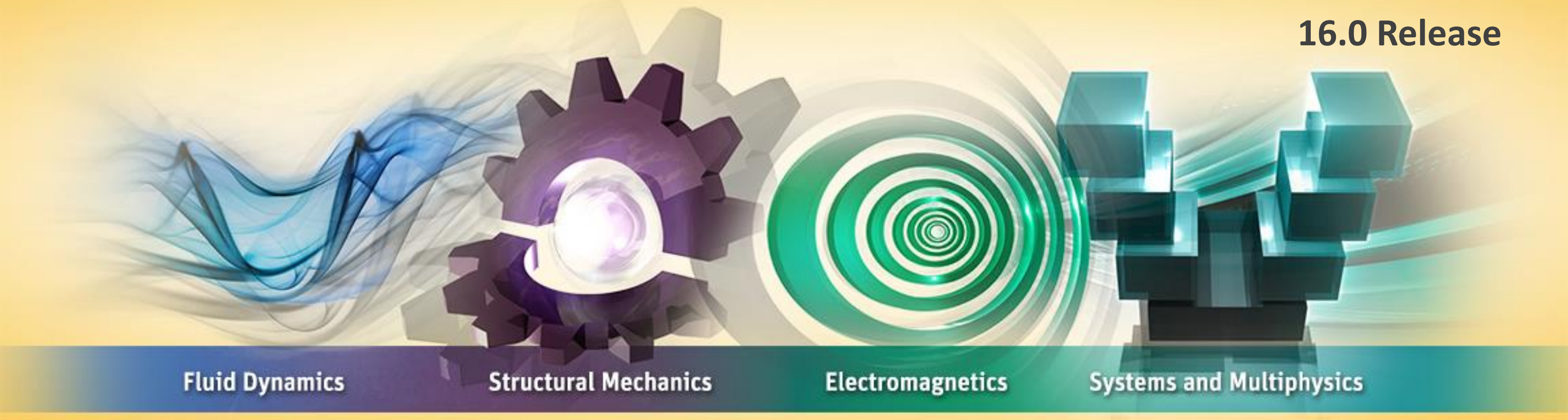
Lecture 3:
Introduction to CFD Methodology & CFX

16.0 Release



Introduction to ANSYS CFX

Introduction

Lecture Theme:

- All CFD simulations follow the same key stages. This lecture explains how to go from the original planning stage through to analysing the end results. After this CFX will be introduced

Learning Aims – you will learn:

- The basics of what CFD is and how it works
- The different steps involved in a successful CFD Project
- How to work with CFX

What is CFD

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

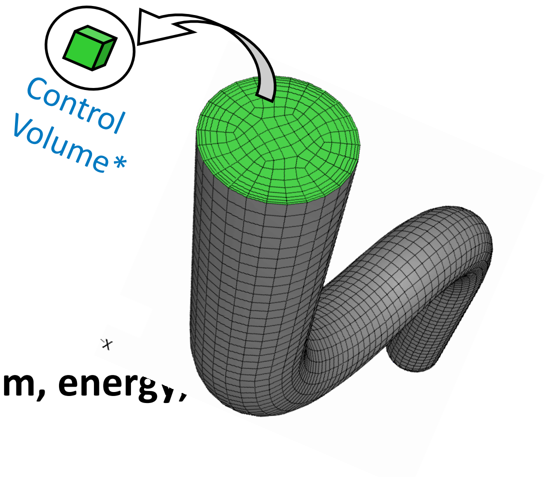
The equations for the conservation of mass, momentum, energy, etc.

CFD is used in all stages of the design process:

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

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

How Does CFD Work?



ANSYS CFD solvers are based on the finite volume method

- **Domain is discretized into a set of control volumes**
- **General conservation (transport) equations for mass, momentum, energy, species, etc. are solved on this set of control volumes**

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

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**

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

What degree of accuracy is required?

How quickly do you need the results?

Is CFD an appropriate tool?

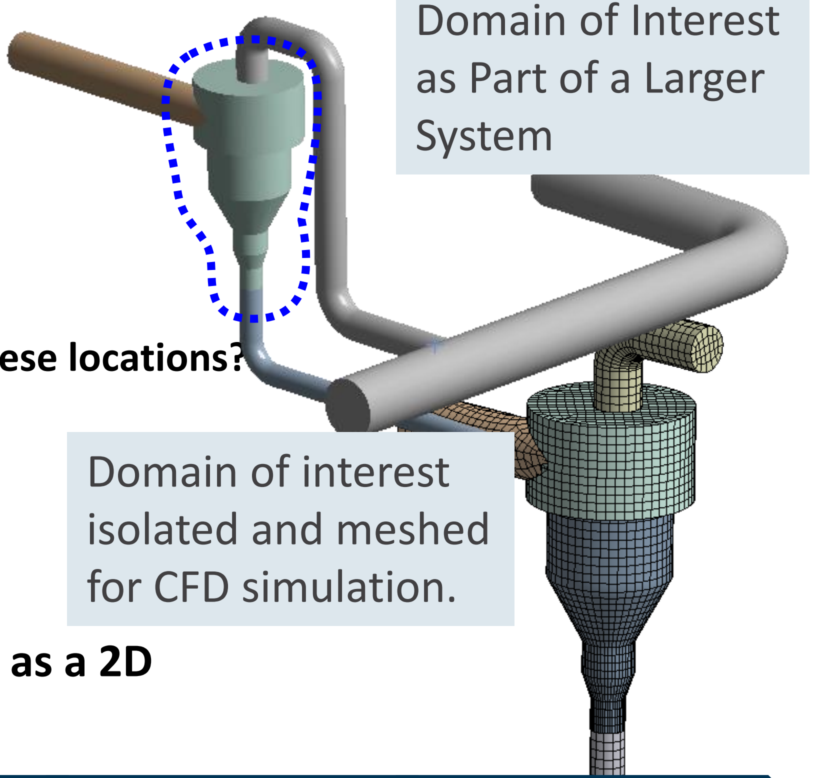
Step 2 – Identify the Domain to Model

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 locations?**
- **Can the boundary condition types accommodate that information?**
- **Can you extend the domain to a point where reasonable data exists?**

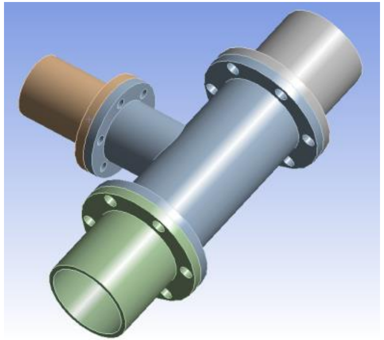
Can the problem be simplified or approximated as a 2D or axisymmetric problem?



Step 3 – Create a Solid Model

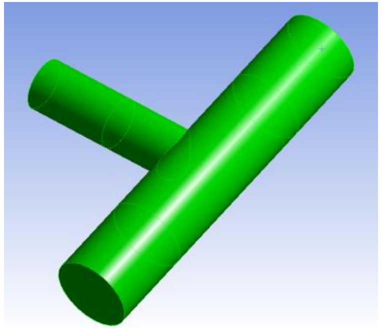
How will you obtain a model of the *fluid* region?

- Make use of existing CAD models?
- Extract the fluid region from a solid part?
- Create from scratch?

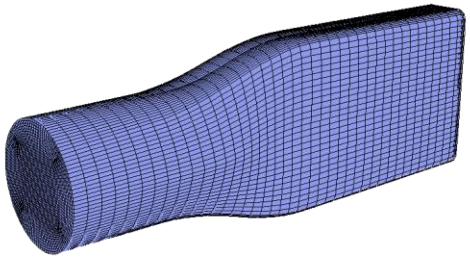


Can you simplify the geometry?

- Remove unnecessary features that would complicate meshing (fillets, bolts…)?
- Make use of symmetry or periodicity if both the solution and boundary conditions are symmetric / periodic?



Step 4 – Design and Create the Mesh



What is the required mesh resolution?

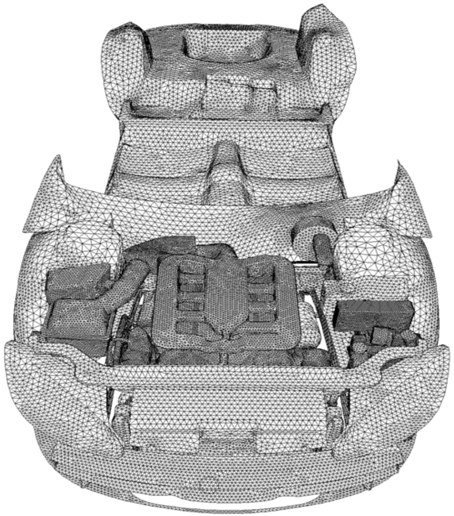
- Resolves geometric features of interest and capture gradients of concern, e.g. velocity, pressure, temperature gradients

What type of mesh is most appropriate?

- Can you use a hexahedral mesh?
- Are non-conformal interfaces needed?

Do you have sufficient CPU resources?

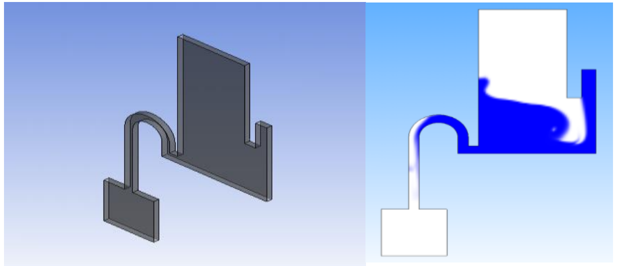
- How many cells/nodes are required?
- How many physical models will be used?



Step 5 – Set up the Solver

For a given problem you will need to:

- **Define material properties**
  - **Fluid, Solid, or Mixture**
- **Select appropriate physical models**
  - **Turbulence, combustion, multiphase, etc.**
- **Prescribe boundary conditions**
- **Provide initial values of 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

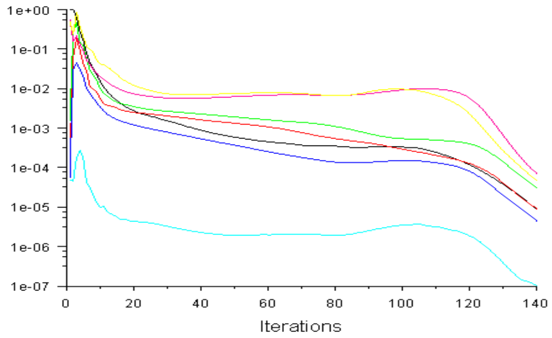
Step 6 – Compute the Solution

Convergence is reached when:

- **Changes from one iteration to the next are negligible**
  - **Residuals monitor this trend**
- **Overall property conservation is achieved**
  - **Imbalances measure global conservation**
- **Quantities of interest have reached steady values**
  - **Monitor points track quantities of interest**

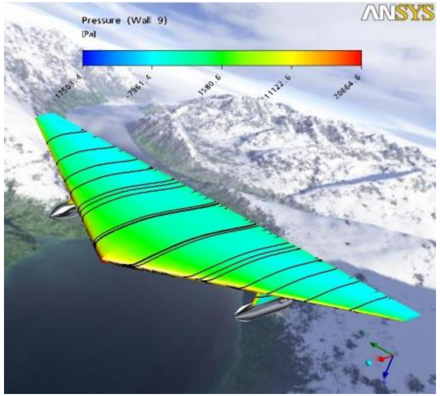
The accuracy of a *converged* solution depends on

- **Appropriateness and accuracy of physical models**
- **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

Use Visualization tools to answer:

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

Calculate quantitative results:

- **Forces and moments**
- **Average heat transfer coefficients**
- **Surface and volume integrated quantities**

Examine results to ensure property conservation and correct physical behavior. High residuals may be caused by just a few poor quality cells.

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

Step 8 – Consider Model Revisions

Are the physical models appropriate?

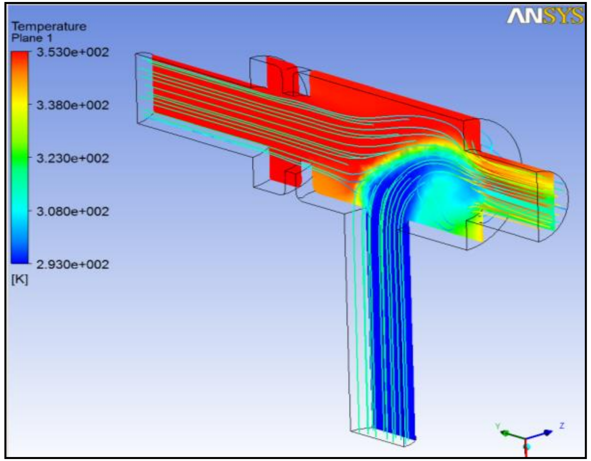
- Is the flow turbulent or unsteady?
- Any compressibility effects or 3D effects?

Are the boundary conditions correct?

- Computational domain large enough?
- Boundary values reasonable?

Is the mesh adequate?

- Does the solution change with a refined mesh, or is the solution mesh independent?



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

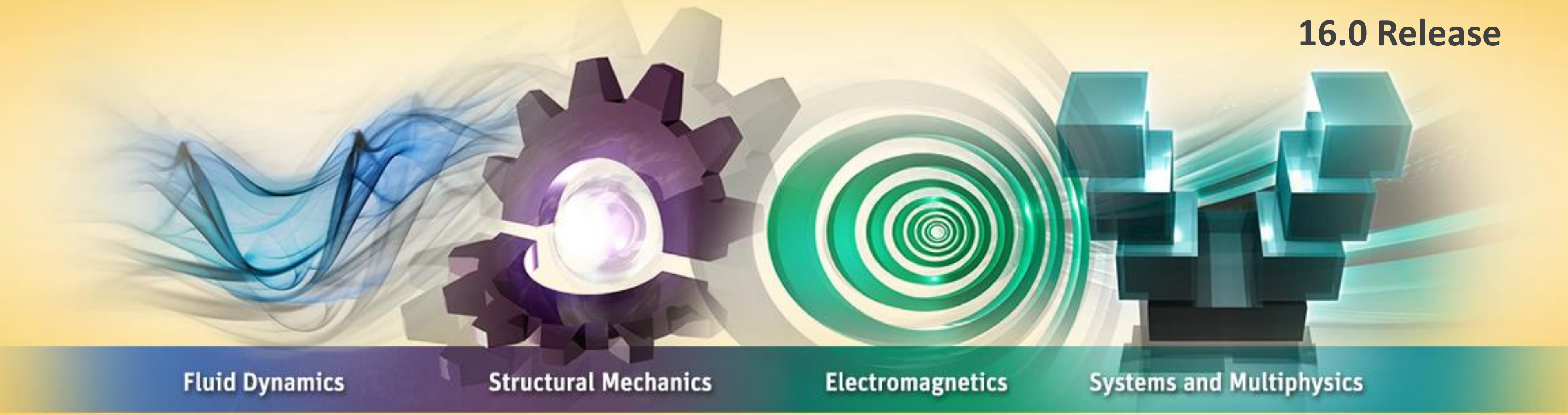
Introduction | CFD Approach | Pre-Processing | Solution | Summary

Summary and Conclusions

- All CFD simulations are approached using the steps just described
- Remember to think first about what the aims of the simulation are prior to creating the geometry and mesh
- Make sure the solver is applying the appropriate physical models and that the simulation is fully converged
- Examine the results carefully
  - You may need to rework some of the earlier steps in light of the flow field predicted by CFD
- Next step: We'll look at an overview of the workflow in CFX and follow that with a demonstration of CFX in action

ANSYS CFX

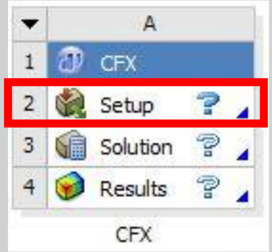
16.0 Release



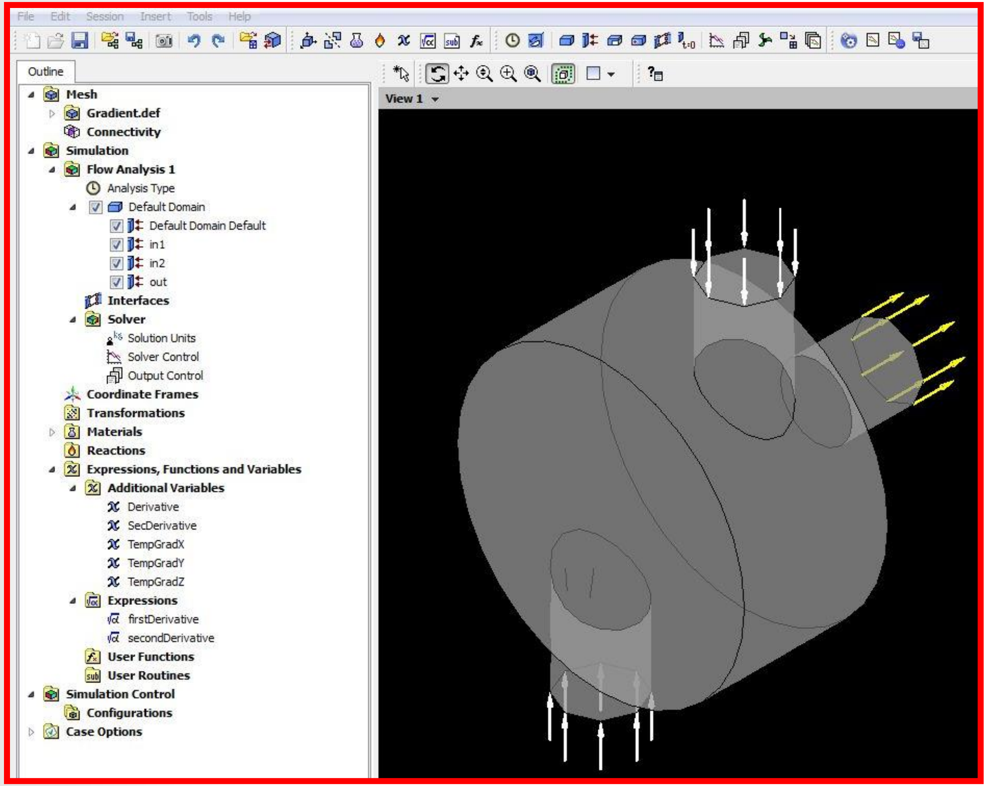
Introduction to ANSYS CFX

Realize Your Product Promise®

CFX Pre – Workflow



- Follow the Outline tree from top to bottom
- Double-click entries in the tree to edit
- Right-click on entries in the tree to insert new items or perform operations

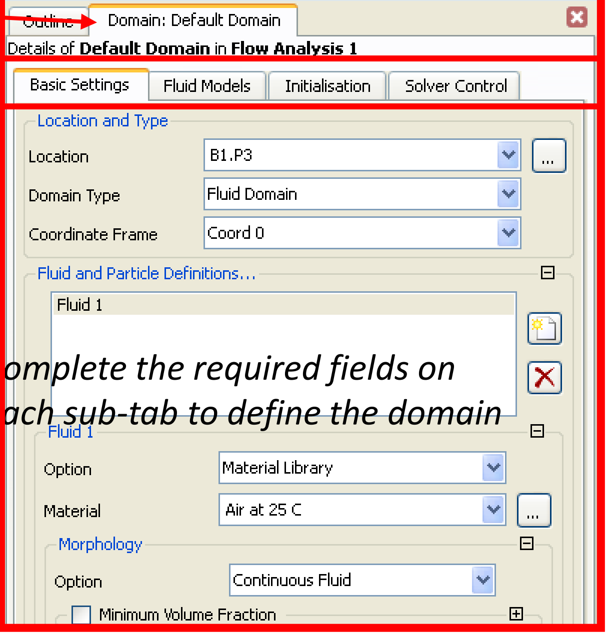


Introduction | Launching CFX | Workflow | Files | Demo

CFX Pre – Workflow

- **Load Mesh**
- **Define Domain Properties**
  - **Right-click on 'Flow Analysis 1' to insert a new domain**
- **Create Boundary Conditions**
  - **Right-click on the domain to insert BCs**
- **Define Solver Settings**
  - **Right-click on Solver Control and pick Edit**

Inserting an item opens a new tab panel with properties.
TIP: Work through all tabs from left to right



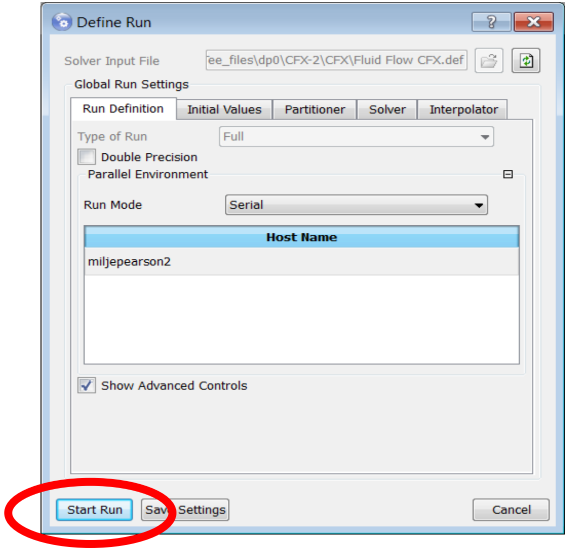
Complete the required fields on each sub-tab to define the domain

CFX-Pre – Workflow Example

Start Solver

- Close CFX-Pre
- Right-click on Solution and select Edit to open Solver Manager

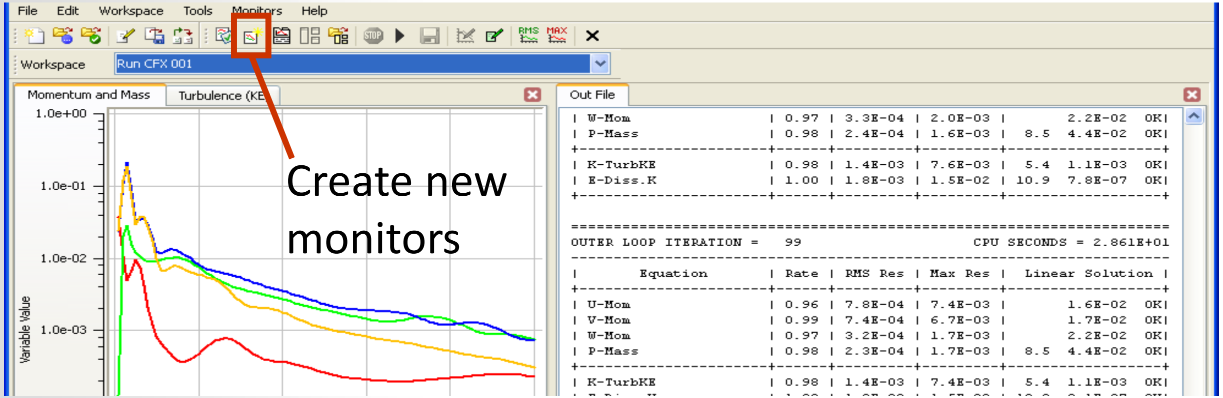
CFX



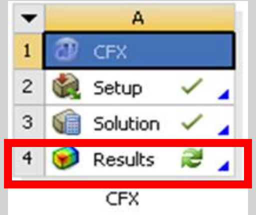
Solver Manager

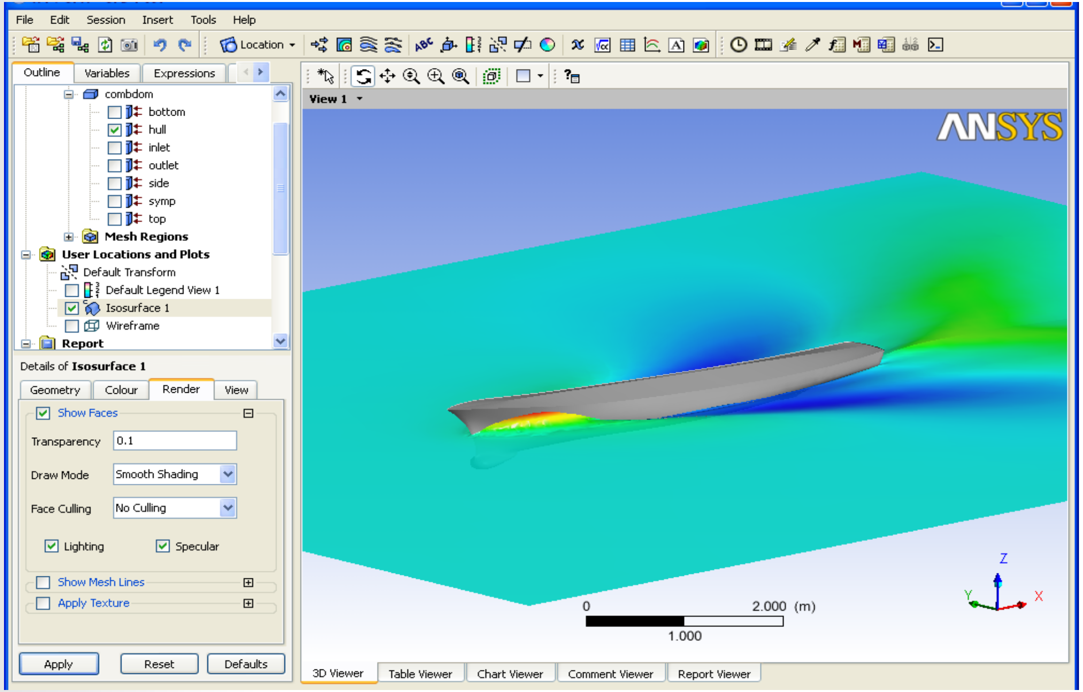
CFX Solver Manager

- Solution Monitors monitor the convergence of the solver
  - Plot residuals, imbalances, monitor points, forces, fluxes…
- Text output from the Solver shows lots of info in here
  - Can also view the .out file in a text editor



CFD-Post





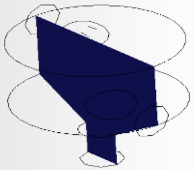
Outline tree displays all post-processing objects. Double-click to edit in the Details Pane

Introduction | Launching CFX | Workflow | Files | Demo

CFD-Post

General Workflow

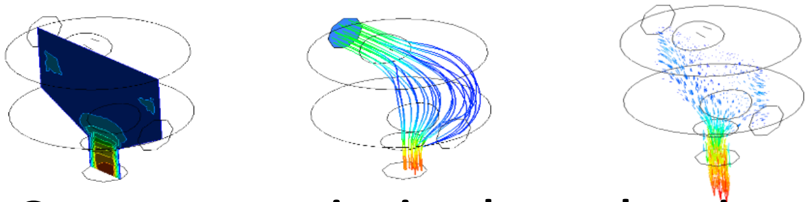
- Prepare locations for data extraction or plot generation
  - e.g. Planes, Isosurface



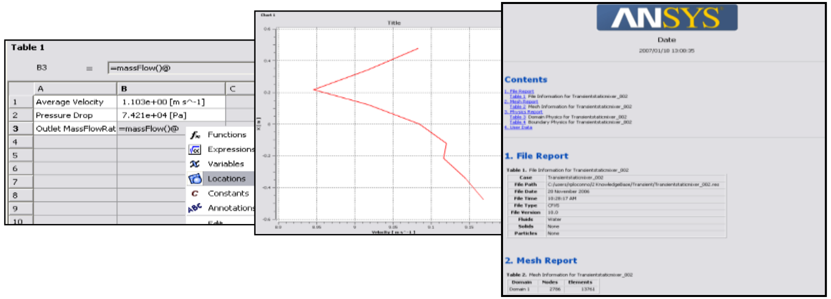
- Create variables, expressions which will be used to extract data
  - e.g. Drag, pressure ratio

Fresh Liq	(1-tracer)
Mix Effectiveness	volumeAve(Fresh Liq)@Digester
Reference Pressure	1 [atm]
t	Time

- Generate qualitative data at locations



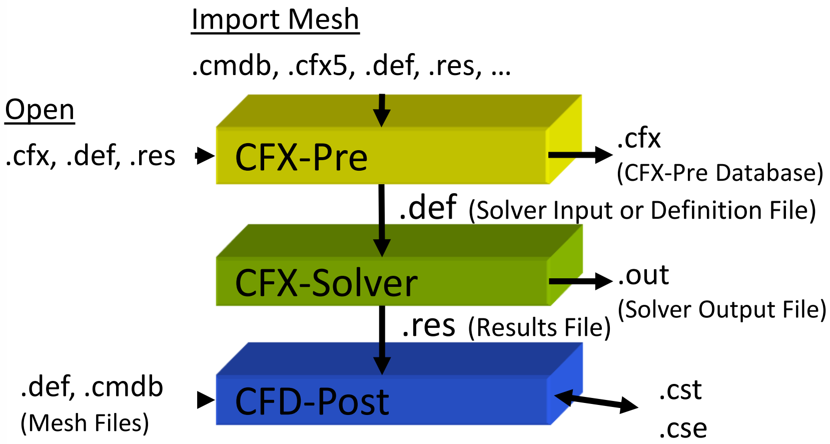
- Generate quantitative data at locations



- Generate Reports

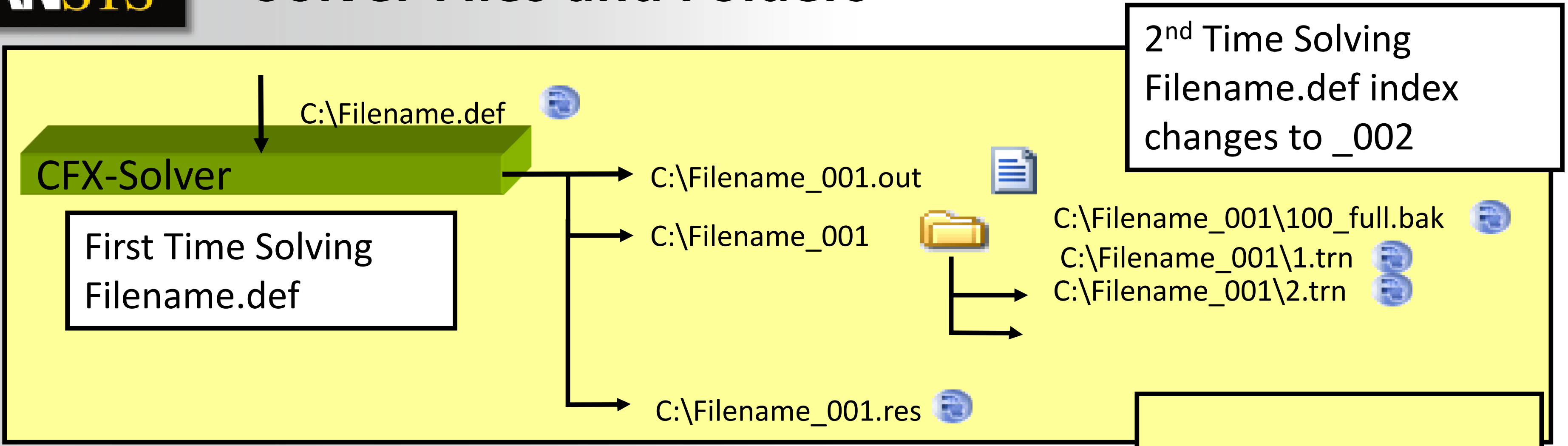
Common File Types

- .cfx files contain mesh and physics data and can be opened by CFX-Pre
- .def files contain mesh and physics data and can be run in the Solver
- .res files contain mesh, physics and results data and can be opened in CFD-Post



- CFX-Pre can open .def and .res files to recover mesh and physics.
- The CFX-Solver can use .res files to continue a run or as an initial guess
- CFD-Post can also open .def and .cmdb files to view the mesh

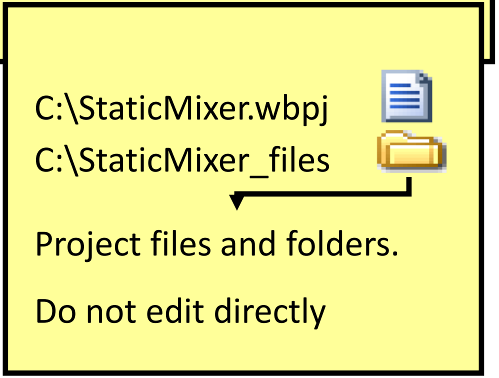
Solver Files and Folders



Standalone files are saved to your current working directory

When running in Workbench only the project file (.wbpj) is saved to the current working directory

- **All other files are saved to a *name_files* subdirectory**



Introduction | Launching CFX | Workflow | Files | Demo

DEMO & Workshop 01 – Mixing Tee

Demonstration of Static Mixer

Introductory tutorial for CFX

- Starting from existing mesh
  - generated in earlier tutorial during the DM / Meshing session
- Model set-up, solution and post-processing
- Mixing of cold and hot water in a T-piece
- How well do the fluids mix?
- What are the pressure drops?