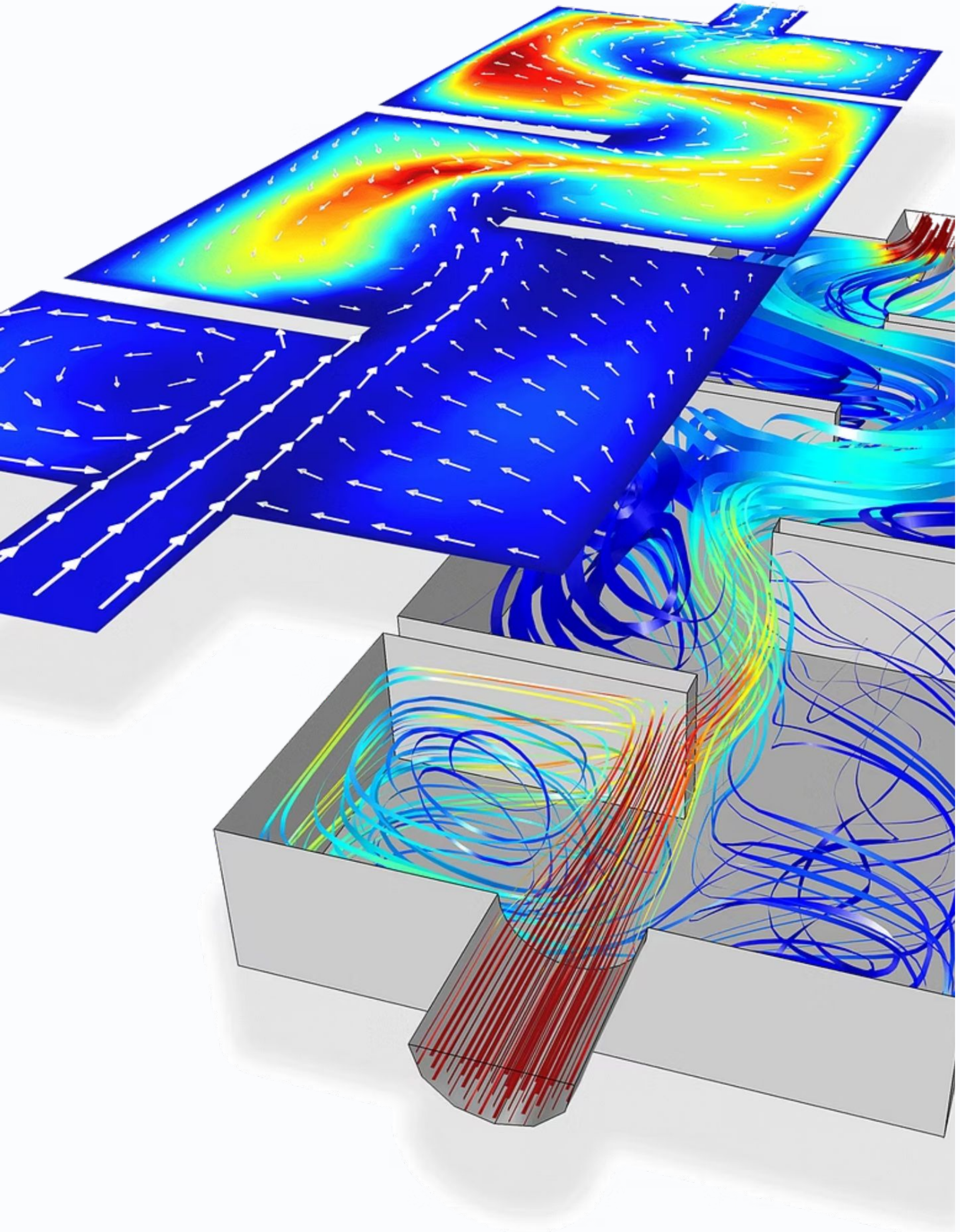


CFD Best Practices for Engineering Design

A comprehensive guide to executing accurate, reliable, and actionable CFD simulations—from physics definition through design optimization and engineering decisions.

1. Project Framing & Physics Definition

Establishing clear simulation objectives and physics fundamentals ensures CFD delivers actionable engineering insights rather than decorative visualizations.

Define the Simulation Objective

Critical Question

State the physical question CFD must answer with quantifiable outputs that directly inform design decisions.

Example Objective

Estimate pressure drop and velocity uniformity in a 6-tube heat exchanger header operating at 0.2 kg/s mass flow rate.

Why This Matters

Without a clear goal, CFD becomes a post-processing exercise that consumes resources without delivering design value. Every simulation must answer: "What decision does this enable?"

Common Mistakes

- Vague goals like "run CFD to see flow"
- No quantifiable success metrics
- Missing link between results and design decisions

Identify Flow Regime & Physics

Reynolds Number

Calculate Re to determine laminar, transitional, or turbulent regime—this drives all downstream modeling choices.

Time Dependency

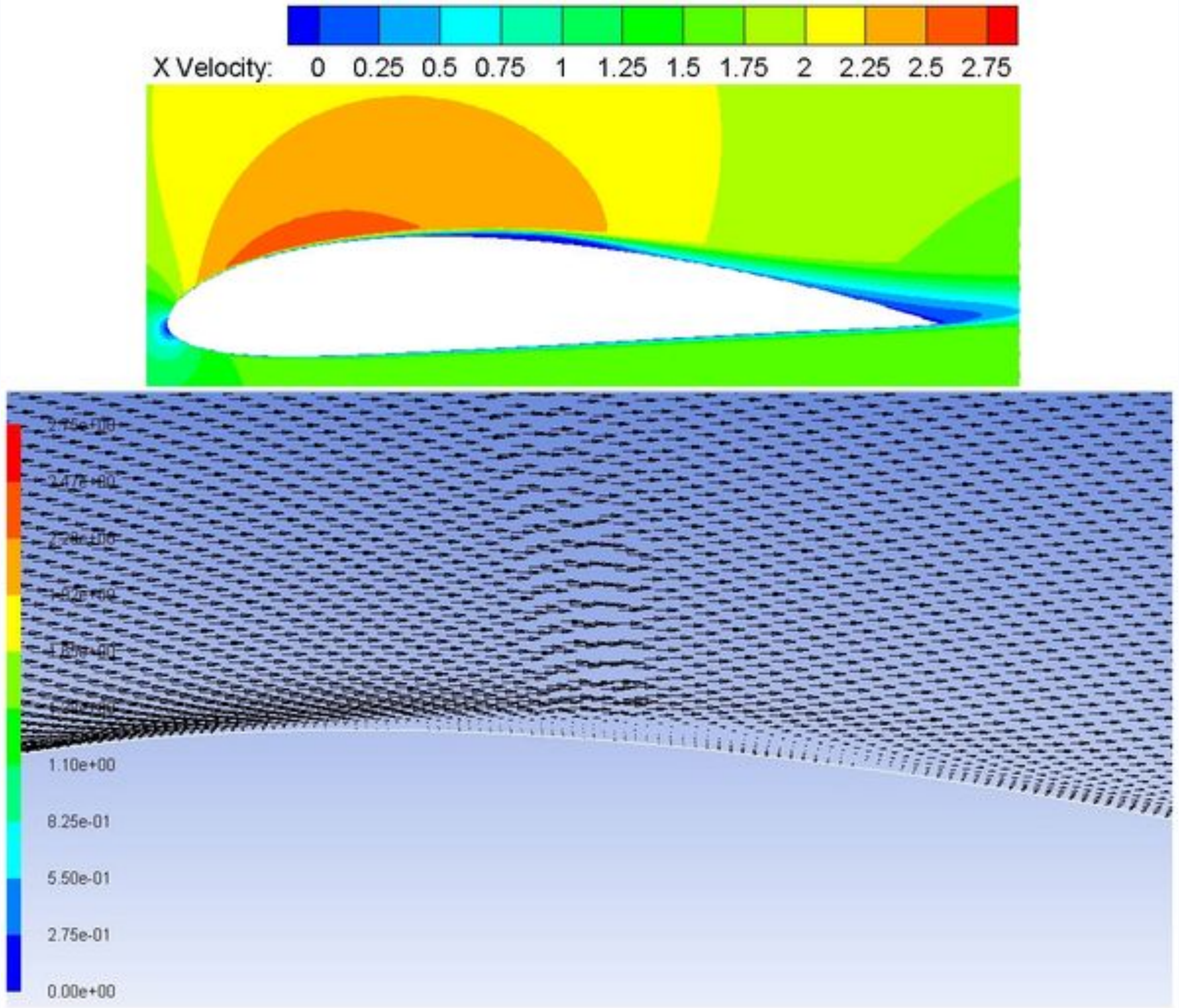
Identify whether flow is steady-state or requires transient analysis for vortex shedding or periodic behavior.

Compressibility

Assess Mach number to determine if density variations must be resolved explicitly in the governing equations.

Multiphase Nature

Determine if multiple fluid phases, free surfaces, or dispersed particles require specialized modeling approaches.



Establish Key Outputs & Success Criteria

Define Measurable Metrics

- Identify specific engineering parameters to extract : pressure drop, heat transfer coefficient, drag coefficient, mixing index, or flow uniformity.
- These quantities must be monitored during solution convergence and validated against design requirements or experimental benchmarks.

Set Acceptance Thresholds

- Establish numerical targets that define simulation success and guide iterative refinement of geometry or operating conditions.

<1 kPa

Target Pressure Drop

Maximum allowable at 0.2 kg/s flow rate

±5%

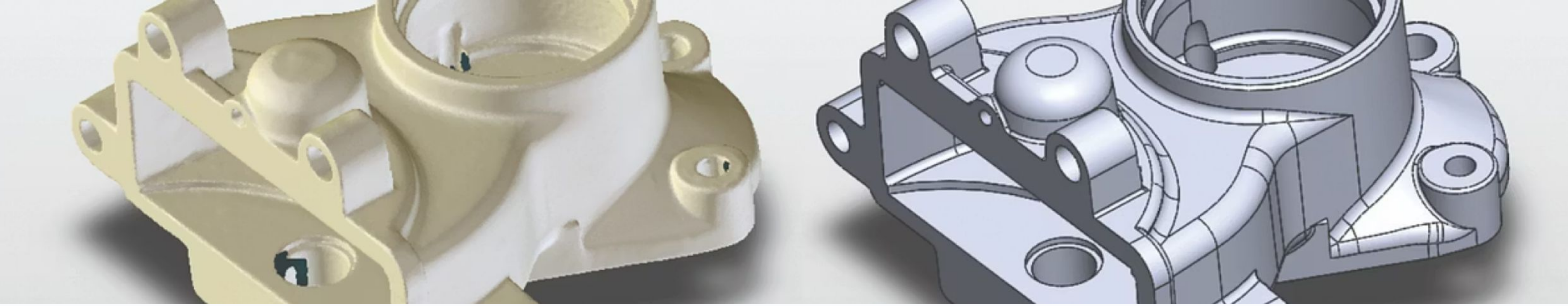
Velocity Uniformity

Across outlet plane cross-section

10^{-4}

Convergence Criteria

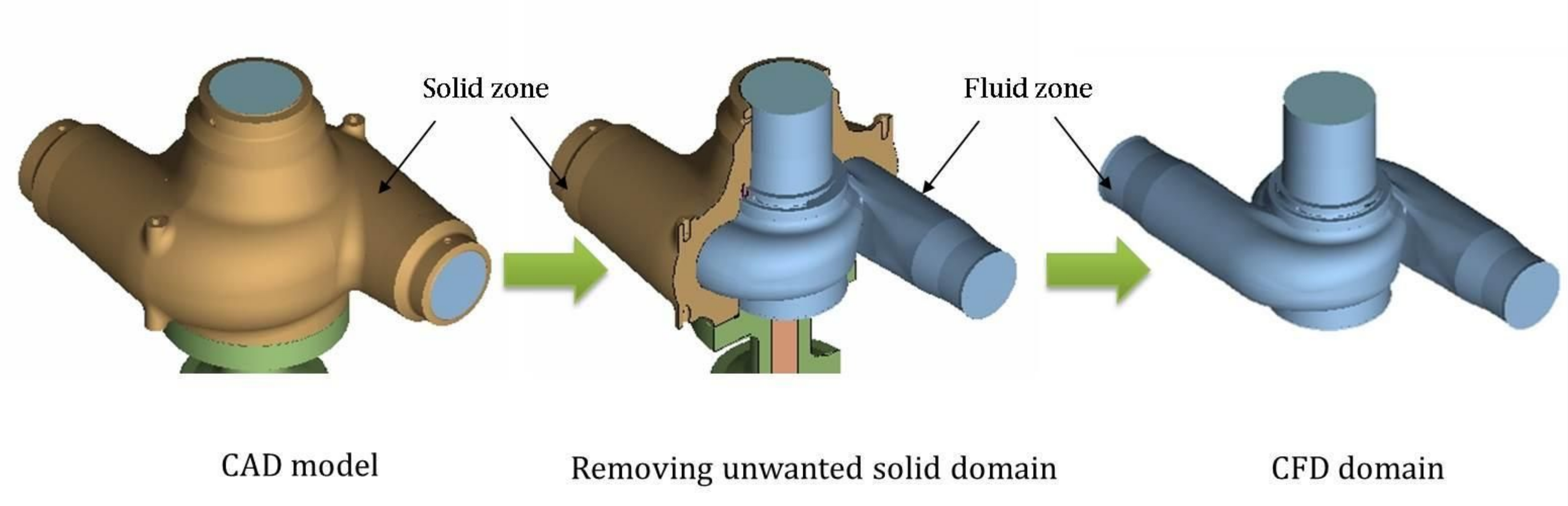
Residual threshold with stable monitors



2. Geometry Preparation & Domain Definition

Proper geometry preparation is foundational—a watertight, simplified fluid domain prevents convergence failures and ensures physically meaningful results.

CAD Cleanup



01

Remove Gaps & Overlaps

Eliminate surface discontinuities smaller than 0.05 mm that prevent volume extraction. Use Boolean operations to create seamless interfaces.

02

Delete Non-Fluid Components

Remove bolts, mounting brackets, and structural elements that occupy space but don't affect flow physics.

03

Verify Watertight Volume

Confirm closed surfaces with consistent normal vectors pointing into the fluid domain—critical for successful meshing.

> **Common mistake:** Importing unsealed CAD solids leads to negative volumes and mesh generation failures that consume hours of troubleshooting.

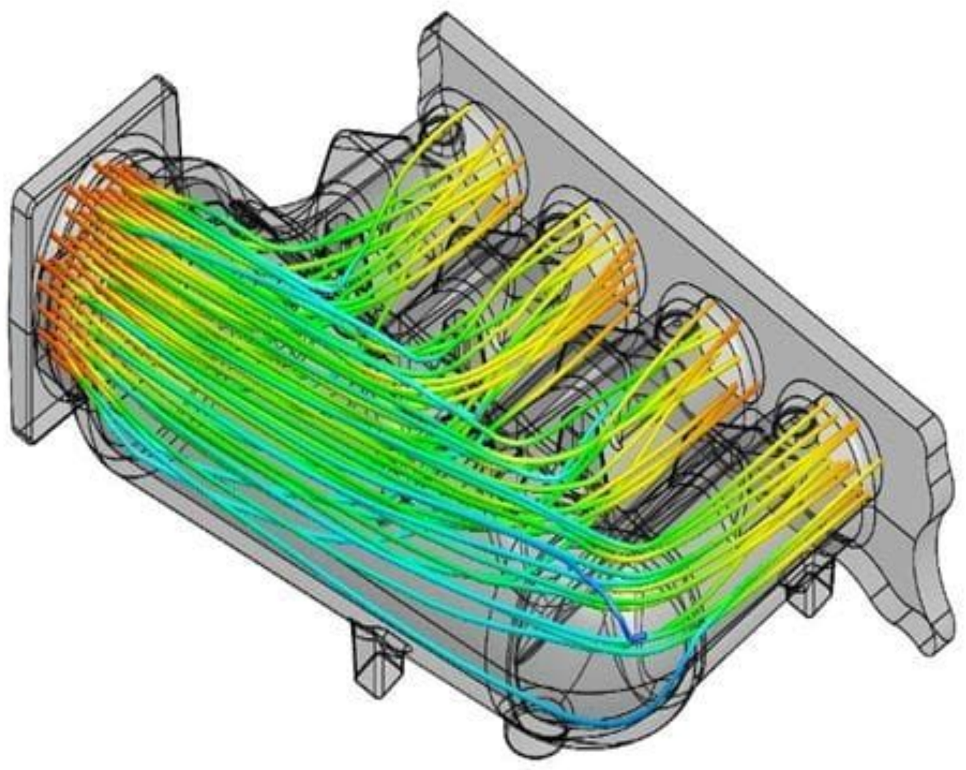
Flow Domain Extraction

Subtract Solid Boundaries

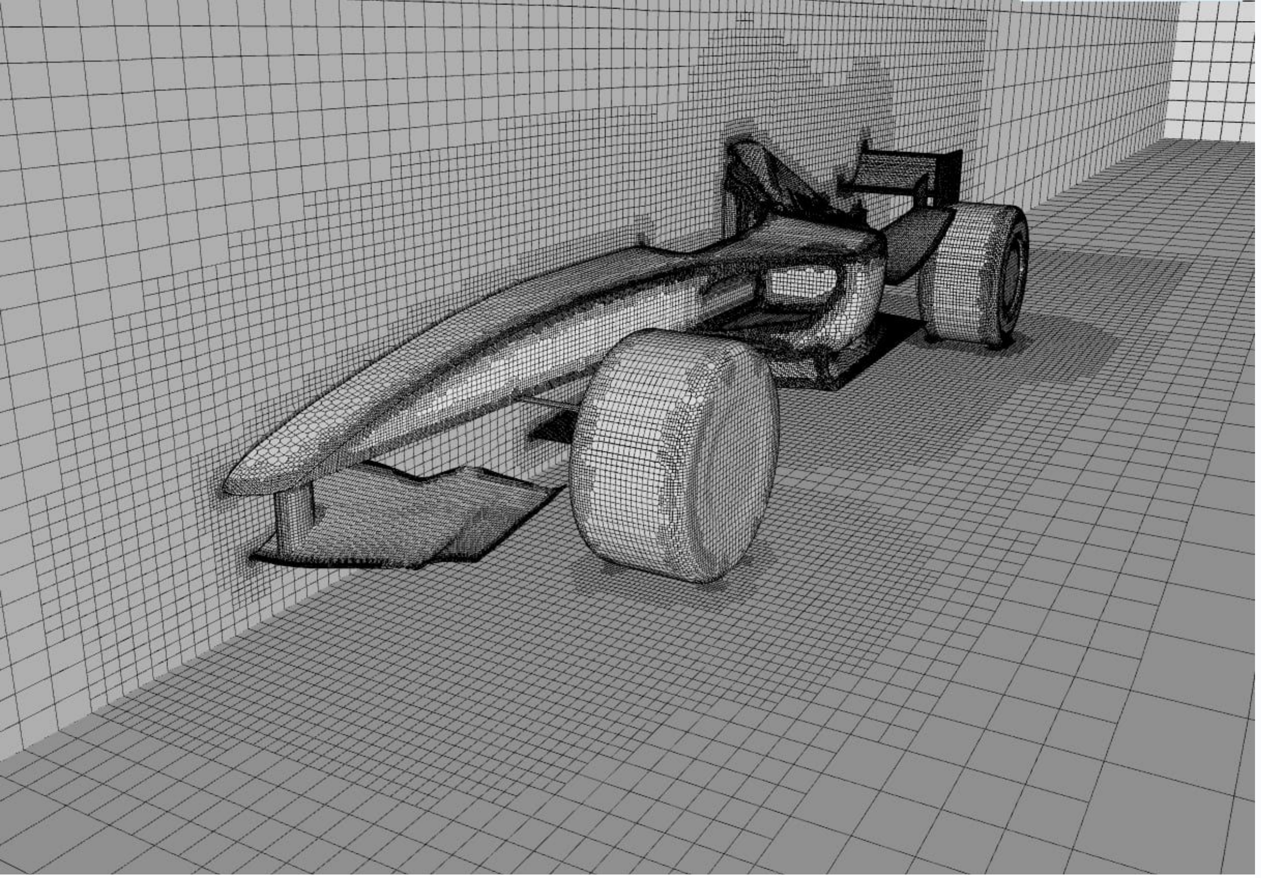
Create the fluid region by performing Boolean subtraction—subtract pipe walls, housing components, and all solid parts from a bounding volume that encompasses the flow path.

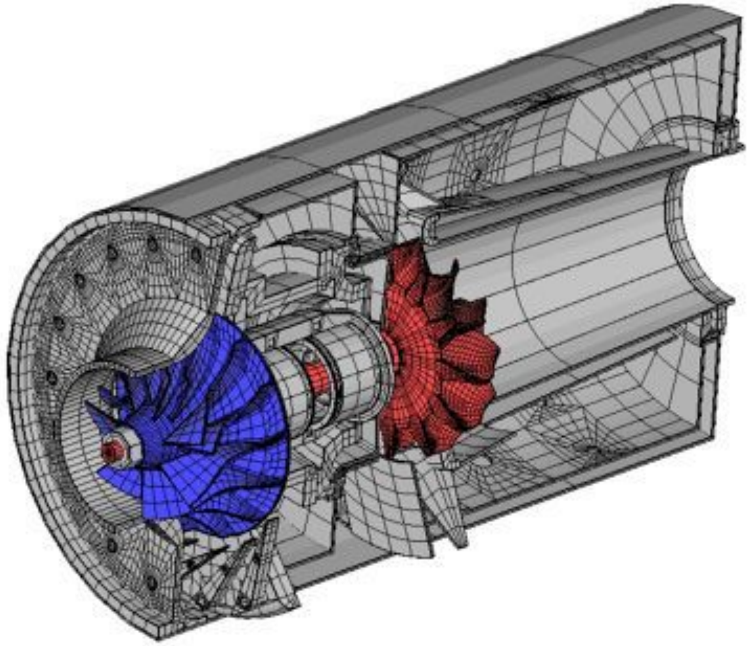
Validate Domain Integrity

- Check for open ends that cause non-physical leaks
- Verify inlet and outlet faces are properly defined
- Ensure no isolated fluid pockets remain



Example: For an internal manifold, subtract all walls and baffles to obtain the continuous fluid volume where flow equations will be solved.





3. Mesh Generation & Quality Control

Mesh quality directly determines solution accuracy—poor meshing causes numerical diffusion, false convergence, and unreliable predictions regardless of physics model sophistication.

Mesh Type Selection

Structured/Hexahedral

Ideal for simple geometries. Provides superior accuracy with fewer cells but requires significant setup effort for complex shapes.

Unstructured/Tetrahedral

Automatic generation for complex geometry. Requires more cells for equivalent accuracy—avoid for boundary-layer-dominated flows without prism layers.

Polyhedral

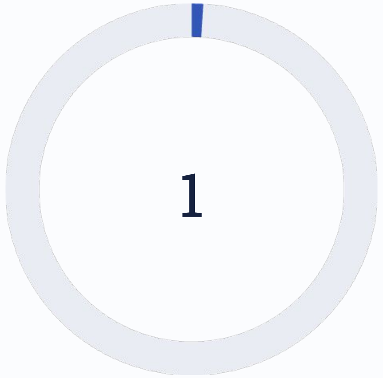
Balances automation with accuracy. Reduces cell count by 30-50% versus tetrahedral while maintaining solution quality for industrial applications.

Recommended approach: Hexahedral core with prismatic inflation layers at walls combines efficiency with accurate boundary layer resolution.

Boundary Layer Meshing

Y+ Control Strategy

The dimensionless wall distance Y+ determines whether viscous sublayer is resolved or modeled via wall functions—critical for accurate wall shear stress and heat transfer.

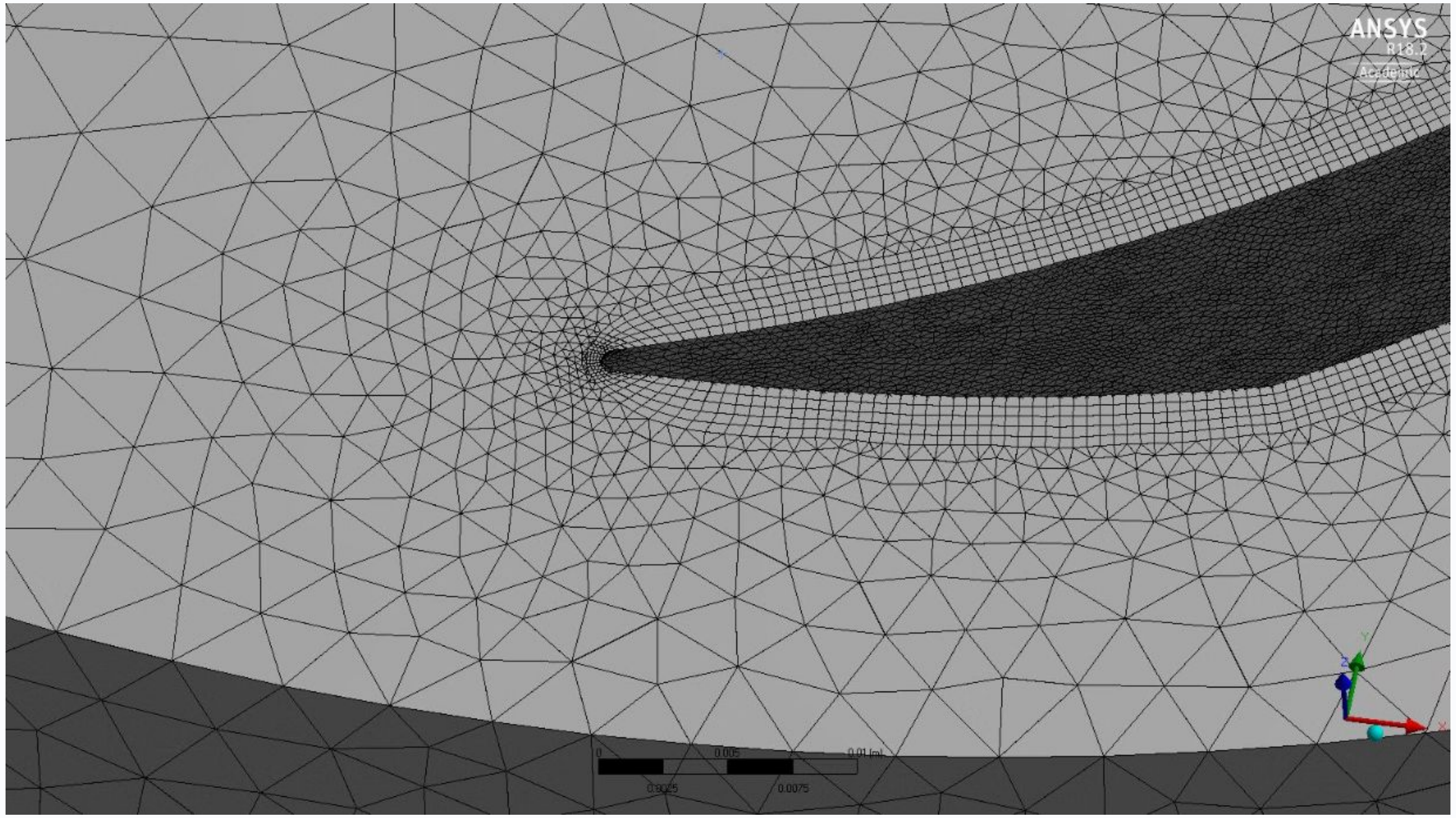


Low-Re Models

k-ω SST, Spalart-Allmaras

Wall Functions

Standard k-ε models



Prism Layer Requirements

1. Minimum 5 layers for gradient resolution
2. Growth ratio ≤ 1.2 for smooth transitions
3. First cell height computed from target Y+
4. Consistent resolution across all walls

> Common mistake: Single inflation layer or Y+ mismatch with turbulence model yields incorrect friction factors and heat transfer coefficients.

Mesh Independence Study

Coarse Mesh

Baseline: 500K cells for initial convergence testing

Fine Mesh

Further refinement: 4M cells to verify independence

1 2 3

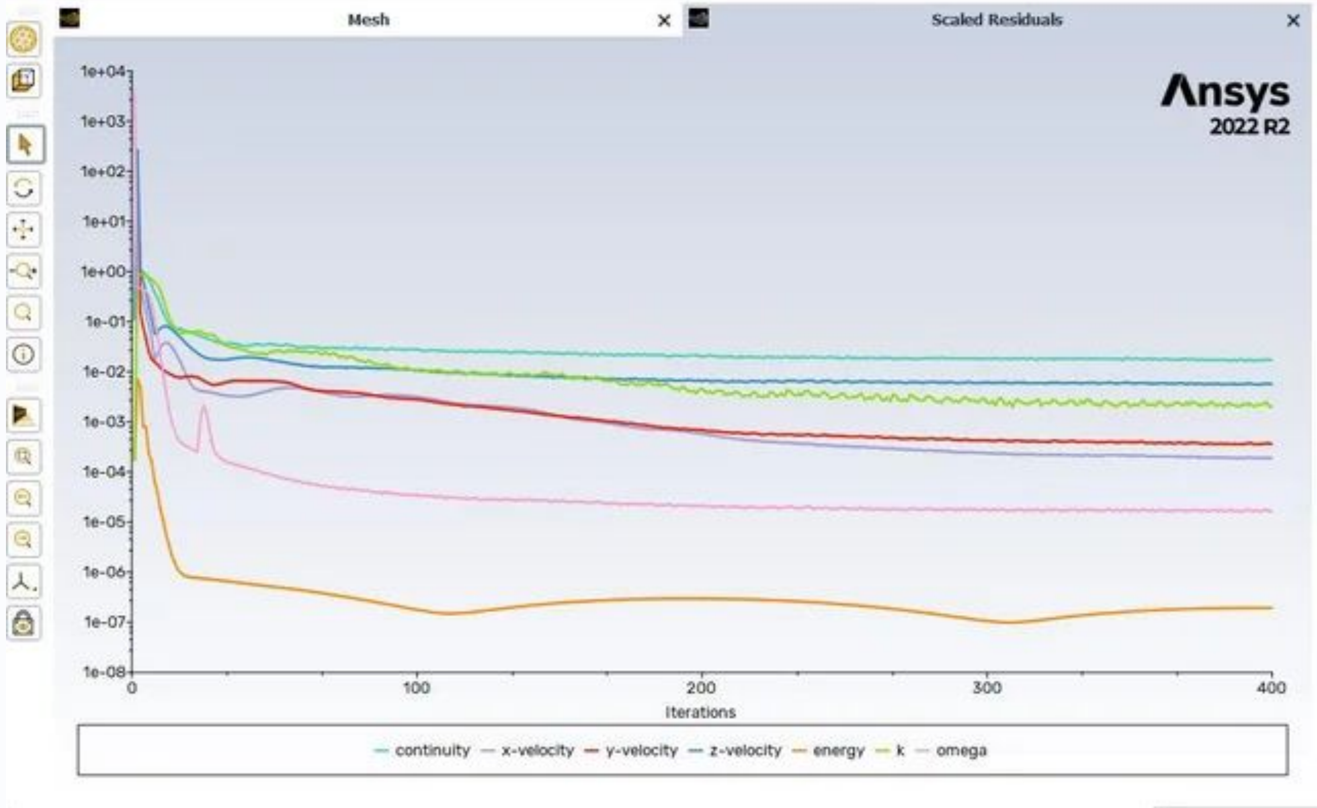
Medium Mesh

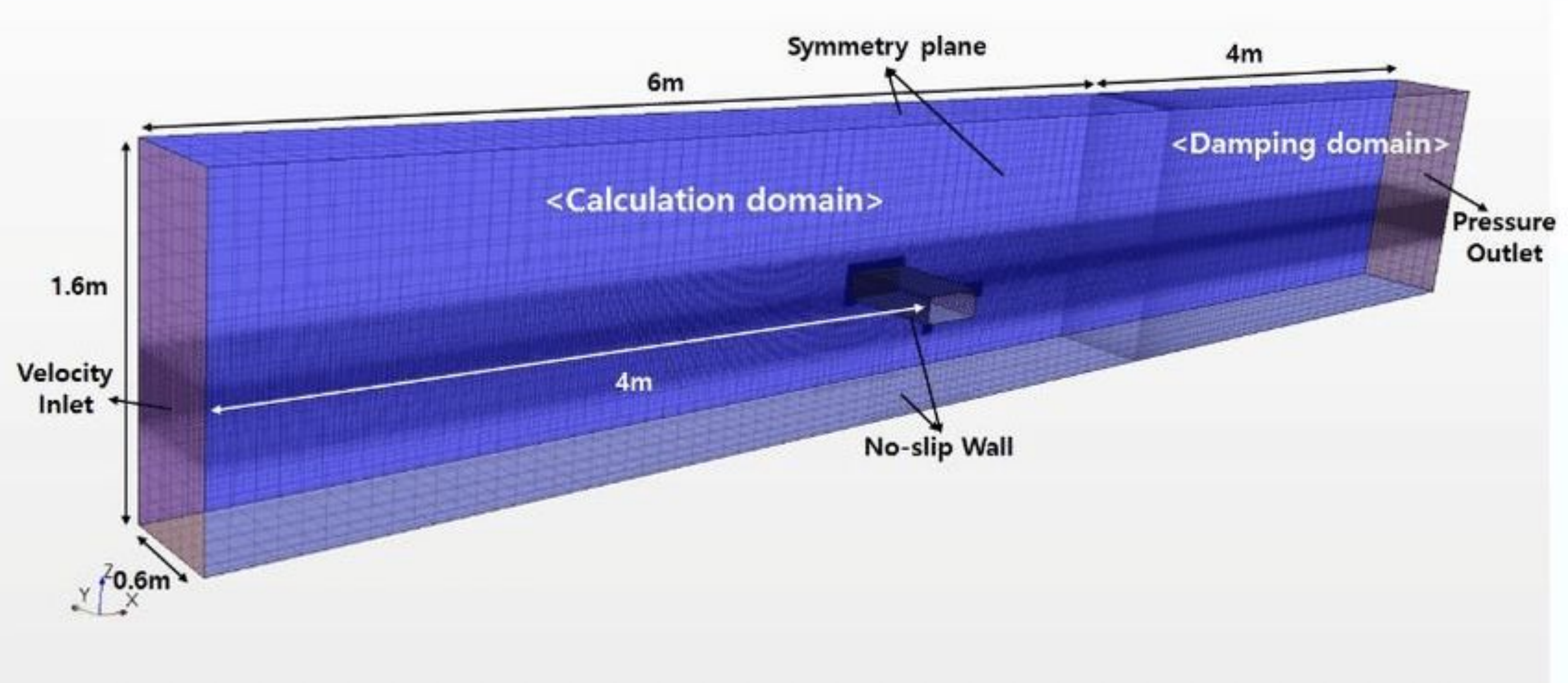
2× refinement: 1.5M cells for production runs

Convergence Criteria

Solution is mesh-independent when key engineering parameters change by less than 2% between successive refinements. Monitor:

- Global quantities (ΔP, drag, heat flux).
- Local values at critical locations.
- Profile distributions across planes.





4. Boundary Conditions & Physical Models

Accurate boundary conditions and physics selection determine whether CFD captures real-world behavior—wrong choices invalidate even the finest mesh.

Define Inlets & Outlets

Mass Flow Inlet

Specify mass flow rate when total flow is known—solver distributes flow based on resistance. Include turbulence intensity (typically 1-10%) and hydraulic diameter.

Velocity Inlet

Use when flow distribution is prescribed. Requires velocity magnitude and direction—suitable for uniform developed profiles or CFD-mapped distributions.

Pressure Outlet

Static pressure boundary (gauge or absolute). Allows backflow if solution requires—use target mass flow rate to stabilize initial iterations.

Critical Rule: Never over-specify—setting both inlet velocity AND outlet flow rate creates mathematical inconsistency. Pressure-based solvers require one velocity/mass-flow boundary and one pressure boundary.

Wall & Symmetry Conditions

Wall Boundary Properties

No-slip condition: Zero velocity at wall (standard for viscous flows)
Roughness height: Equivalent sand-grain roughness in microns
Thermal condition: Adiabatic, constant temperature, or heat flux
Motion: Stationary, rotating, or translating reference frame

Symmetry Plane Guidelines

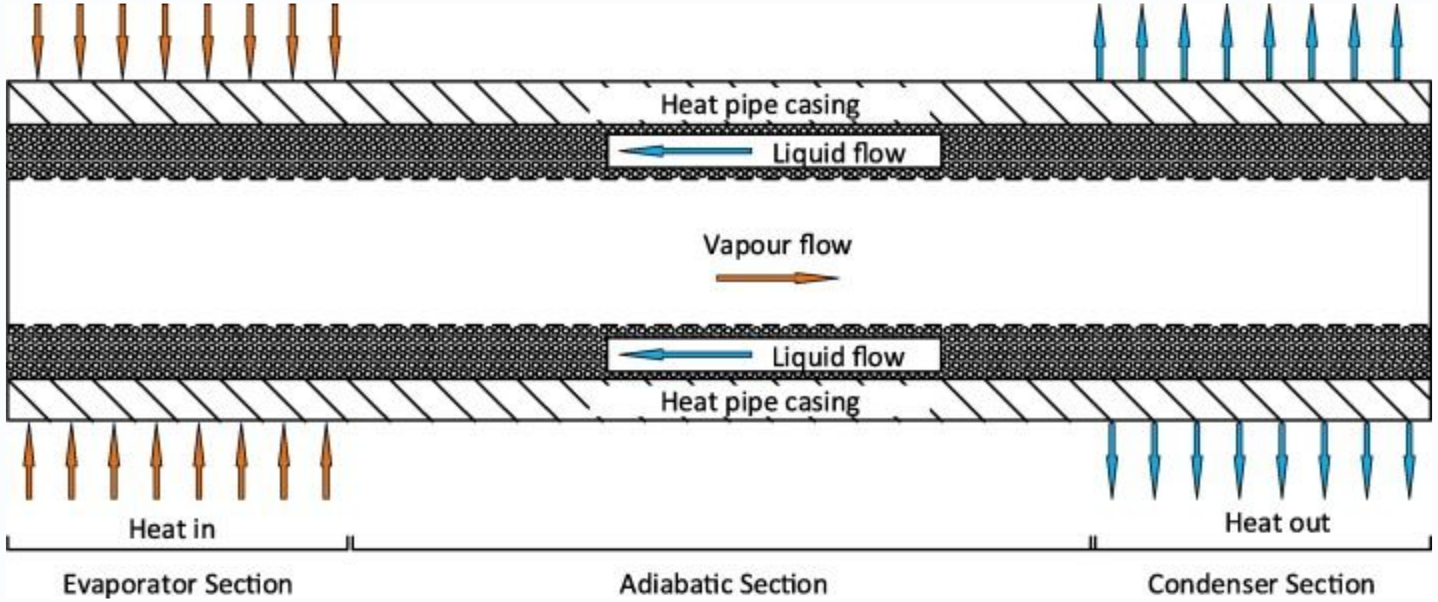
Use only when geometry AND flow field are truly symmetric. Zero normal gradients enforced—inappropriate for asymmetric separation or vortices.

Example Configuration

Pipe walls: Adiabatic, smooth (Ks = 0)

Heated surface: 350 K constant temperature

Symmetry: Midplane for symmetric manifold



Physical Model Setup

Turbulence Modeling

k-ω SST: Best for adverse pressure gradients, separation. k-ε Realizable: Good for free shear flows. LES/DES: For transient vortex-dominated flows with sufficient resolution.

Energy Equation

Enable when temperature variations affect density, viscosity, or are outputs of interest. Include viscous dissipation for high-speed flows.

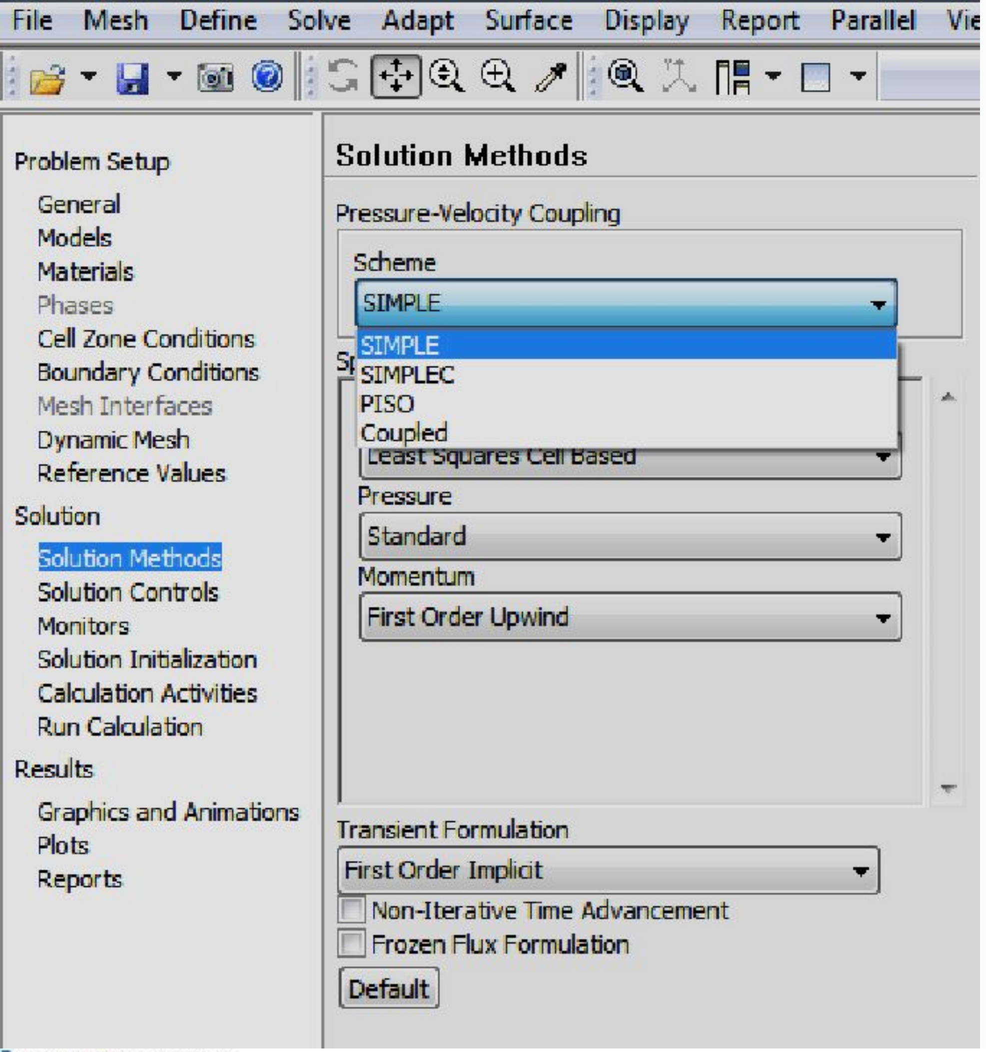
Radiation Models

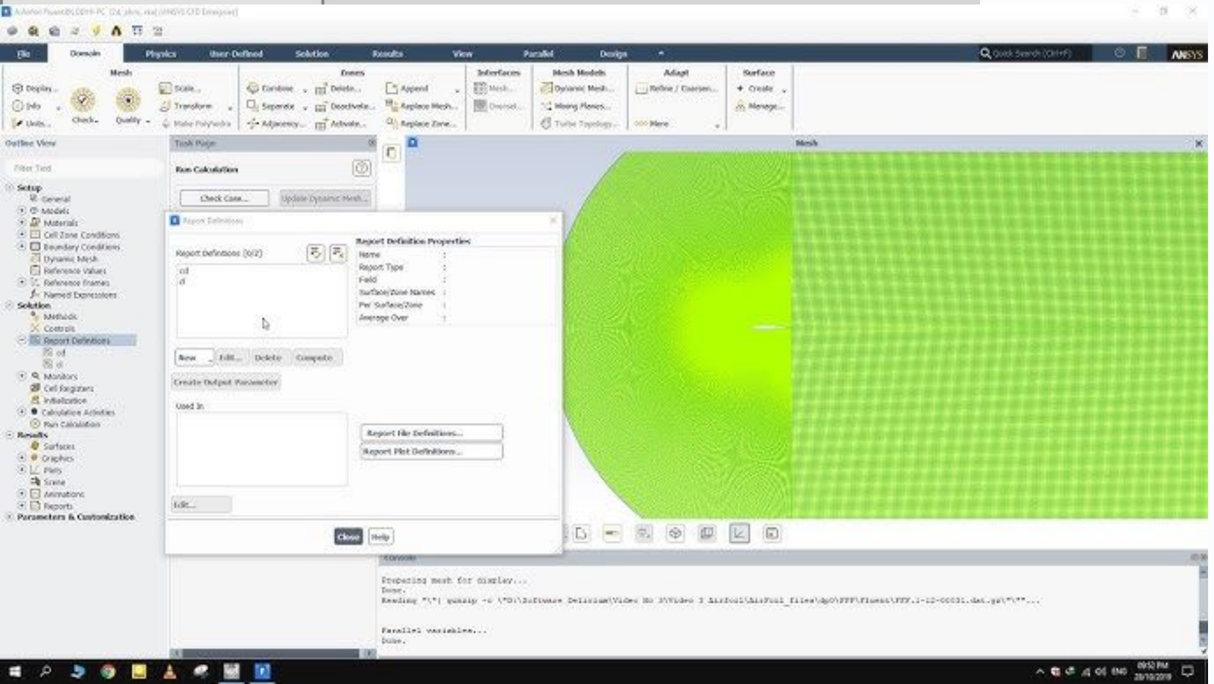
Discrete Ordinates (DO) or Surface-to-Surface (S2S) for combustion, furnaces, or high-temperature systems where radiation heat transfer is significant.

Multiphase Models

VOF for free surfaces, Eulerian for dispersed bubbly flows, DPM for dilute particle tracking—choose based on volume fraction and coupling strength.

Engineering judgment: Activate only necessary models—unnecessary physics increases computational cost and introduces instability without improving accuracy for your specific engineering question.





5. Solver Setup & Numerical Controls

Solver configuration and discretization schemes control numerical stability and accuracy—even correct physics fails with poor numerics.

Solver Type & Formulation

Pressure-Based Solver

Default for incompressible and low-speed compressible flows (Mach < 0.3). SIMPLE or SIMPLEC algorithm couples pressure and velocity through continuity constraint. Robust for most industrial applications.

Density-Based Solver

Required for high-speed compressible flows (Mach > 0.3) where density variations are significant. Solves coupled momentum and continuity simultaneously—better for supersonic nozzles, turbomachinery.

Steady vs. Transient

Use steady-state for time-independent flows to save computational cost. Switch to transient when capturing vortex shedding, pulsatile flow, or periodic phenomena—requires appropriate time step selection.

Example: Compressible nozzle flow (Mach 1.8) → density-based coupled solver with transient formulation, time step $\Delta t = 1\times10^{-6}$ s based on acoustic CFL condition.

Discretization Schemes

Spatial Discretization

First-order upwind: Stable but diffusive—use only for initial iterations

Second-order upwind: Production standard—balances accuracy and stability

QUICK/MUSCL: Third-order accuracy for refined meshes and smooth flows

Temporal Discretization

First-order implicit: Unconditionally stable, acceptable for slow transients

Second-order implicit: Preferred for accurate transient phenomena

Under-relaxation factors control solution stability—reduce from defaults (0.7 momentum, 0.3 pressure) for difficult cases.

2nd

Momentum

Second-order upwind scheme

2nd

Time Integration

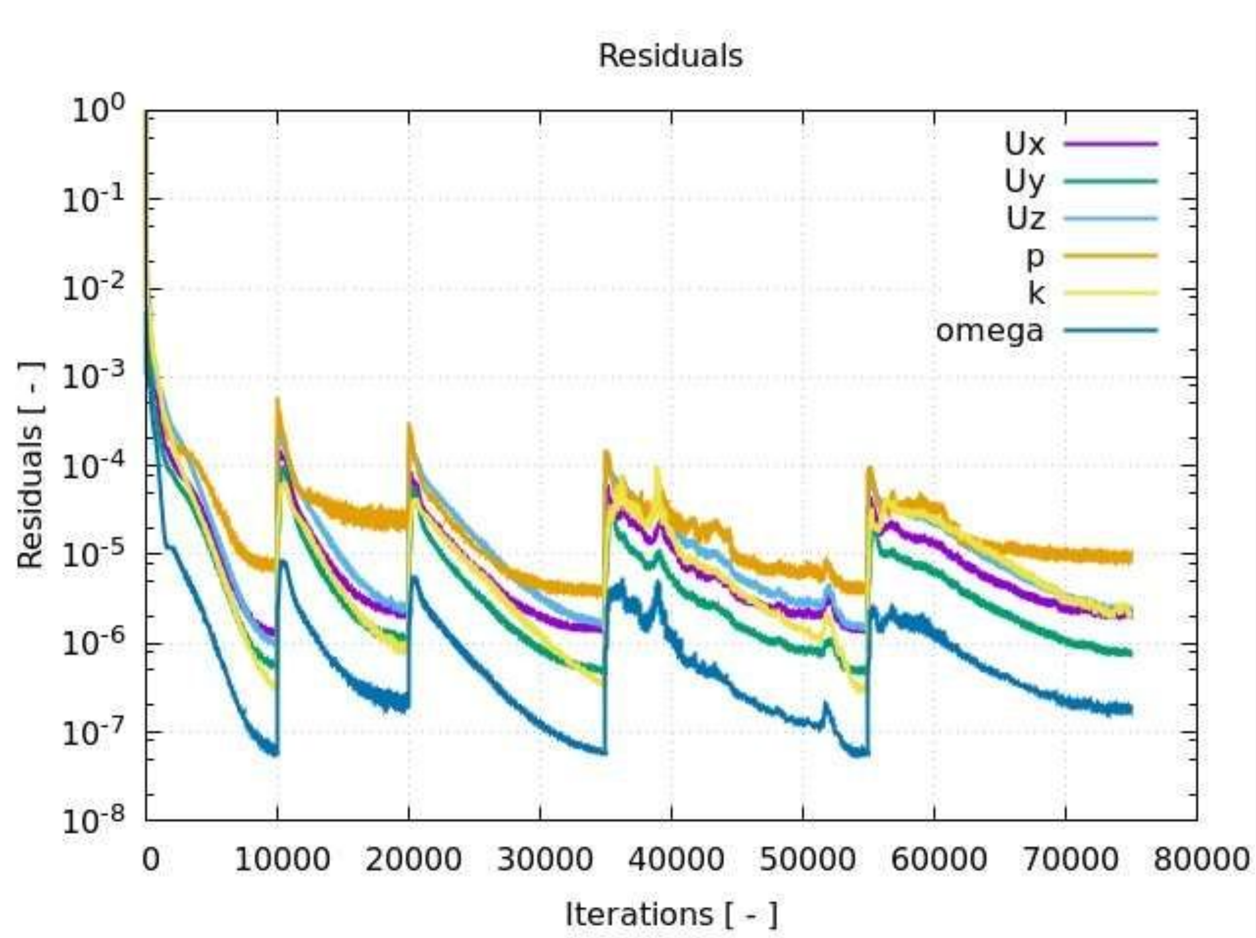
Second-order implicit

0.7

Relaxation Factor

Momentum equation

> Common mistake: Using first-order everywhere produces smeared gradients and incorrect separation locations—visual results look reasonable but quantitative outputs are wrong.



Convergence & Residuals

01

Set Residual Targets

Continuity, momentum: 10^{-4} for most flows. Energy: 10^{-6} for accurate heat transfer. Stricter for precision applications.

02

Monitor Key Quantities

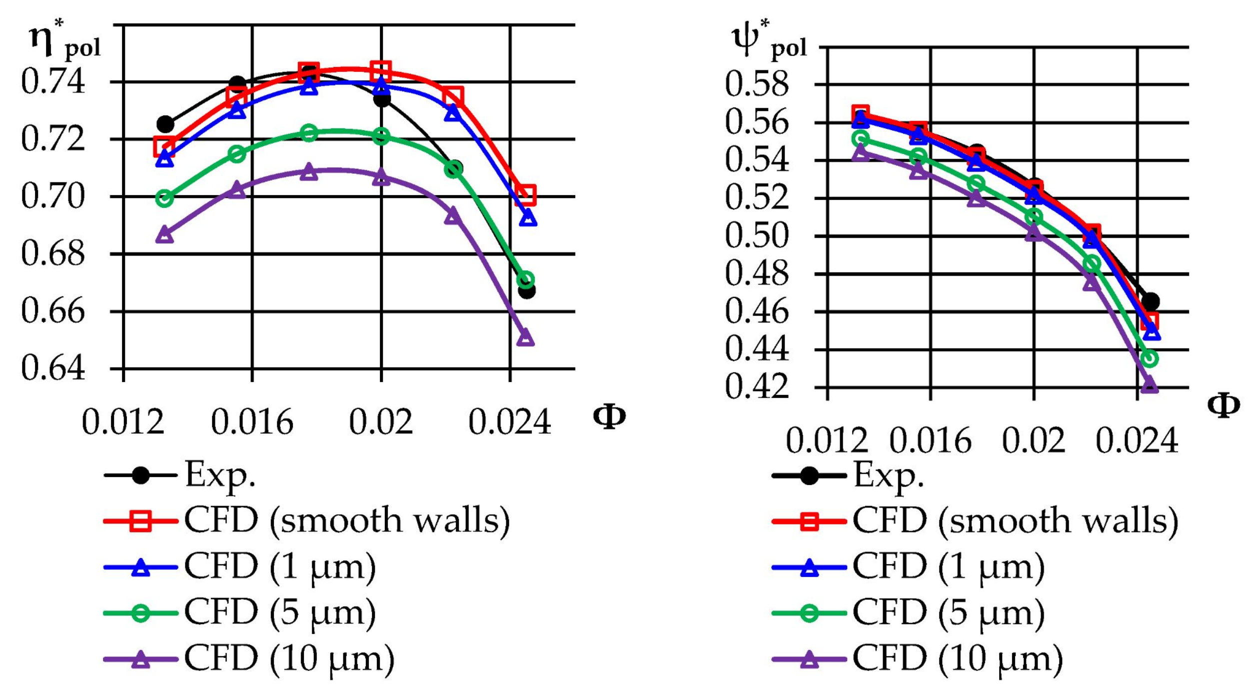
Track integral values: mass flow rate, forces, pressure drop, average temperatures. Solution is converged when these stabilize, not just residuals.

03

Verify Stationarity

Check that monitored quantities oscillate with <1% amplitude for at least 200 iterations after residuals plateau—true indication of convergence.

Critical insight: Residual drop indicates equation balance, but physical convergence requires stable engineering outputs. A 10^{-3} residual with oscillating drag coefficient means the solution has not converged.



6. Verification & Validation

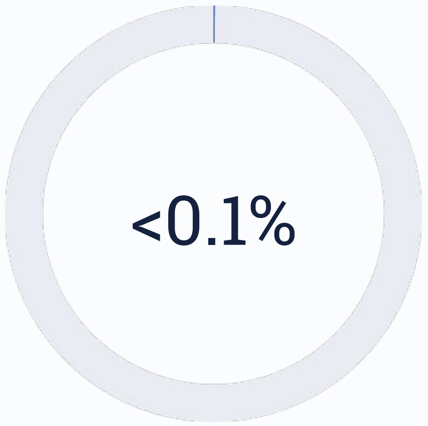
Verification confirms correct equation solution; validation confirms correct physics representation—both are essential for engineering confidence in CFD predictions.

Mass & Energy Balance Check

Conservation Verification

Fundamental laws must be satisfied within numerical tolerance. Post-process to confirm:

- Mass flow rate: inlet = outlet (±0.1%)
- Energy: total enthalpy flux conserved
- Momentum: forces balance pressure and shear



Mass Imbalance

Target for production runs

Imbalance Diagnosis

Persistent imbalance > 1% indicates:

- Non-converged solution
- Poor mesh quality near boundaries
- Incorrect boundary condition specification

Energy Imbalance

Acceptable for thermal cases

> Common mistake: Accepting 2% mass imbalance as "close enough"—this compounds through the domain and invalidates quantitative predictions.

Grid Convergence Index (GCI)

1

Richardson Extrapolation

Estimate exact solution by extrapolating grid-refined sequence to zero cell size

2

Calculate GCI

Quantify discretization uncertainty as percentage of solution value

3

Report Confidence

State engineering result with numerical uncertainty bounds

GCI methodology (Roache 1998) requires three meshes with constant refinement ratio $r \approx 1.3$-2.0. Calculate apparent order of accuracy and discretization error estimate.

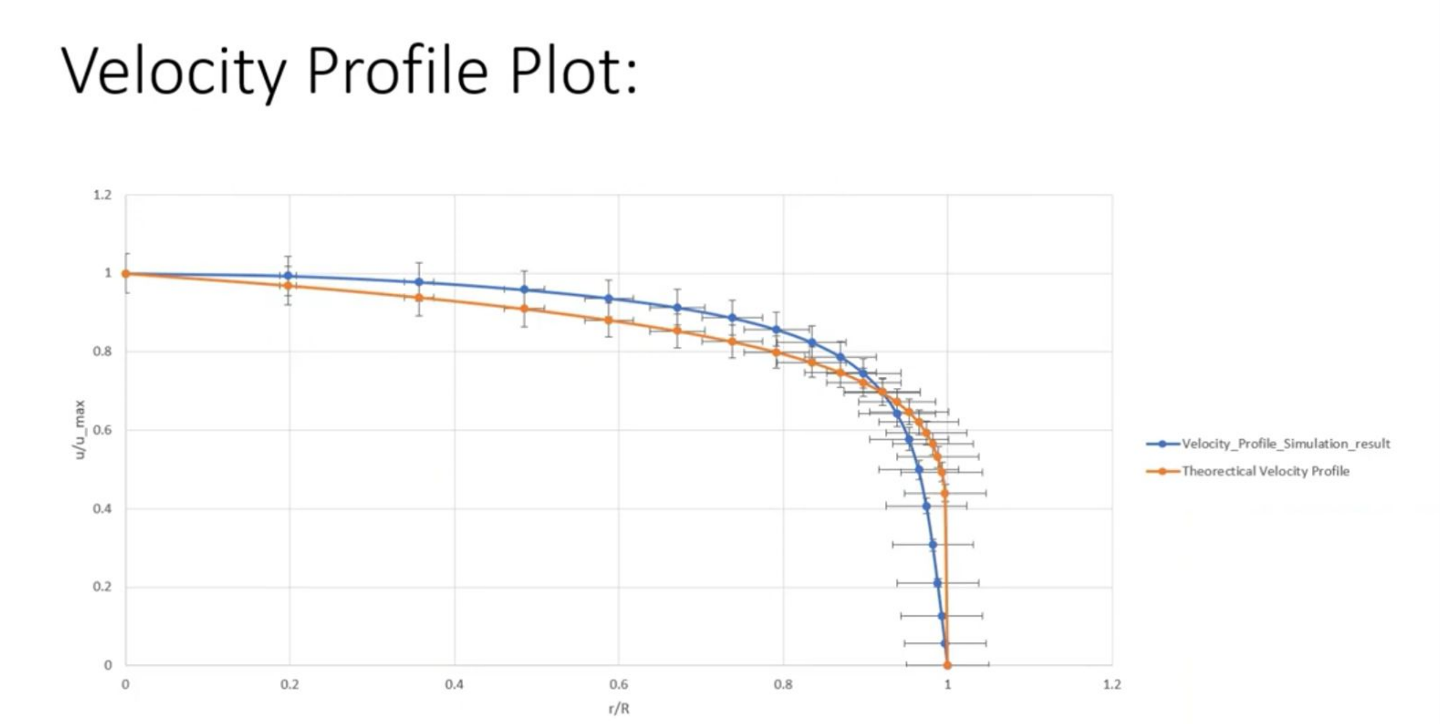
Example: Pressure drop predictions: Coarse mesh = 985 Pa, Medium = 964 Pa, Fine = 959 Pa → GCI = 2.5% → Report: ΔP = 959 ± 24 Pa with 95% confidence.

Validation with Analytical / Empirical Data

Comparison Strategy

1. Identify relevant benchmark: analytical solution, correlation, or experimental data
2. Match all conditions: geometry, Re, boundary conditions, measurement locations
3. Compare appropriate quantities: friction factor, Nusselt number, drag coefficient
4. Document deviations and assess acceptability

Acceptance criteria: Agreement within ±5% for well-established flows; ±10-15% for complex geometries with experimental uncertainty.



Example Validation

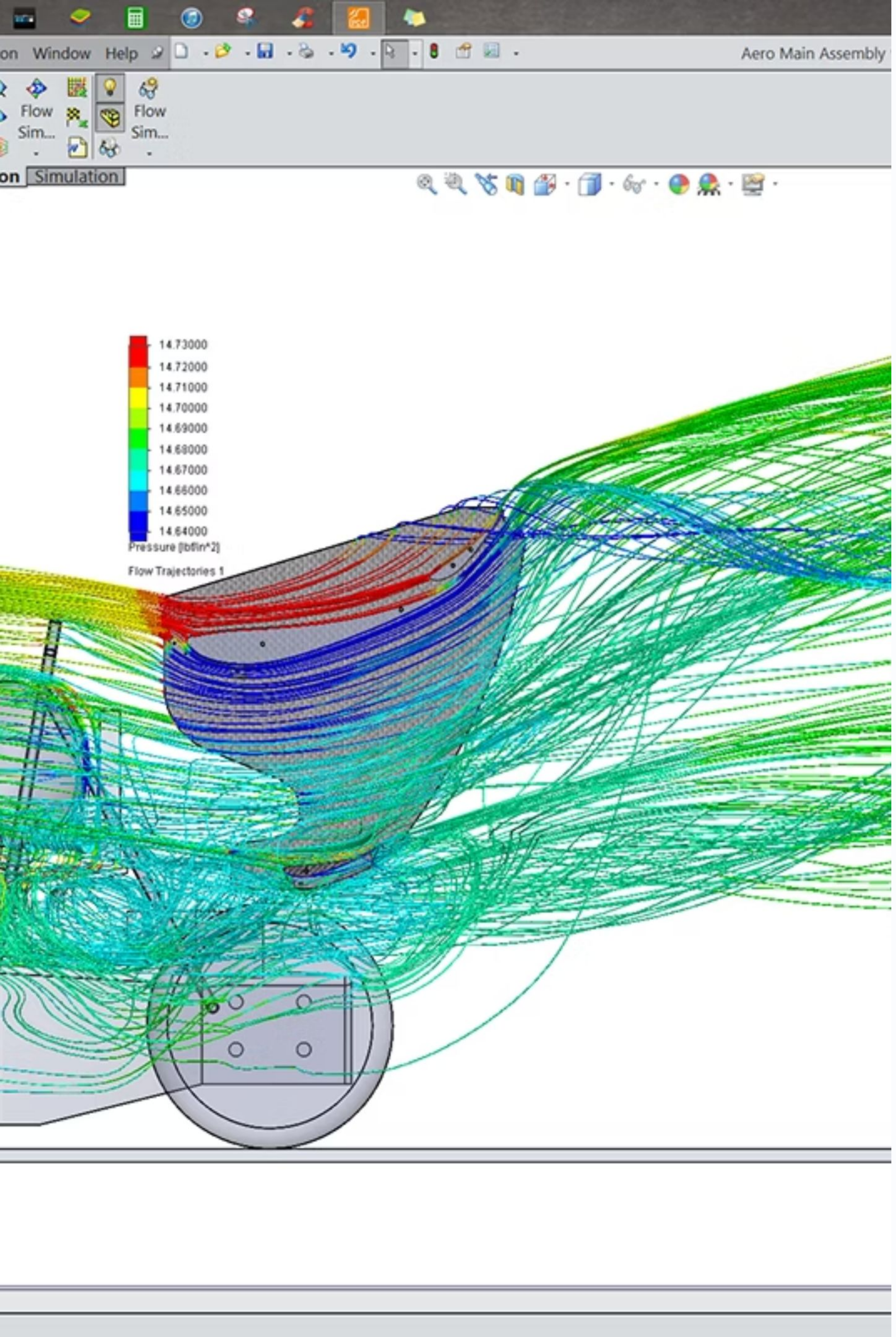
Turbulent pipe flow at Re = 50,000

CFD: f = 0.0182

Moody chart: f = 0.0177

Deviation: +2.8% ✓

Common mistakes: Comparing static pressure to total pressure; neglecting entrance length effects in pipe flow validation; using inappropriate Reynolds number definition.



7. Post-Processing & Interpretation

Effective visualization and quantitative extraction transform numerical results into engineering insights—thoughtful post-processing reveals the physics story.

Visualization & Streamlines

Flow Patterns

Streamlines reveal pathlines, separation zones, and recirculation regions that govern pressure loss and mixing performance.

Scalar Fields

Contour plots display pressure, temperature, velocity magnitude on planes or surfaces—use consistent scales for comparison.

Vector Displays

Arrows indicate local direction and magnitude—essential for understanding complex 3D flow structures and vorticity.

Best practices: Use sufficient seed points for streamlines (50-100), select representative plane locations, annotate key features, and maintain consistent color scales across design iterations for meaningful comparison.

Quantitative Data Extraction

Objective

Derive ΔP, forces, heat flux, etc.

Why important

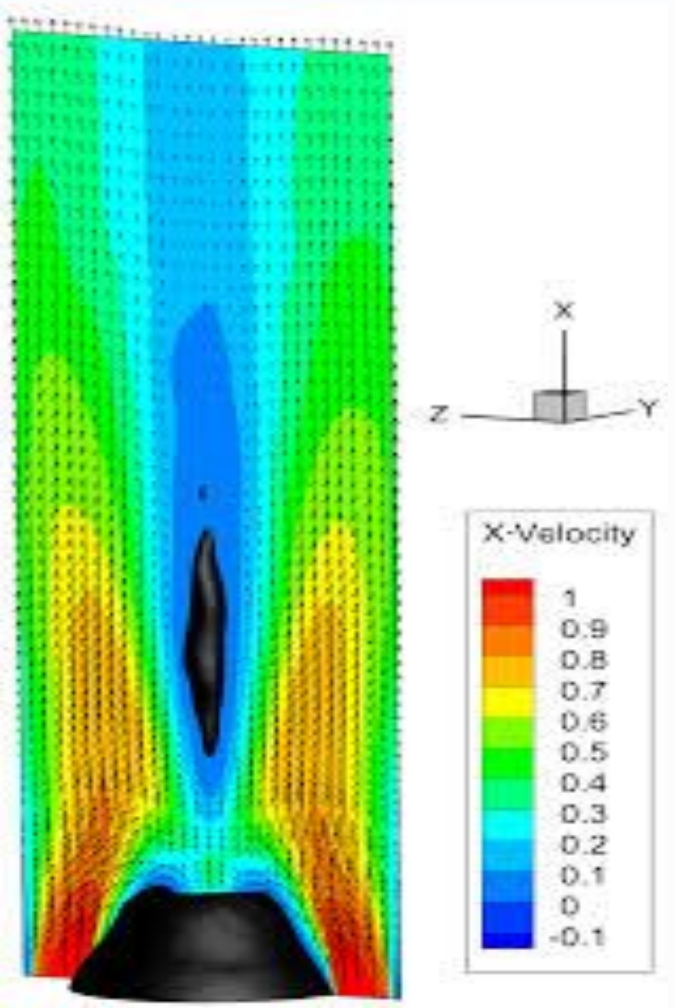
Design decisions depend on these metrics.

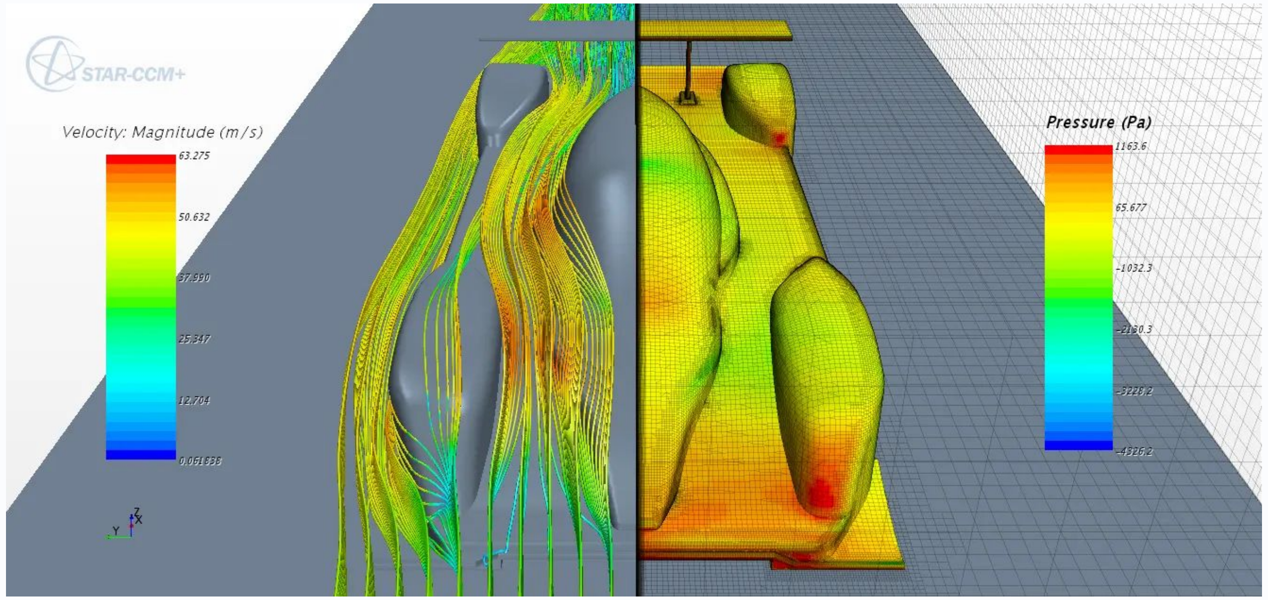
Example

Average Nusselt number = 85; ΔP = 0.9 kPa.

Common mistakes:

- Extracting from single cell instead of area-average
- Inconsistent reference pressure or units





Turbulence & Boundary Layer Review

Objective

Inspect Y+, turbulence kinetic energy, wall shear stress.

Why important

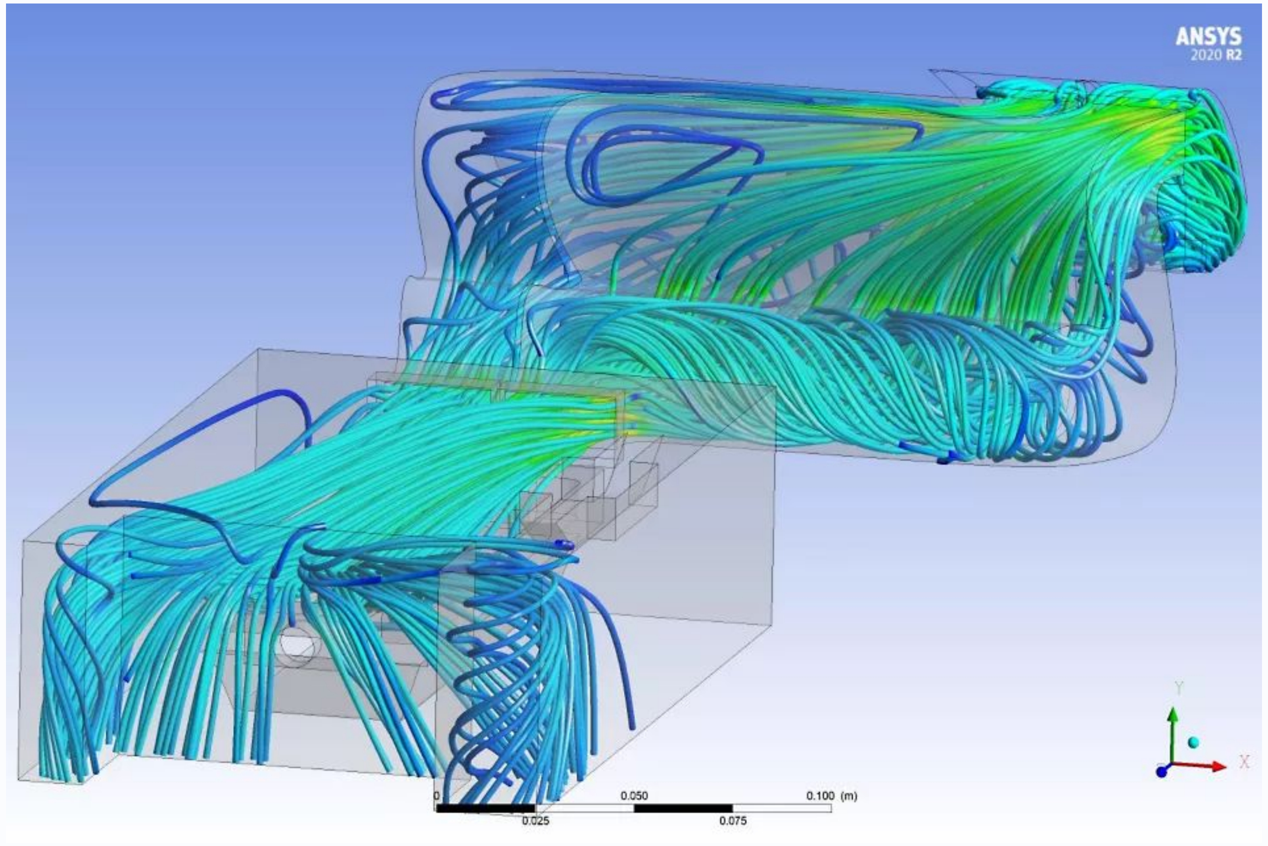
Detects poor near-wall resolution.

Example

Y+ < 1 across 95% wall area.

Common mistakes:

- Ignoring regions with Y+ > 300
- Averaging Y+ over entire wall indiscriminately



Thermal Analysis Results

Objective

Review temperature fields, heat transfer coefficients.

Why important

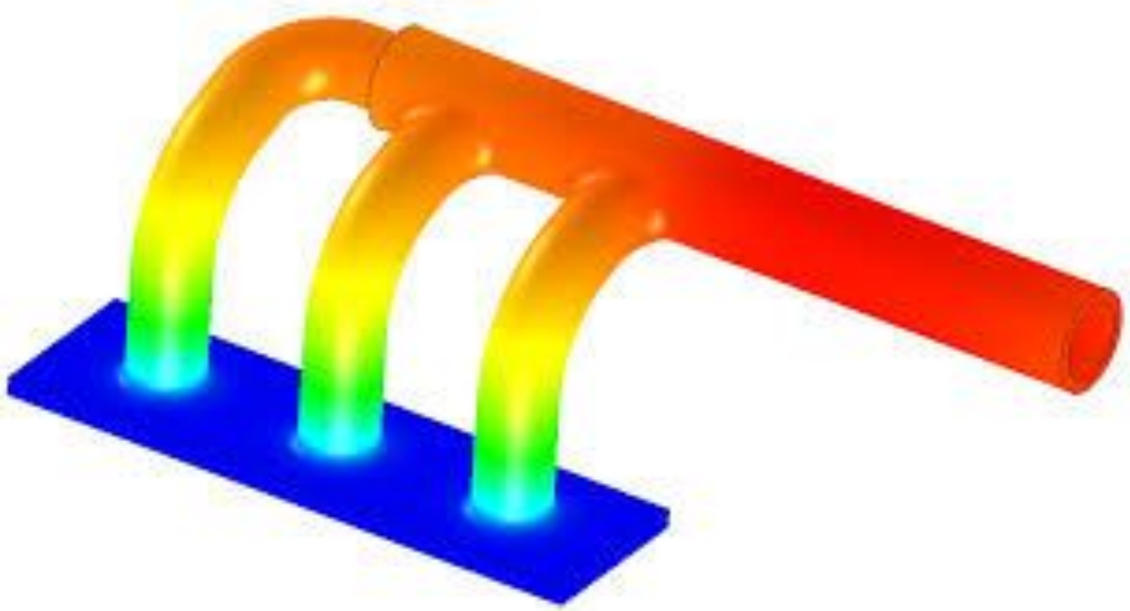
Ensures realistic gradients and heat flux continuity.

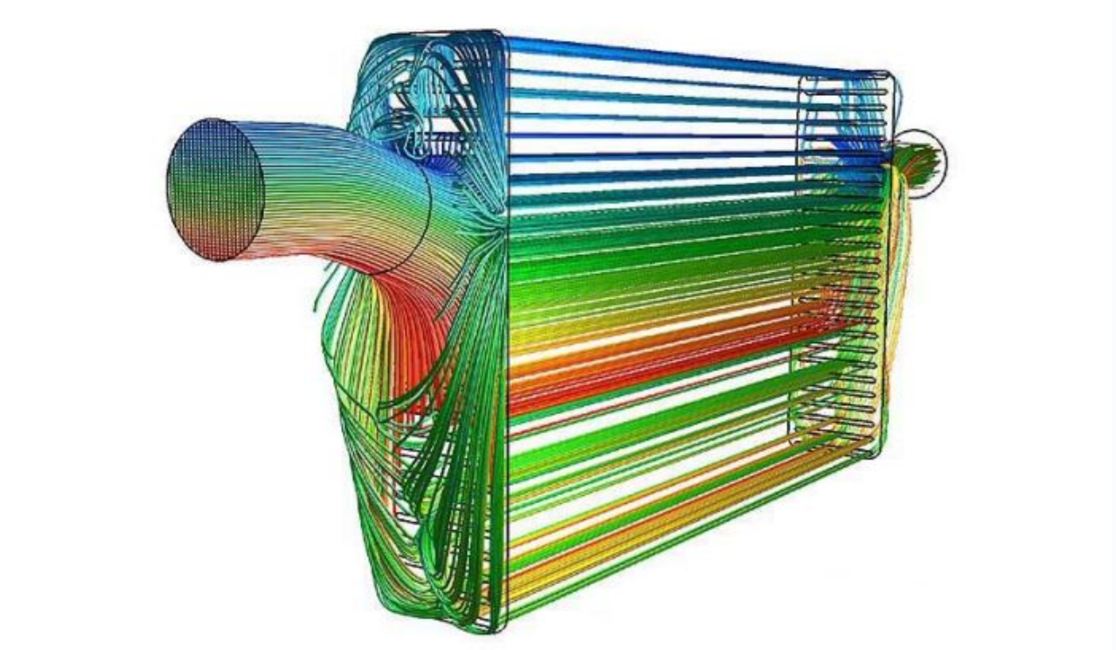
Example

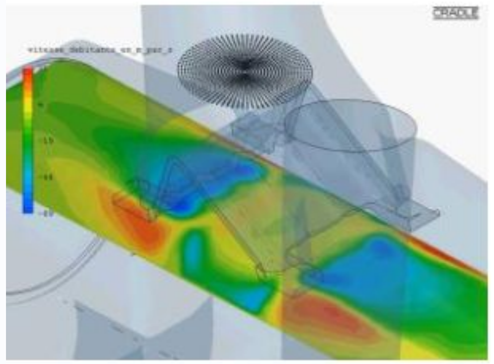
Hot spot at outlet = 105 °C vs. target < 90 °C.

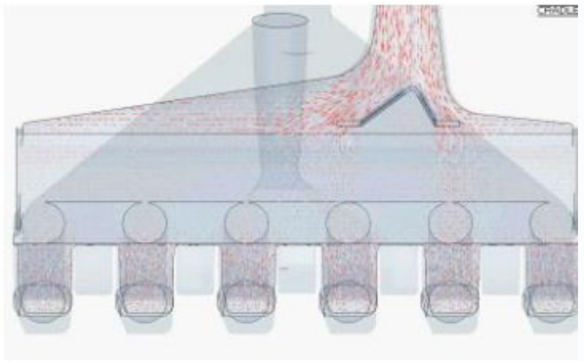
Common mistakes:

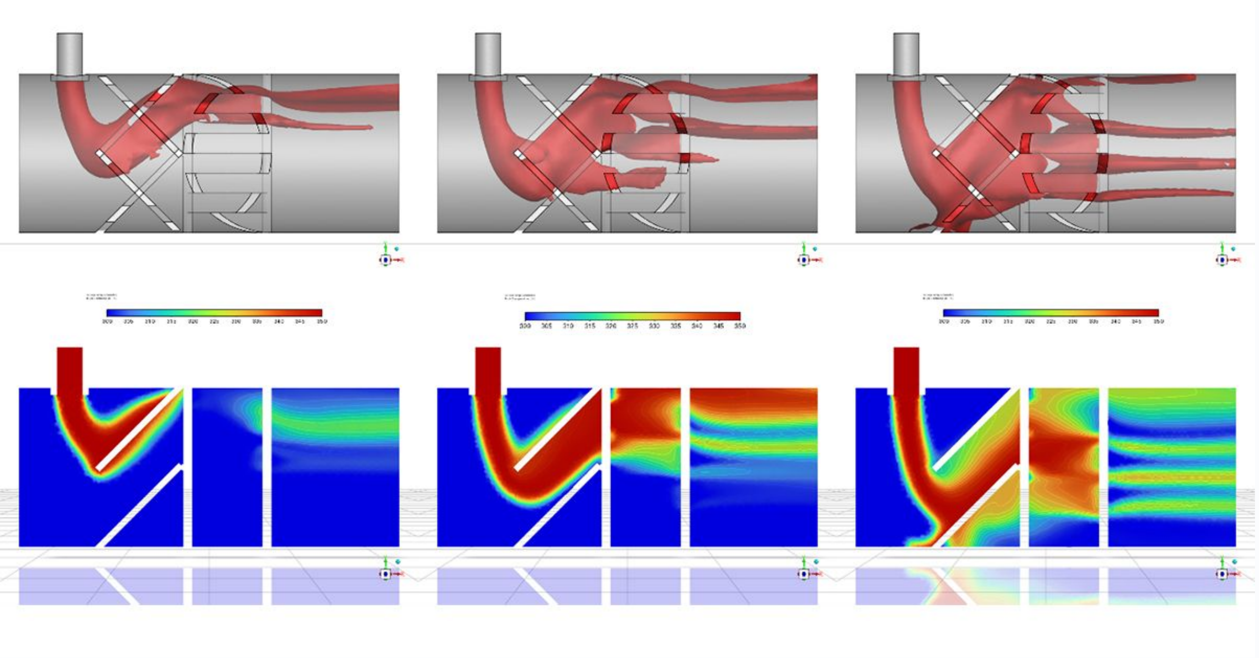
- Non-converged thermal fields
- Using default emissivity = 1.0 in radiation

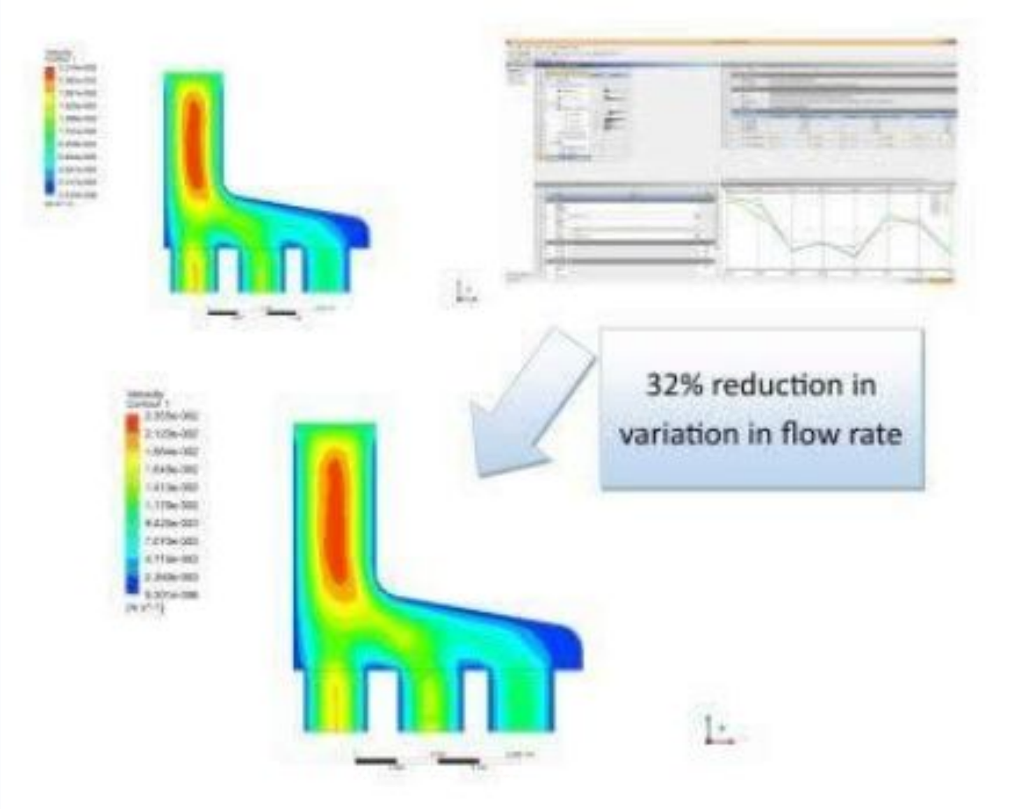












8. Design Optimization & Sensitivity

Design optimization and sensitivity analysis transform CFD from analysis tool to design enabler—systematic parameter studies reveal optimal configurations and quantify design robustness.

Parametric Studies

Objective

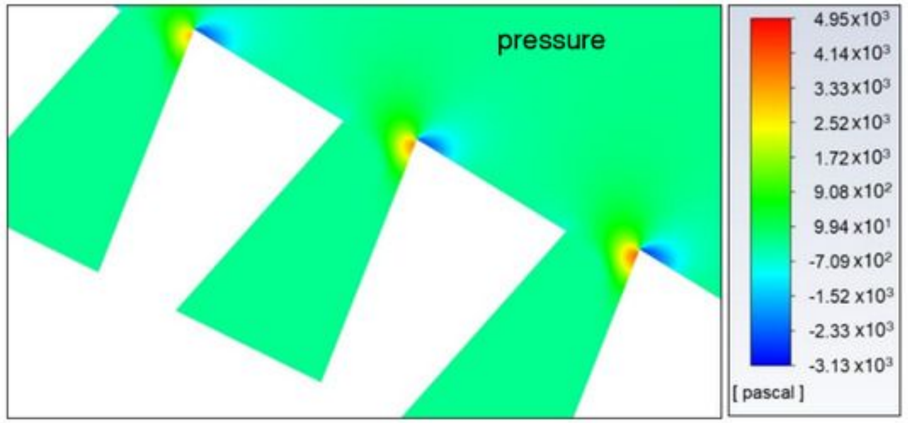
Evaluate performance vs. geometry/operating changes.

Why important

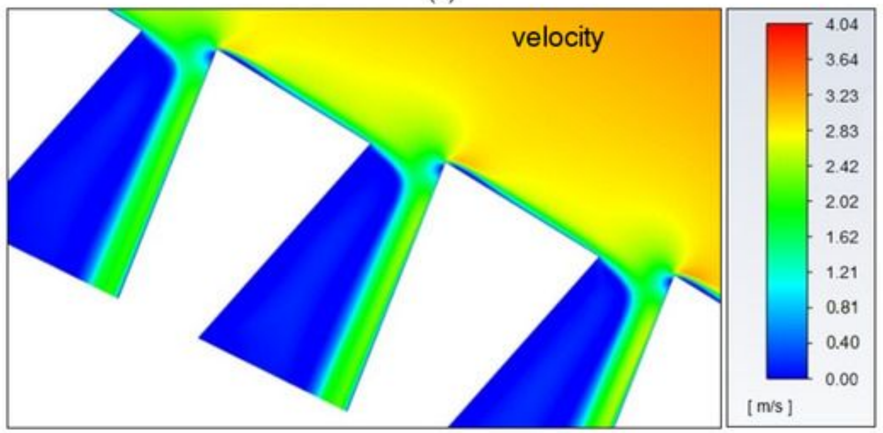
Identifies design levers.

Common mistakes:

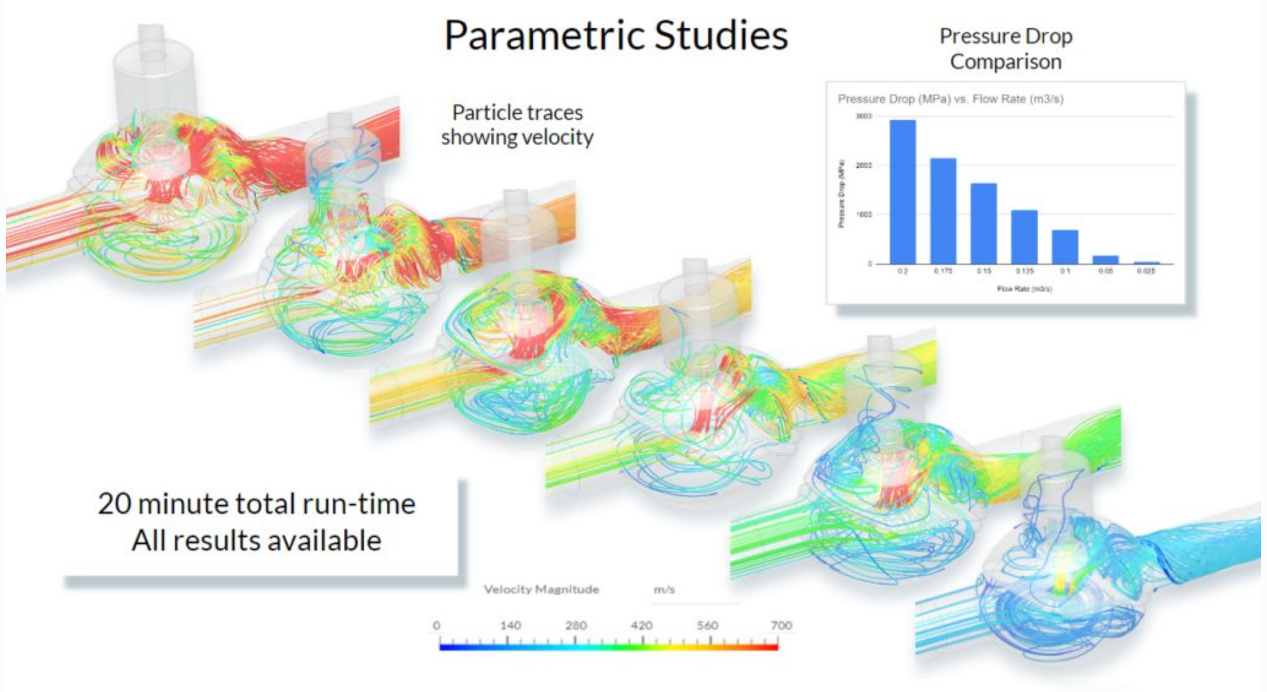
- Changing multiple parameters without DOE planning
- Comparing unconverged runs



(c)



(d)



Sensitivity & Uncertainty Quantification

Objective

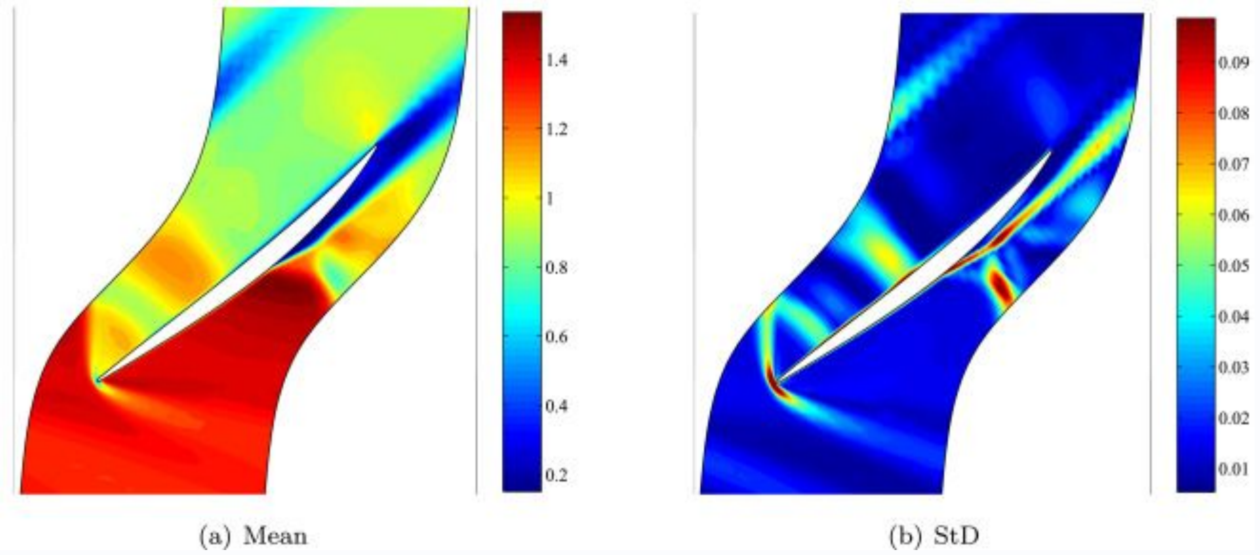
Assess variability impact on results.

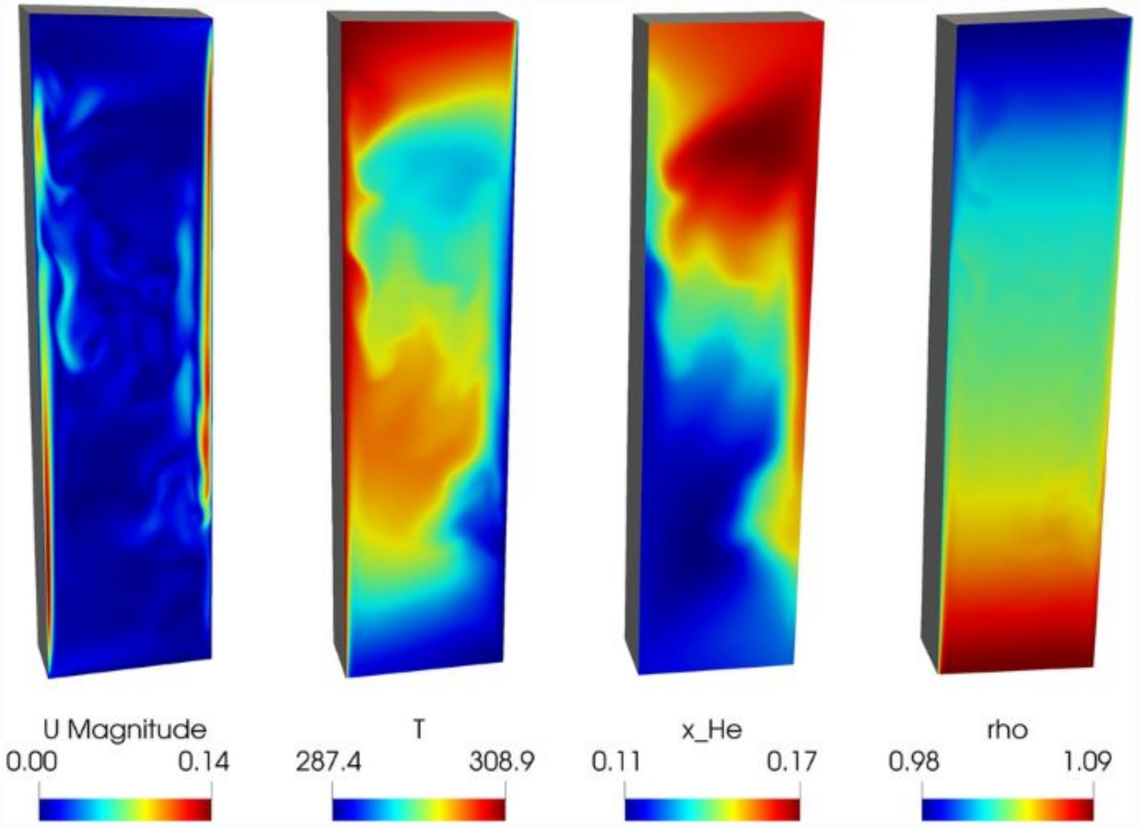
Why important

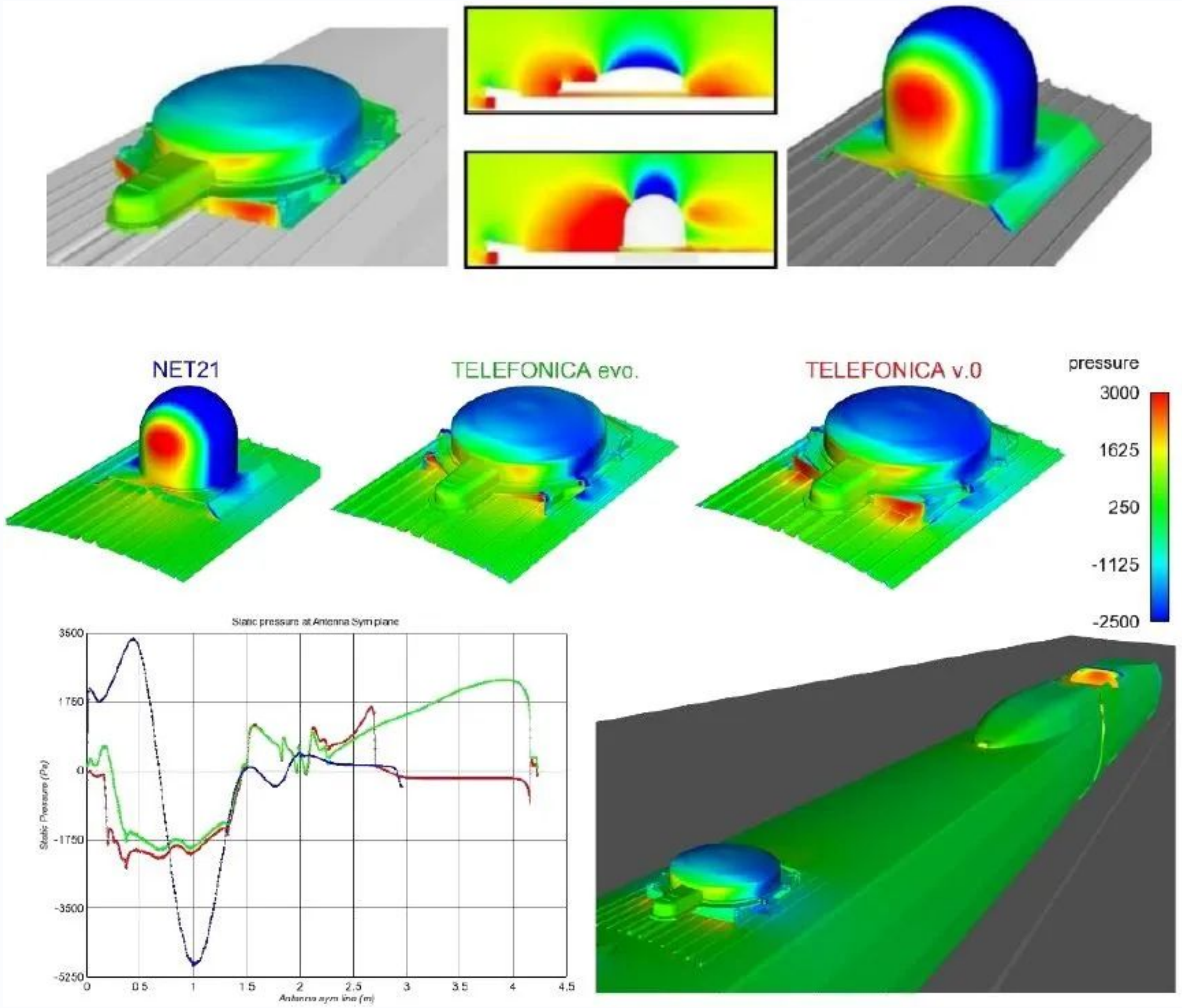
Shows design robustness.

Common mistakes:

- Ignoring turbulence intensity variations
- Assuming deterministic results for stochastic inputs







9. Reporting & Engineering Decision

Proper reporting transforms CFD results into actionable engineering decisions and institutional knowledge—comprehensive documentation ensures repeatability while clear insights drive design improvements.

Documentation & Traceability

Objective

Record all setup parameters, models, and key results.

Why important

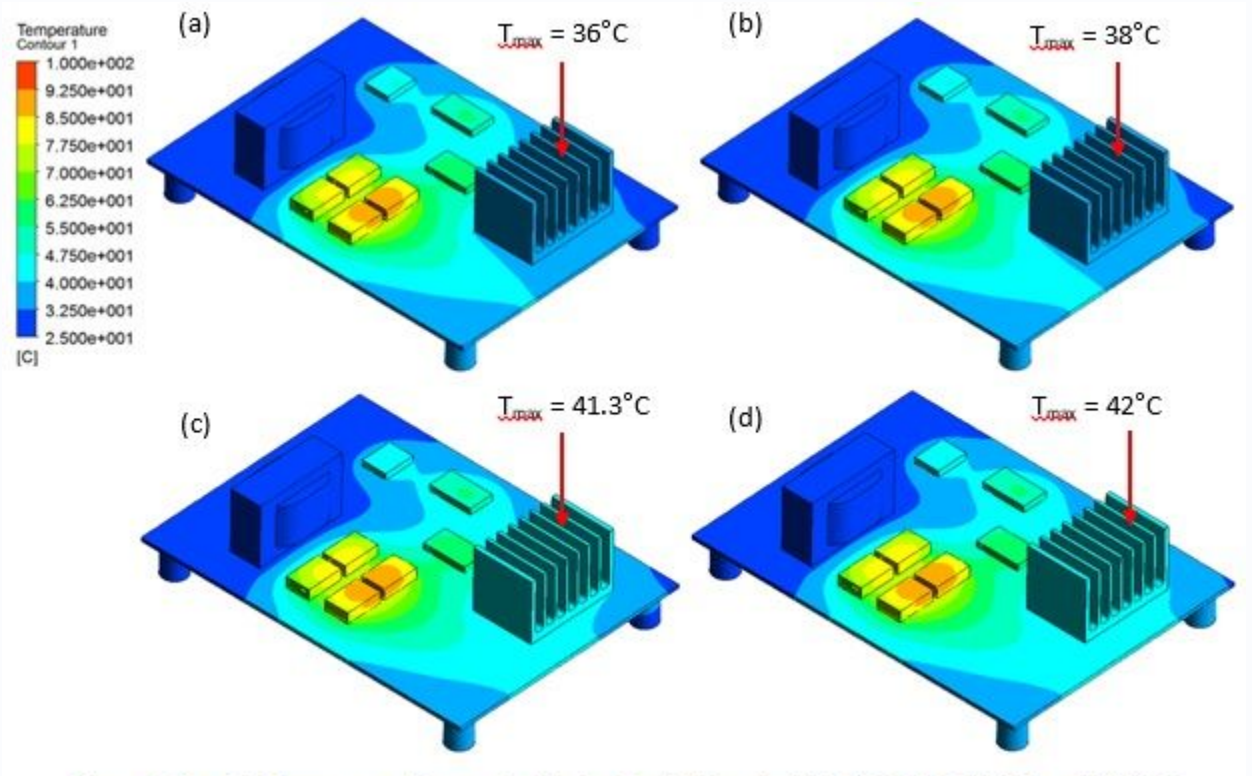
Enables repeatability and peer review.

Example

Include domain image, mesh stats, residual plots, boundary summary.

Common mistakes:

- Missing solver version, model, or mesh count
- Inconsistent naming or units



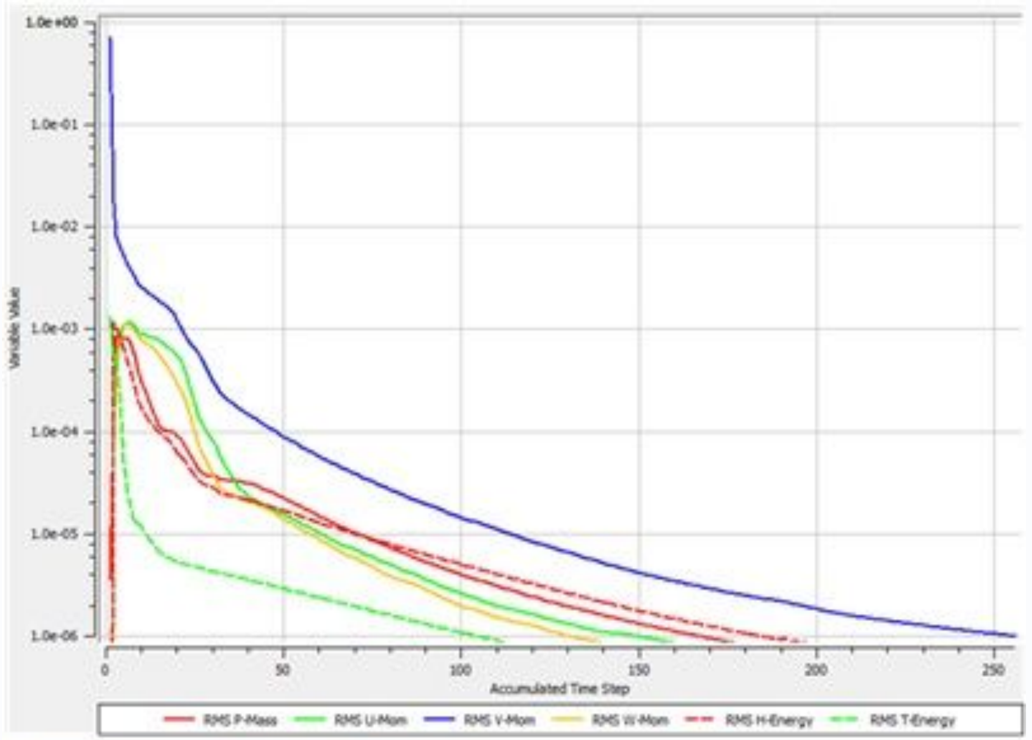


Figure 1 - Example Residual Monitors

Engineering Insights & Design Actions

Objective

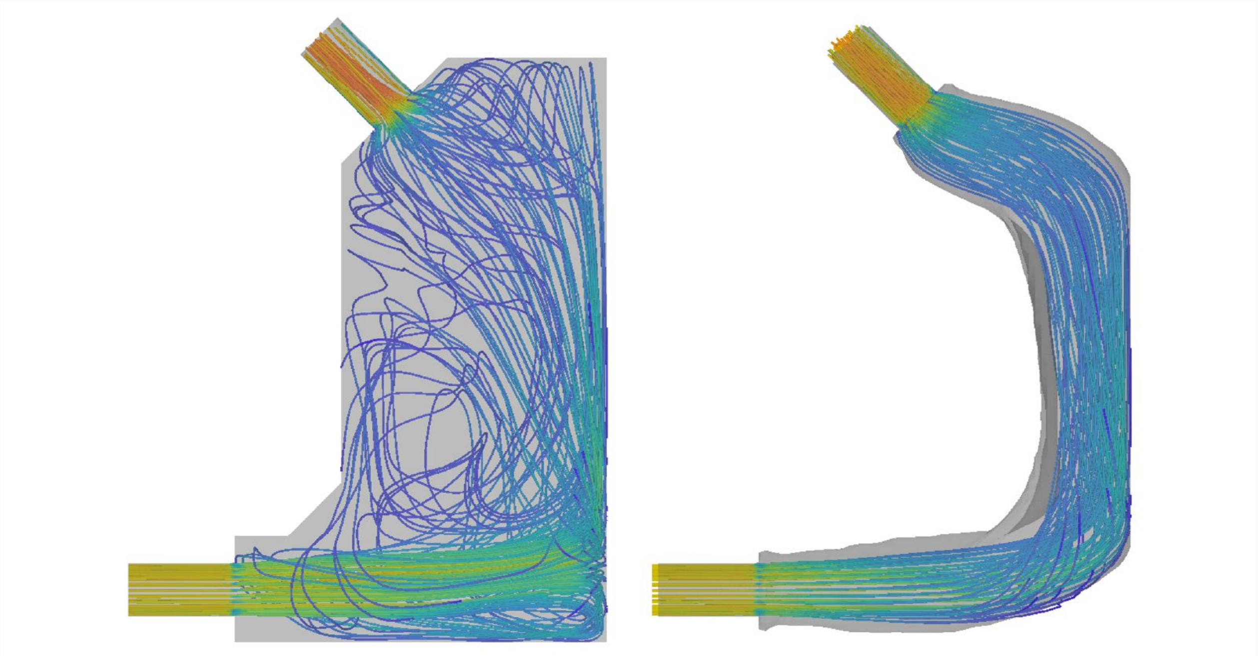
Translate results into tangible recommendations.

Why important

CFD's value = design improvement.

Example

Add guide vane → velocity uniformity +12%, ΔP +1.5%.



Common mistakes:

- Reporting beautiful images without design action
- Over-interpreting marginal results (< 3% change)

Lessons Learned & Future Simulations

Objective

Capture learnings for next iteration.

Why important

Builds institutional knowledge.

Example

Next run: include radiation + finer wall mesh.

> **Common mistakes:**
>
> - Not updating simulation best-practice checklist