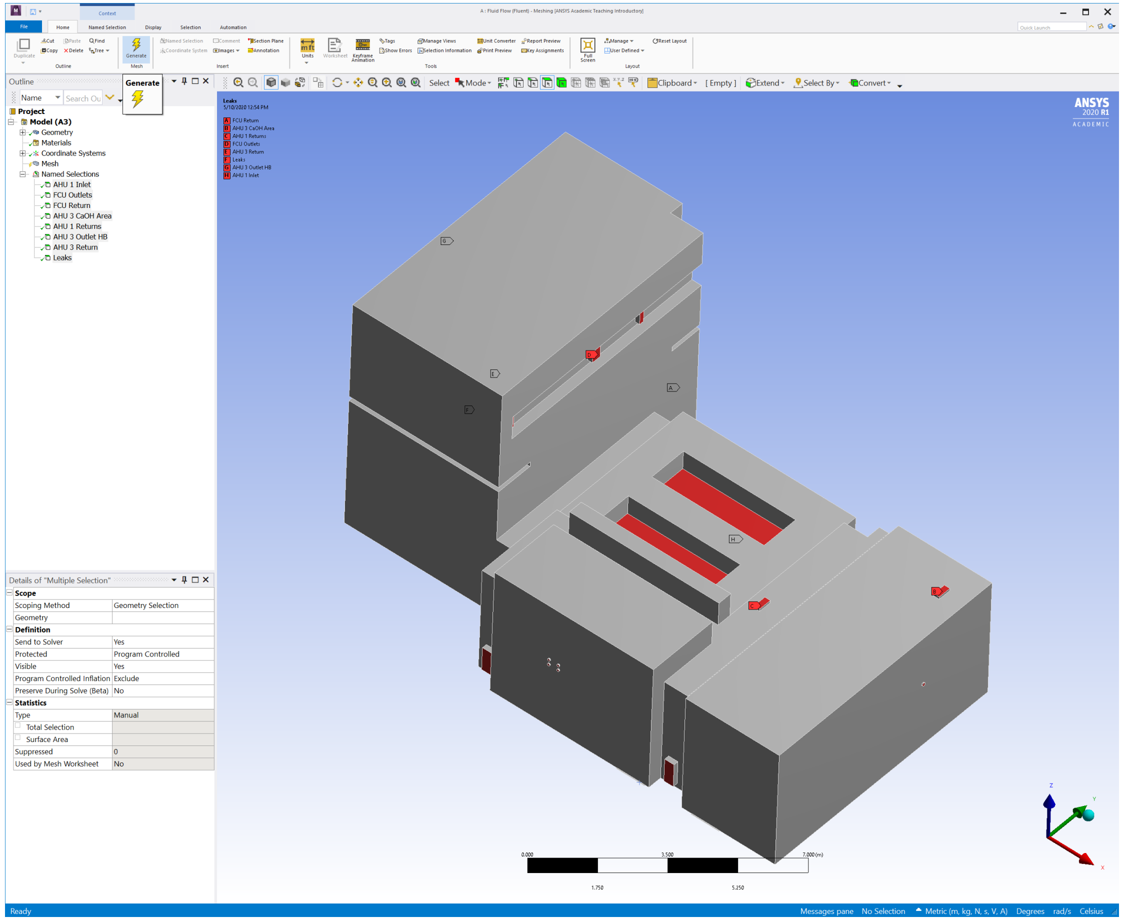


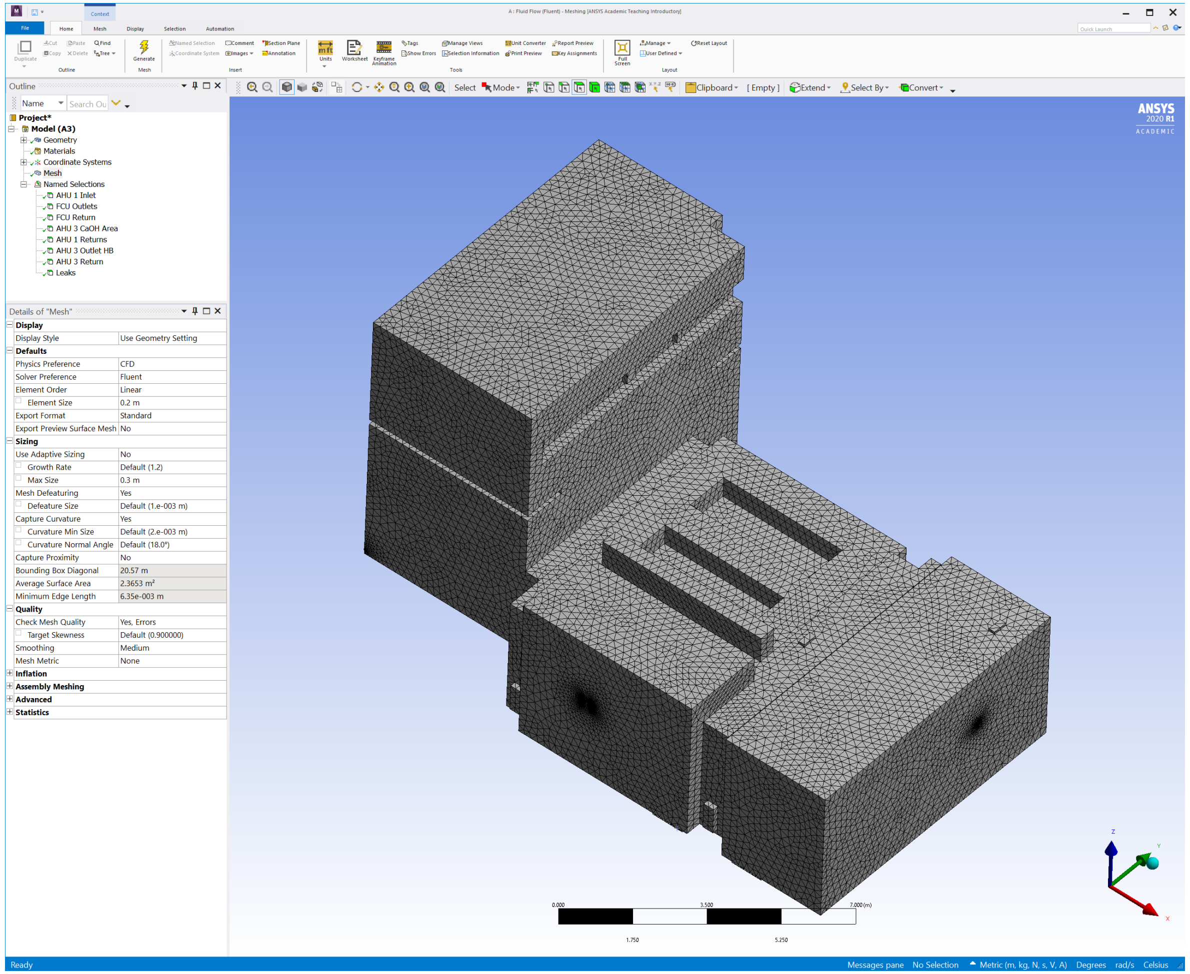
Add named selections to each inlet/outlet



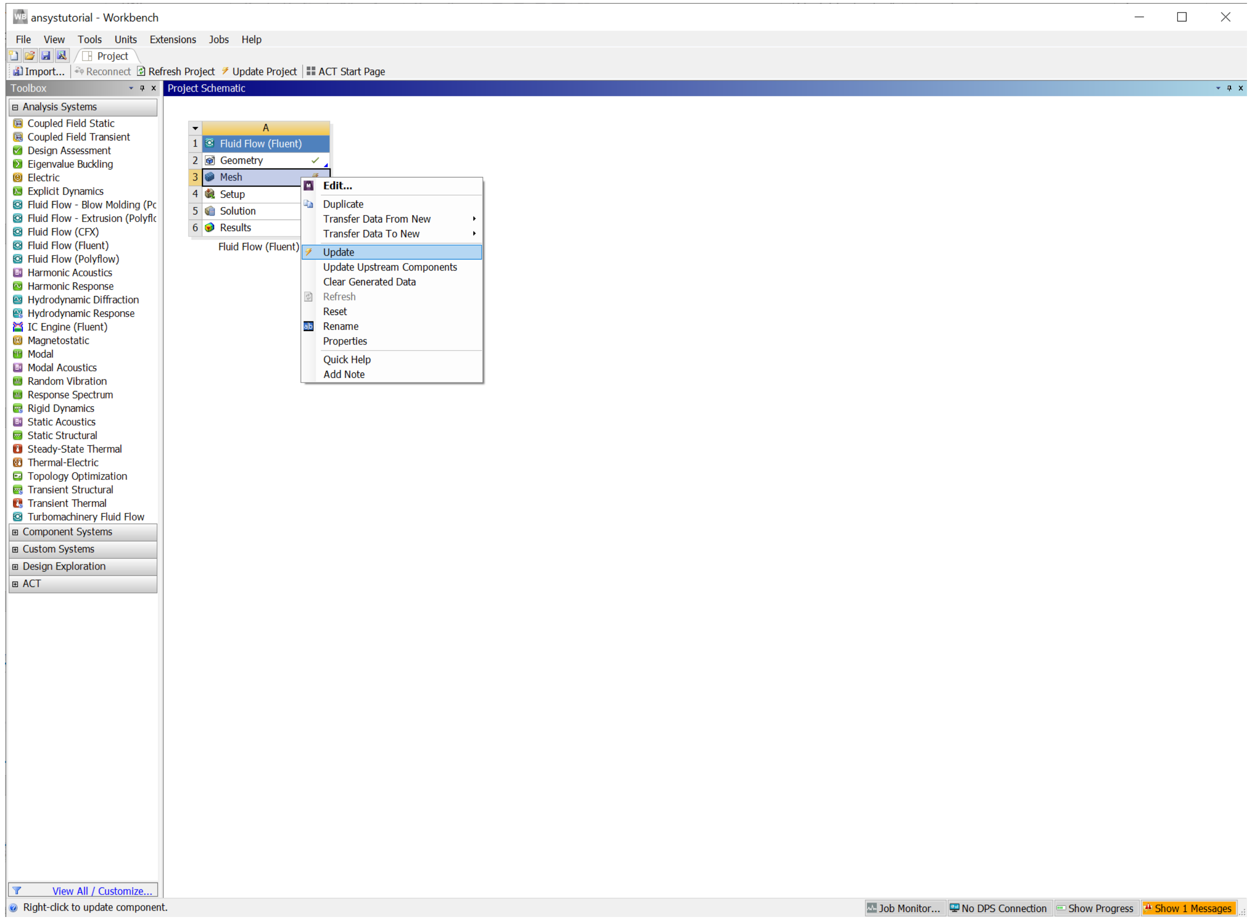


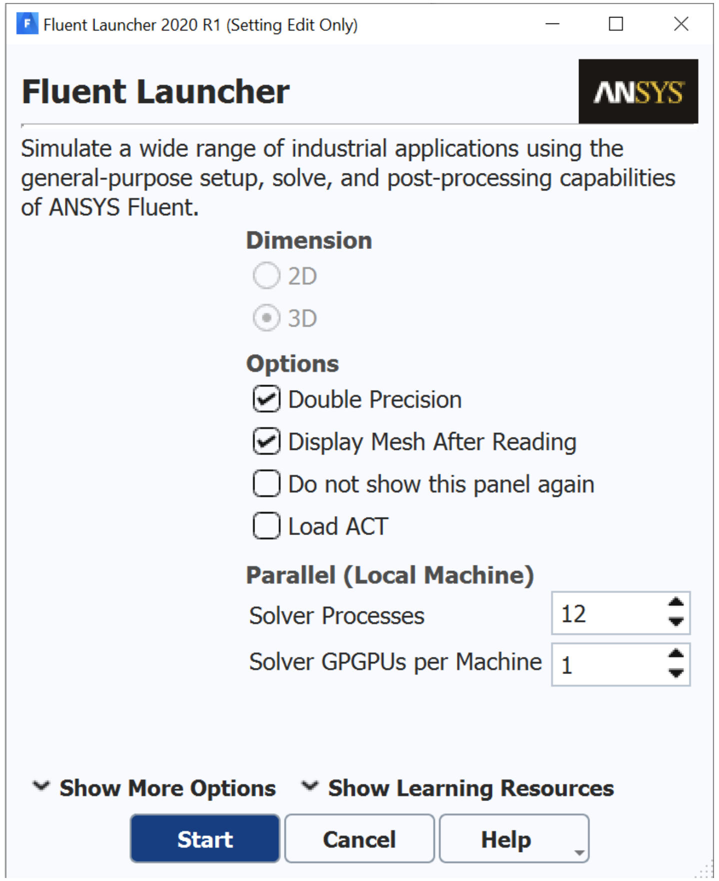
Generate Mesh – Adjust mesh to desired level



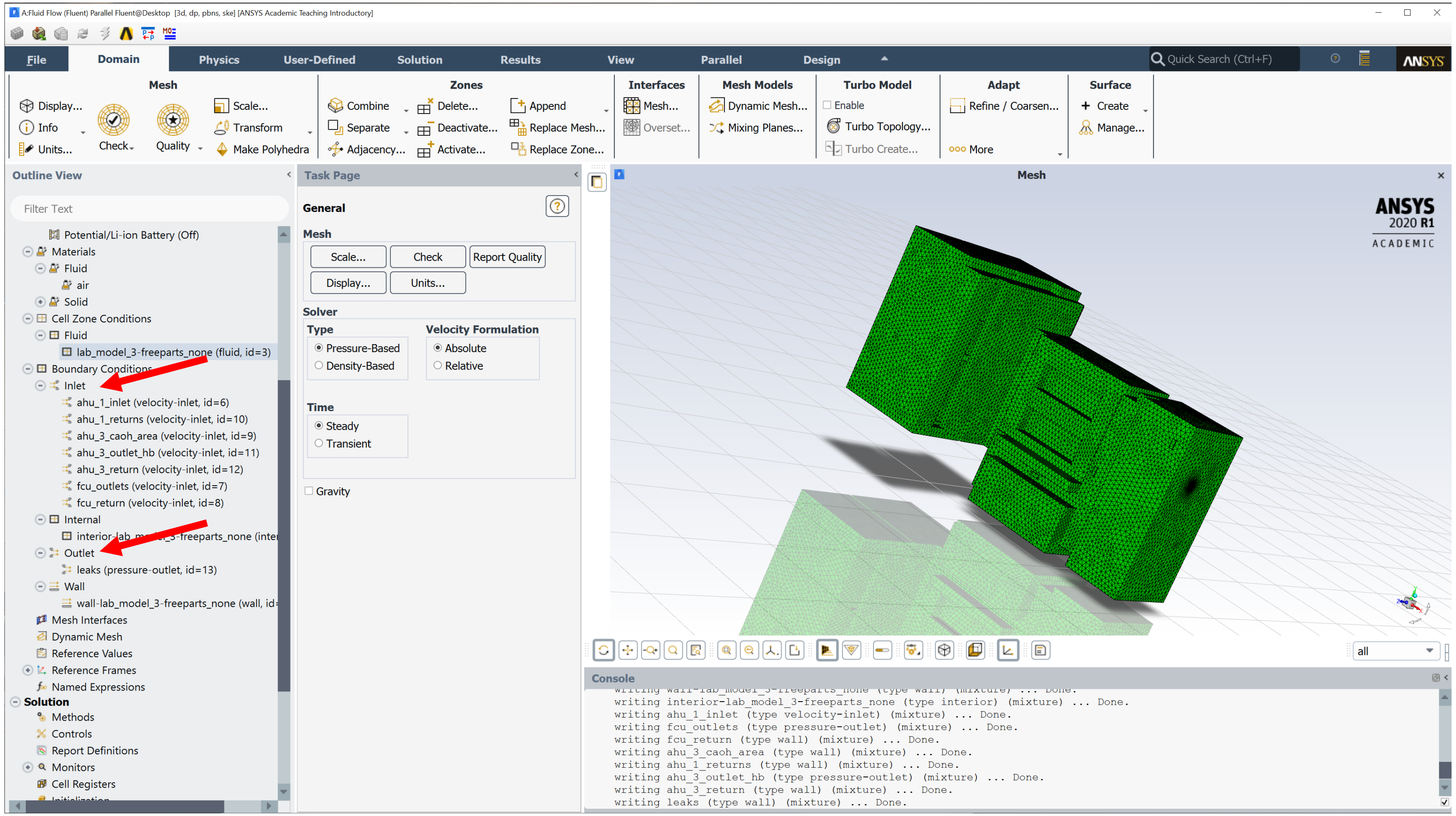


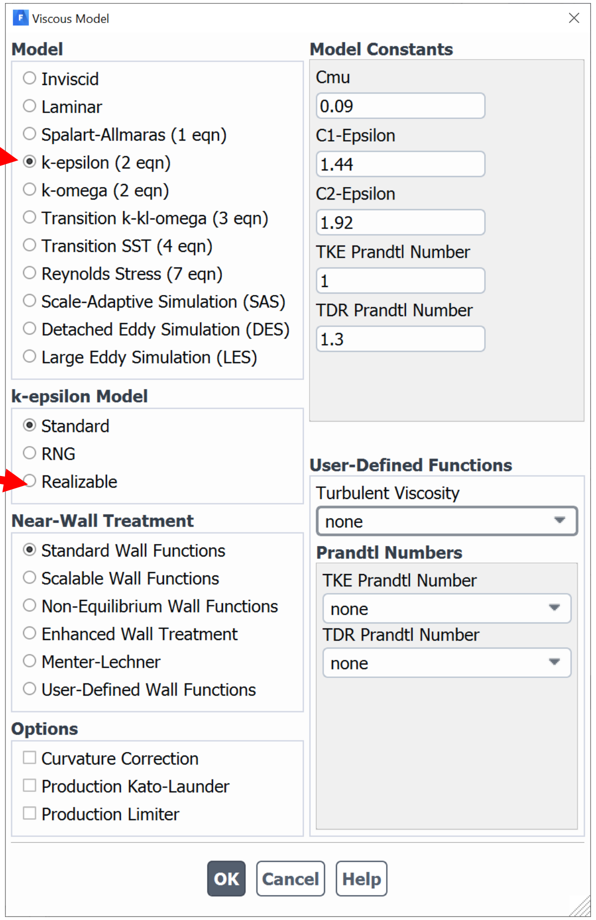
Update Mesh and Launch Fluent



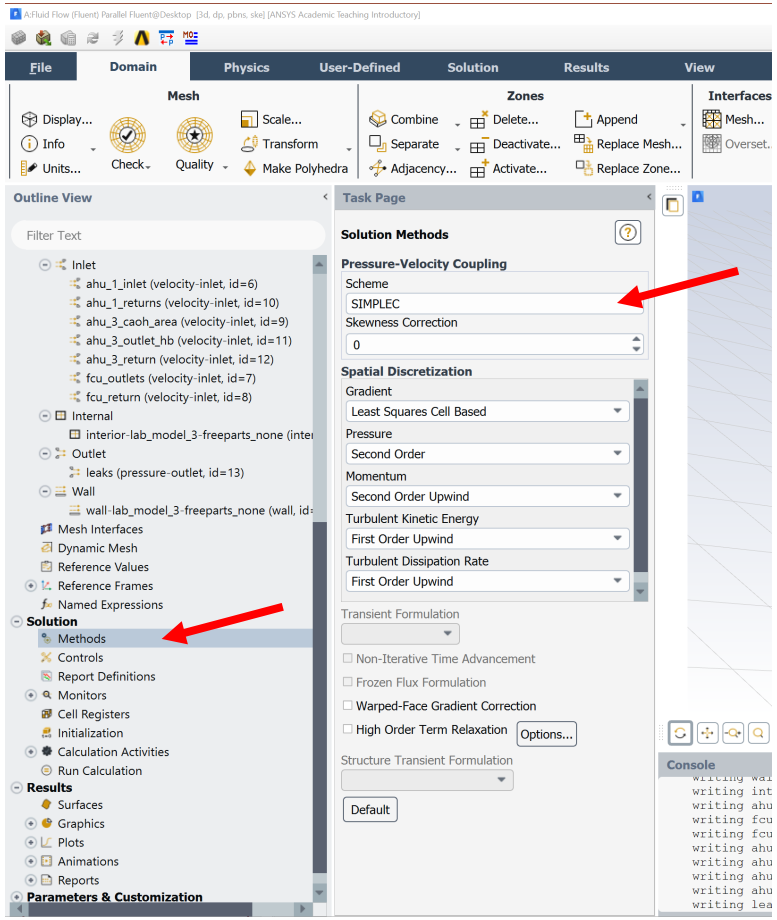


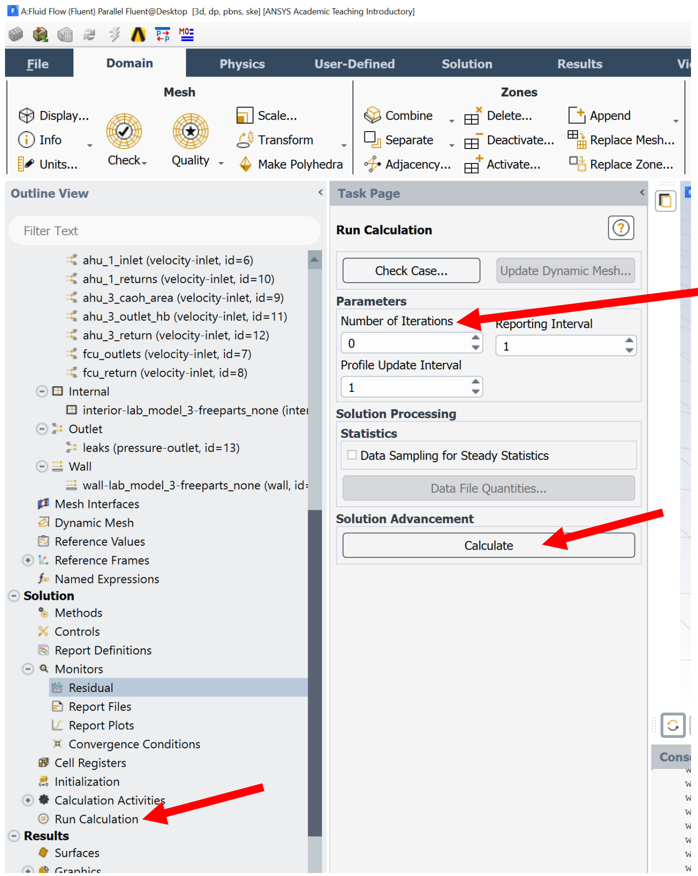
Select velocity inlets for both inlets and outlets and set velocities
Leaks are left as pressure outlets
Select the k-epsilon turbulence model in the viscous model settings





Set segregated flow and Run





Set number of iterations to reach desired stopping limit

Simulations will take about ~1 hour to run on a desktop

General notes about attaining convergence

- Ensure mesh resolution is fine enough in “high” velocity areas (>0.5m/s)
- Ensure mass conservation is attained
- Residuals <~0.001 usually indicate reasonable convergence
- If non converging, stop and plot results. Use areas of unphysical results as indications of possible problems with the model

Once Convergence Reached, Plot Results