SPC 307

Aerodynamics

Lecture 2

Overview of the CFD

February 2, 2016

Overview of the CFD Process

Introduction

Lecture Theme:

All CFD simulations follow the same key stages. This lecture will explain how to go from the original planning stage to analyzing the end results

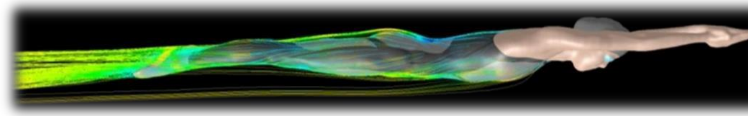
Learning Aims:

You will learn:
- The basics of what CFD is and how it works
- The different steps involved in a successful CFD project

Learning Objectives:

When you begin your own CFD project, you will know what each of the steps requires and be able to plan accordingly

What is CFD?



Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena.

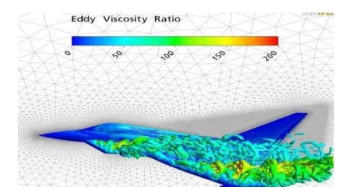
To predict these phenomena, CFD solves equations for conservation of mass, momentum, energy etc..

CFD can provide detailed information on the fluid flow behavior:

- Distribution of pressure, velocity, temperature, etc.
- Forces like Lift, Drag.. (external flows, Aero, Auto..)
- Distribution of multiple phases (gas-liquid, gas-solid..)
- Species composition (reactions, combustion, pollutants..)
- Much more

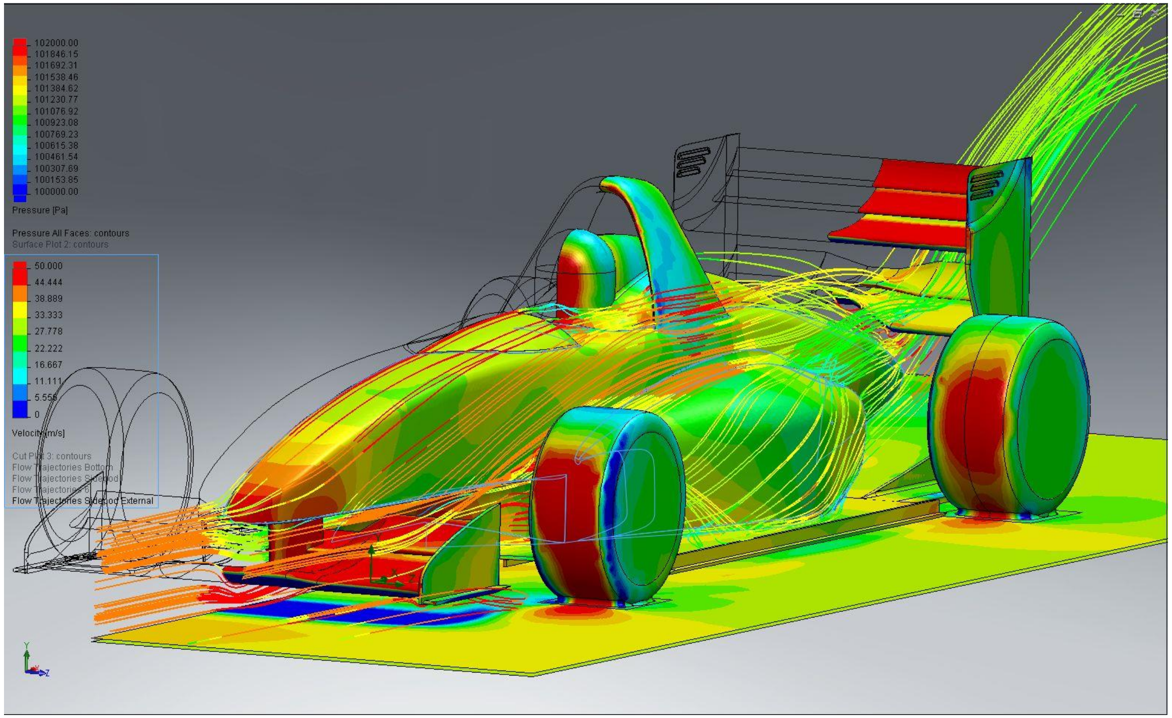
CFD is used in all stages of the engineering process:

- Conceptual studies of new designs
- Detailed product development
- Optimization
- Troubleshooting
- Redesign

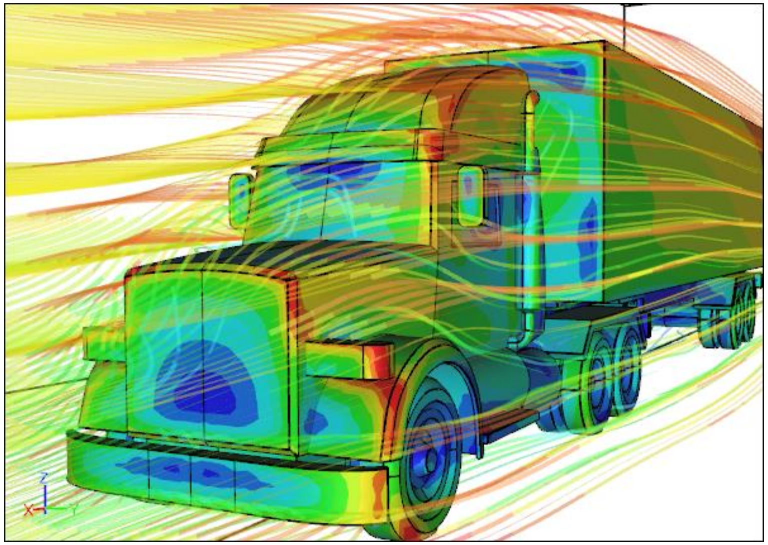


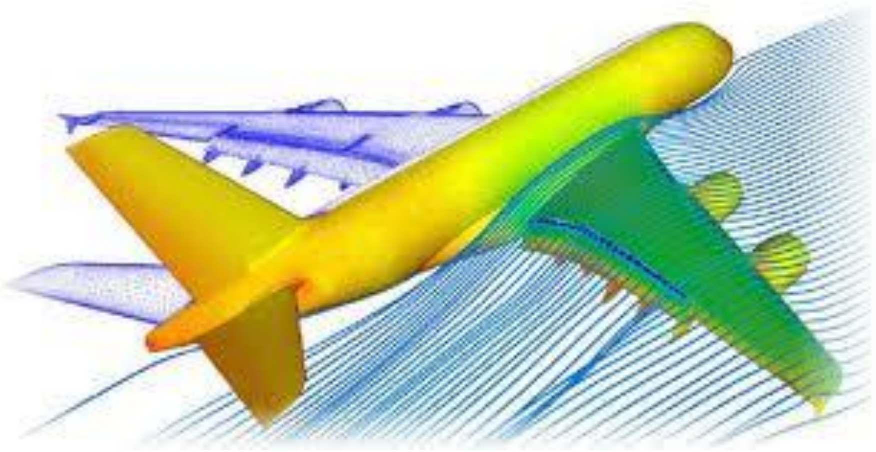
CFD analysis complements testing and experimentation by reducing total effort and cost required for experimentation and data acquisition

CFD Applications



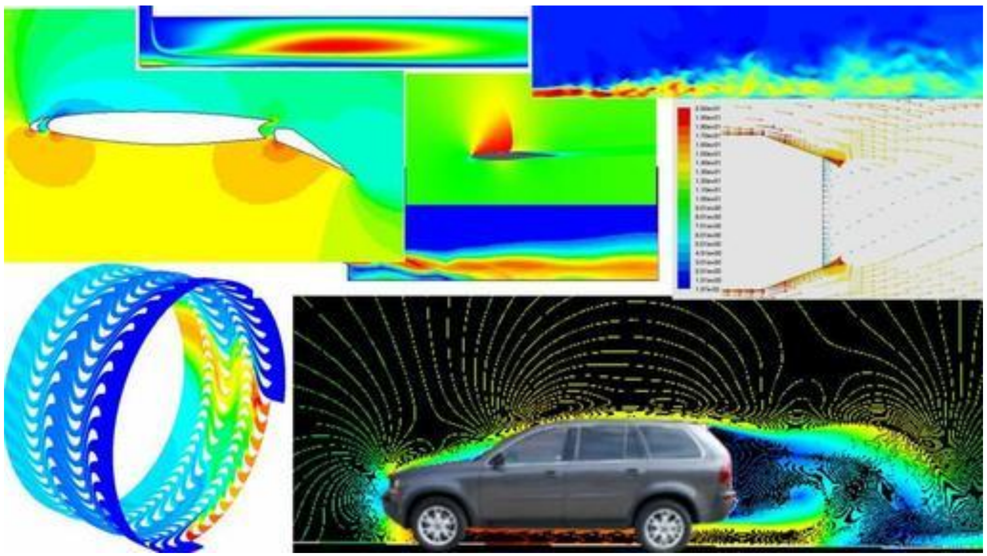
CFD Applications



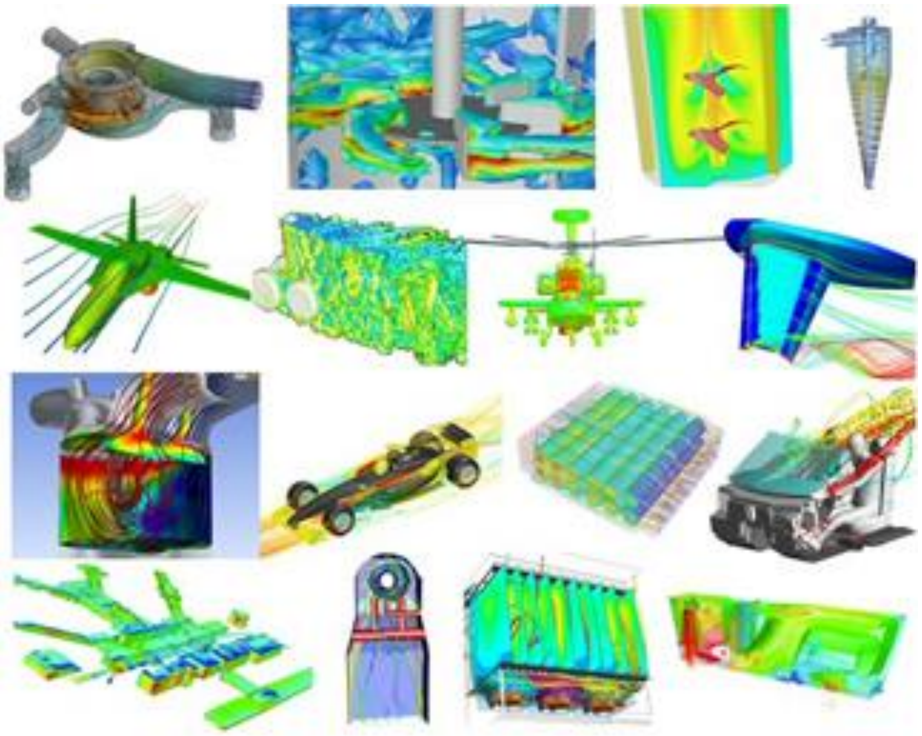


Introduction | CFD Approach | Pre-Processing | Solution | Post-Processing | Summary

CFD Applications



CFD Applications



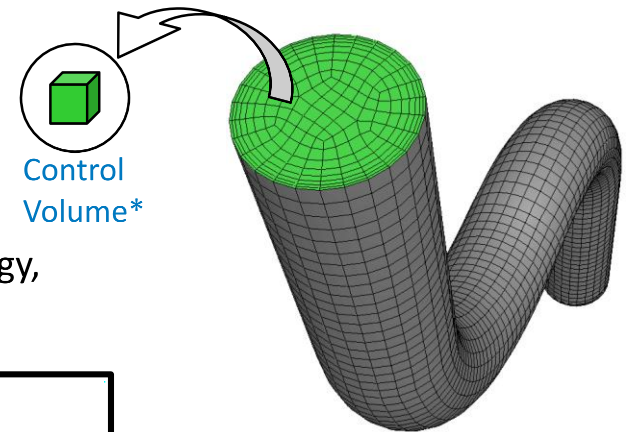
Introduction | CFD Approach | Pre-Processing | Solution | Post-Processing | Summary

How Does CFD Work?

ANSYS CFD solvers are based on the finite volume method

- Domain is discretized into a finite set of control volumes

- General conservation (transport) equations for mass, momentum, energy, species, etc. are solved on this set of control volumes



$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi\, dV}_{\text{Unsteady}} + \underbrace{\oint_A \rho\phi\mathbf{V}\cdot d\mathbf{A}}_{\text{Convection}} = \underbrace{\oint_A \Gamma_\phi \nabla\phi \cdot d\mathbf{A}}_{\text{Diffusion}} + \underbrace{\int_V S_\phi\, dV}_{\text{Generation}}$$

Equation	ϕ
Continuity	1
X momentum	u
Y momentum	v
Z momentum	w
Energy	h

- Partial differential equations are discretized into a system of algebraic equations

- All algebraic equations are then solved numerically to render the solution field

Step 1. Define Your Modeling Goals

- **What results are you looking for (i.e. pressure drop, mass flow rate), and how will they be used?**

- **What are your modeling options?**
  - What simplifying assumptions **can you make** (i.e. symmetry, periodicity)?
  - What simplifying assumptions do you **have to make**?
  - What physical models will need to be included in your analysis

- **What degree of accuracy is required?**

- **How quickly do you need the results?**

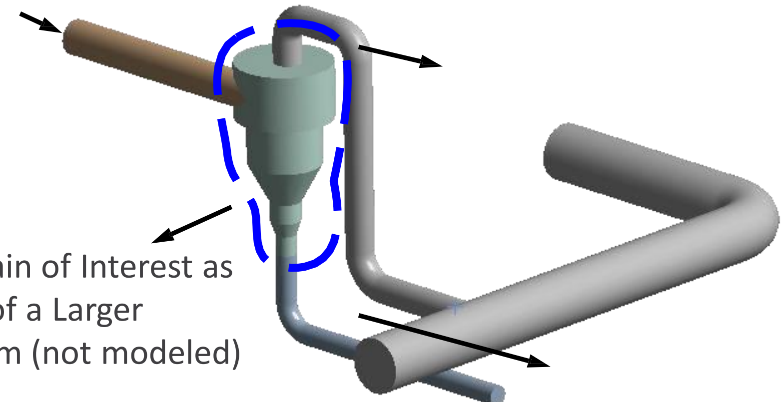
- **Is CFD an appropriate tool?**

Step 2. Identify the Domain You Will Model

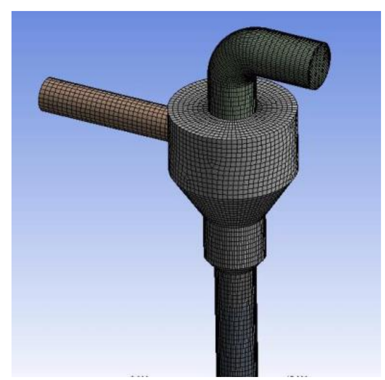
- **How will you isolate a piece of the complete physical system?**

- **Where will the computational domain begin and end?**
  - Do you have boundary condition information at these boundaries?
  - Can the boundary condition types accommodate that information?
  - Can you extend the domain to a point where reasonable data exists?

- **Can it be simplified or approximated as a 2D or axi-symmetric problem?**



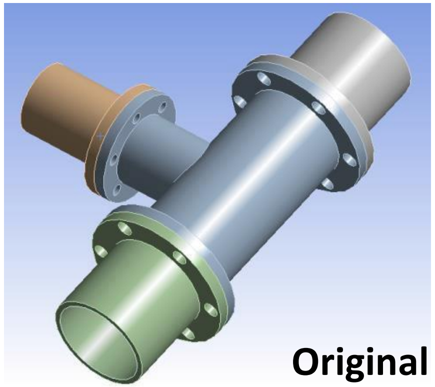
Domain of Interest as Part of a Larger System (not modeled)



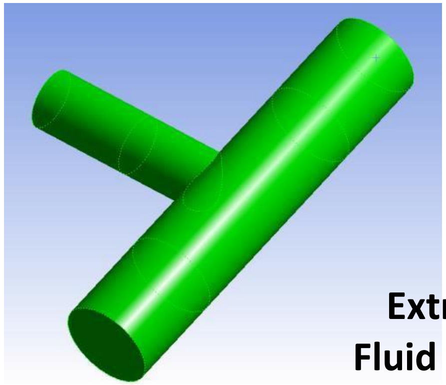
Domain of interest isolated and meshed for CFD simulation.

Step 3. Create a Solid Model of the Domain

- **How will you obtain a model of the *fluid* region?**
  - Make use of existing CAD models?
  - Extract the fluid region from a solid part?
  - Create from scratch?
- **Can you simplify the geometry?**
  - Remove unnecessary features that would complicate meshing ;fillets, bolts…?
  - Make use of symmetry or periodicity?
    - Are both the flow <u>and</u> boundary conditions symmetric / periodic?
- **Do you need to split the model so that boundary conditions or domains can be created?**



Original CAD Part

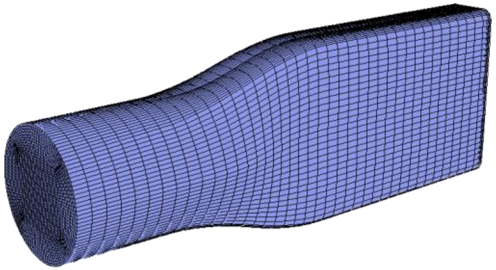


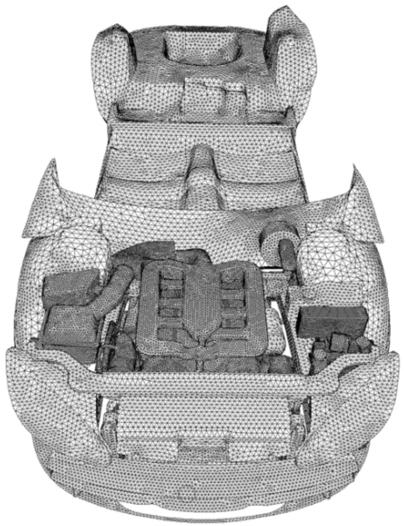
Extracted Fluid Region

Introduction | CFD Approach | Pre-Processing | Solution | Post-Processing | Summary

Step 4. Design and Create the Mesh

- **What degree of mesh resolution is required in each region of the domain?**
  - Can you predict regions of high gradients?
    - The mesh must resolve geometric features of interest and capture gradients of concern, e.g. velocity, pressure, temperature gradients
  - Will you use adaption to add resolution?
- **What type of mesh is most appropriate?**
  - How complex is the geometry?
  - Can you use a quad/hex mesh or is a tri/tet or hybrid mesh suitable?
  - Are non-conformal interfaces needed?
- **Do you have sufficient computer resources?**
  - How many cells/nodes are required?
  - How many physical models will be used?





Step 5. Set Up the Solver

- **For a given problem, you will need to:**

– Define material properties
  - Fluid
  - Solid
  - Mixture

– Select appropriate physical models
  - Turbulence, combustion, multiphase, etc.

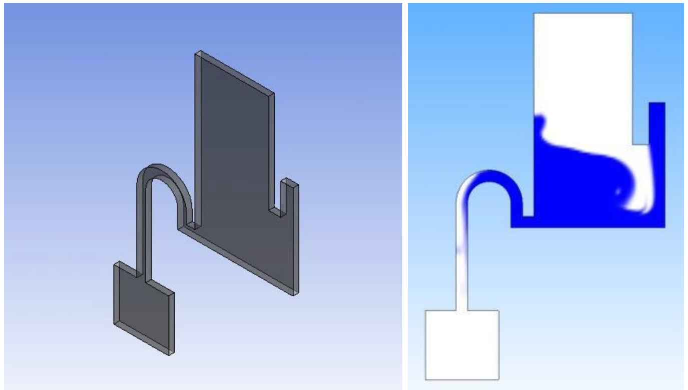
– Prescribe operating conditions

– Prescribe boundary conditions at all boundary zones

– Provide initial values or a previous solution

– Set up solver controls

– Set up convergence monitors



For complex problems solving a simplified or 2D problem will provide valuable experience with the models and solver settings for your problem in a short amount of time

Step 6. Compute the Solution

- **The discretized conservation equations are solved iteratively until convergence**
- **Convergence is reached when:**

– Changes in solution variables from one iteration to the next are negligible

  - Residuals provide a mechanism to help monitor this trend

– Overall property conservation is achieved

•Imbalances measure global conservation

– Quantities of interest (e.g. drag, pressure drop) have reached steady values

•Monitor points track quantities of interest

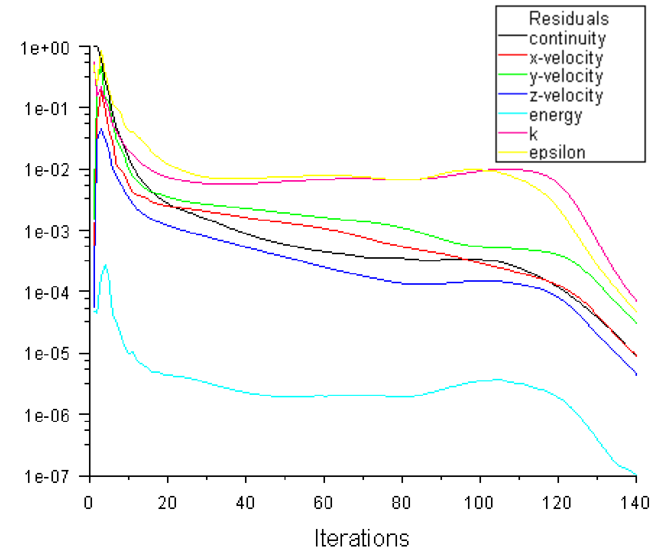
- **The accuracy of a *<u>converged</u>* solution is dependent upon:**

– Appropriateness and accuracy of physical models

– Assumptions made

– Mesh resolution and independence

– Numerical errors



A converged and mesh-independent solution on a well-posed problem will provide useful engineering results!

Step 7. Examine the Results

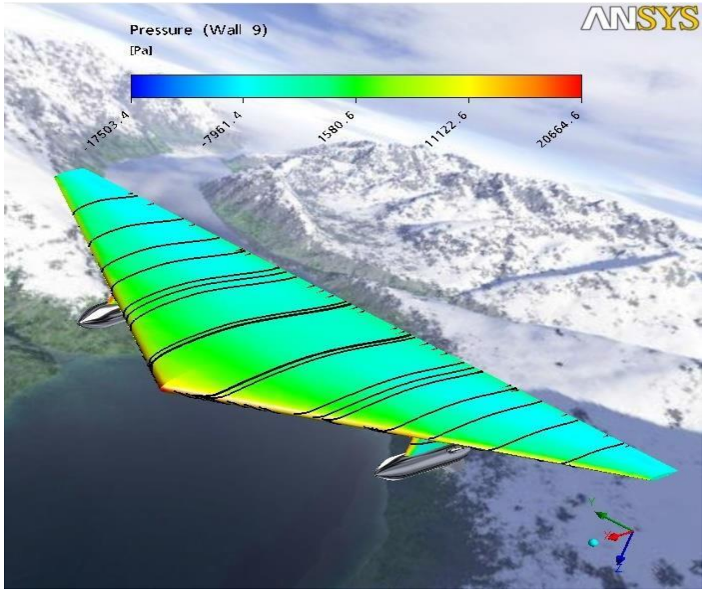
- **Examine the results to review solution and extract useful data**

– Visualization Tools can be used to answer such questions as:

  - What is the overall flow pattern?
  - Is there separation?
  - Where do shocks, shear layers, etc. form?
  - Are key flow features being resolved?

– Numerical Reporting Tools can be used to calculate quantitative results:

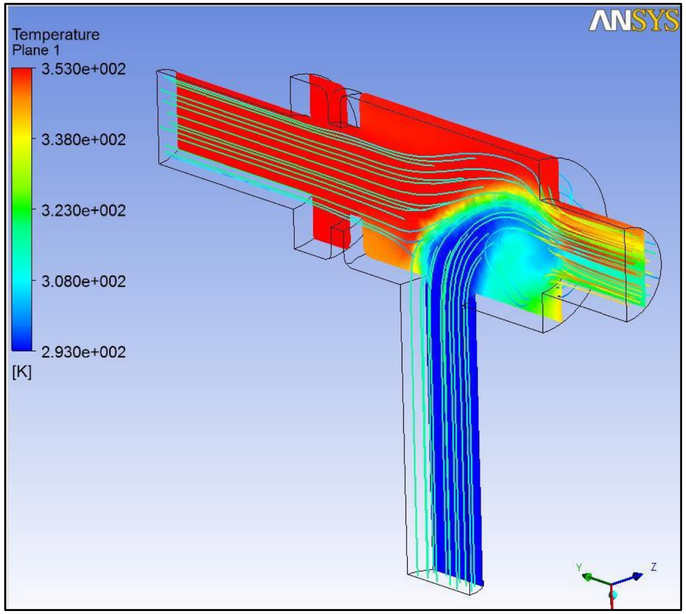
  - Forces and Moments
  - Average heat transfer coefficients
  - Surface and Volume integrated quantities
  - Flux Balances



Examine results to ensure correct physical behavior and conservation of mass energy and other conserved quantities. High residuals may be caused by just a few poor quality cells.

Step 8. Consider Revisions to the Model

- **Are the physical models appropriate?**
  - Is the flow turbulent?
  - Is the flow unsteady?
  - Are there compressibility effects?
  - Are there 3D effects?
- **Are the boundary conditions correct?**
  - Is the computational domain large enough?
  - Are boundary conditions appropriate?
  - Are boundary values reasonable?
- **Is the mesh adequate?**
  - Does the solution change significantly with a refined mesh, or is the solution mesh independent?
  - Does the mesh resolution of the geometry need to be improved?
  - Does the model contain poor quality cells?



High residuals may be caused by just a few poor quality cells

Use CFD with Other Tools to Maximize its Effect

Problem Identification

1. Define goals
2. Identify domain

Pre-Processing

3. Geometry
4. Mesh Physics
5. Solver Settings
6.

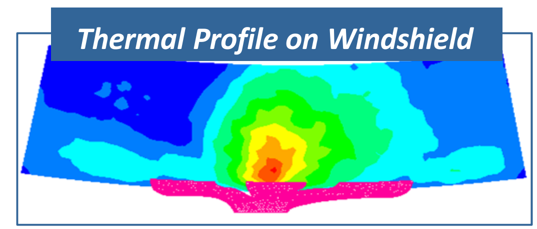
Solve

7. Compute solution

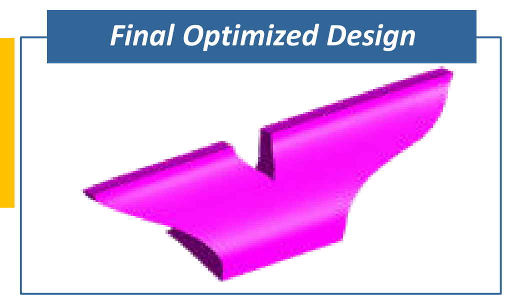
Post Processing

8. Examine results

9. Update Model



Automated Optimization of Windshield Defroster with ANSYS DesignXplorer



Prototype Testing → *Manufacturing*

Summary and Conclusions

- **Summary:**

– All CFD simulations (in all mainstream CFD software products) are approached using the steps just described

– Remember to first think about what the aims of the simulation are prior to creating the geometry and mesh

– Make sure the appropriate physical models are applied in the solver, and that the simulation is fully converged (more in a later lecture)

– Scrutinize the results, you may need to rework some of the earlier steps in light of the flow field obtained

1. **Define Your Modeling Goals**
2. Identify the Domain You Will Model
3. **Create a Geometric Model of the Domain**
4. Design and Create the Mesh
5. **Set Up the Solver Settings**
6. Compute the Solution
7. **Examine the Results**
8. Consider Revisions to the Model