Introduction into Simulation CFD for the Turbomachinery applications

Amal Cheikh rouhou

Technical Support Specialist, M.SC in Mechanical Engineering

Make sure to download the Handout detailing the Model Setup under:
https://autodesk.app.box.com/v/CFD

Introduction into Simulation CFD for the Turbomachinery applications

Amal Cheikh rouhou

Technical Support Specialist, M.SC in Mechanical Engineering

Key learning objectives

1. Understand **how CFD impacts** the **design process**

2. Gain exposure to the Autodesk Simulation CFD interface and **simulation workflow**

3. Learn how to use the powerful **post-processing tools** to interpret simulation results and to make decisions

4. Discover CFD **best practices** and **pitfalls** in Turbomachinery

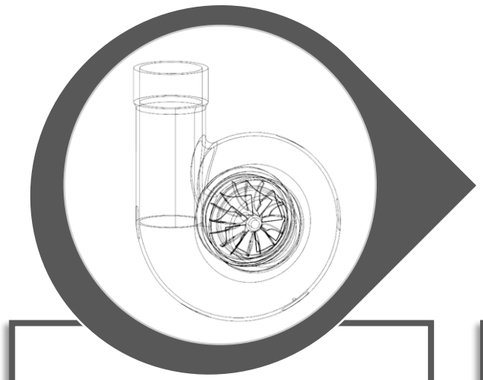
Agenda

- Impact of Simulation CFD on the design process
- Simulation CFD for Turbomachinery
- Exercise: Centrifugal pumps in Simulation CFD
- Questions & Answers

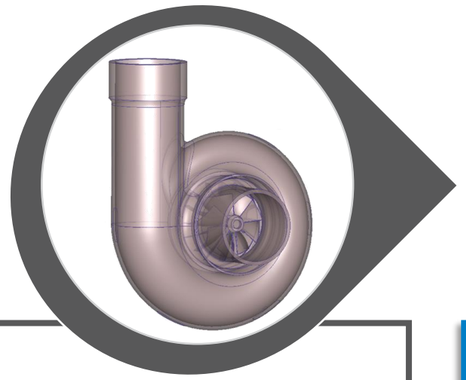
Impact of Simulation CFD on the design process

Traditional Design process

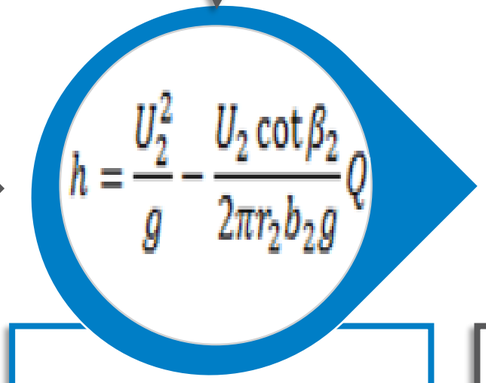
Fail



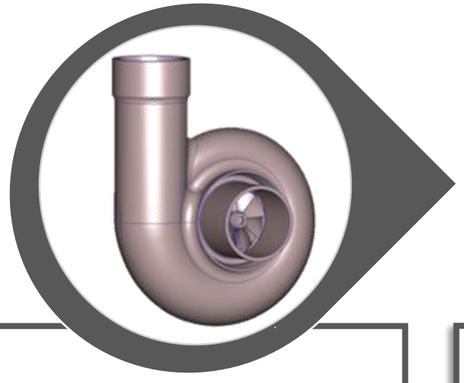
Concept



First CAD



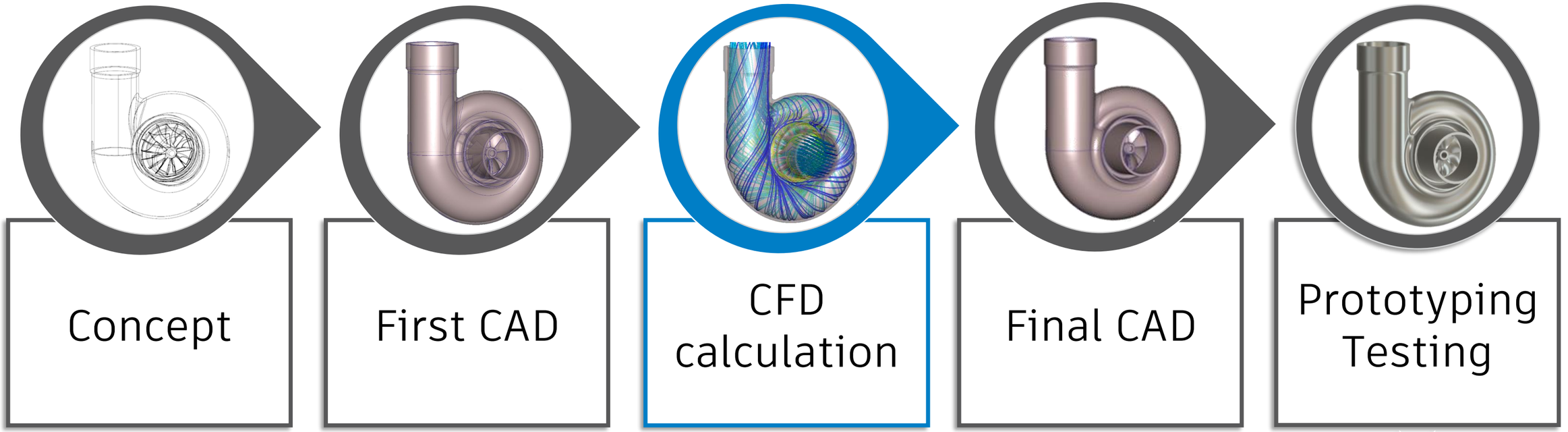
Manual calculation



Final CAD

Prototyping Testing

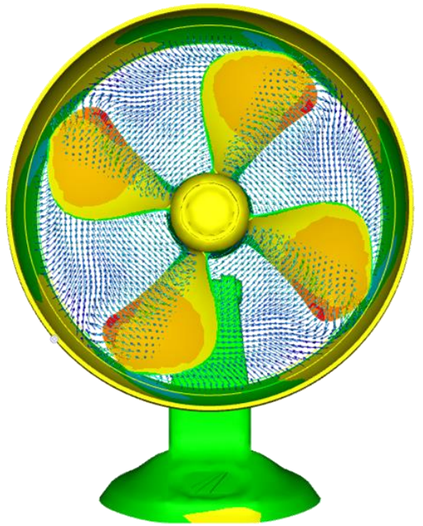
CFD Design process

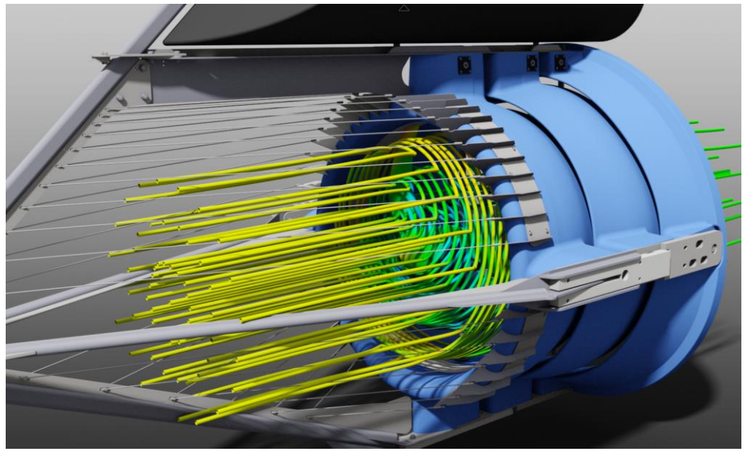


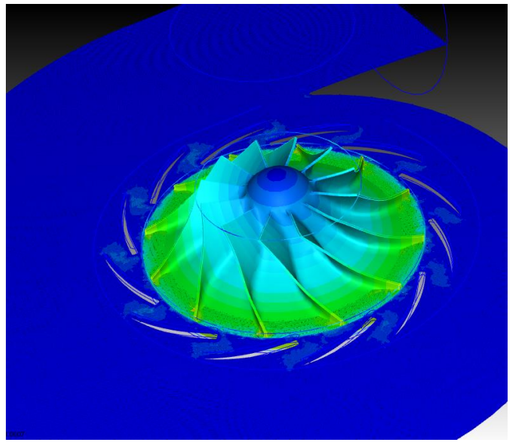
Simulation CFD for Turbomachinery

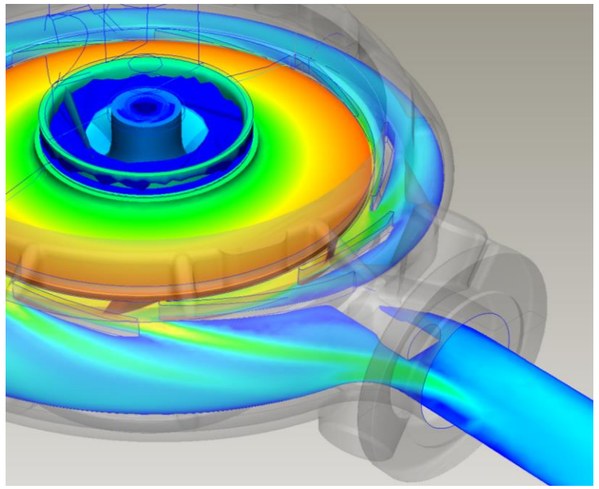
CFD benefits for Turbomachinery

- Performance prediction
- Visualization of flow features
- Energy loss calculation
- Design validation & optimization





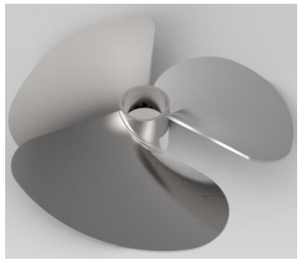
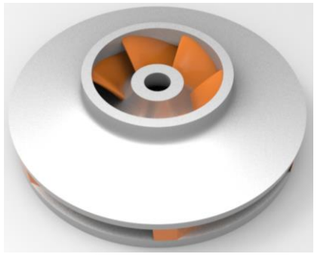




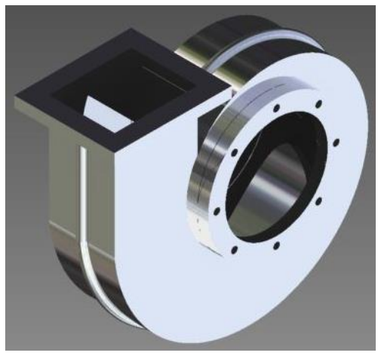
Exercise: Centrifugal pumps in Simulation CFD

Centrifugal pump: What is it?

- Impeller:
  - Rotating device
  - Has blades fixed on a hub plate

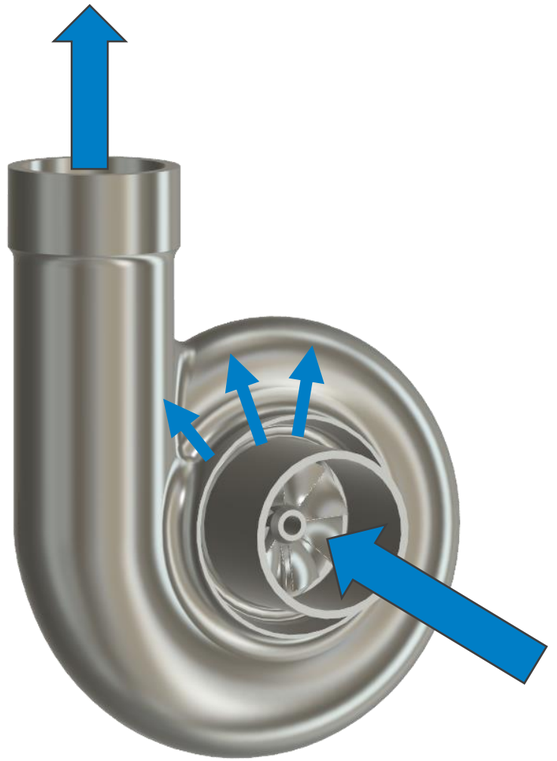





- Volute/Casing:
  - Static device
  - Increasing cross sectional area

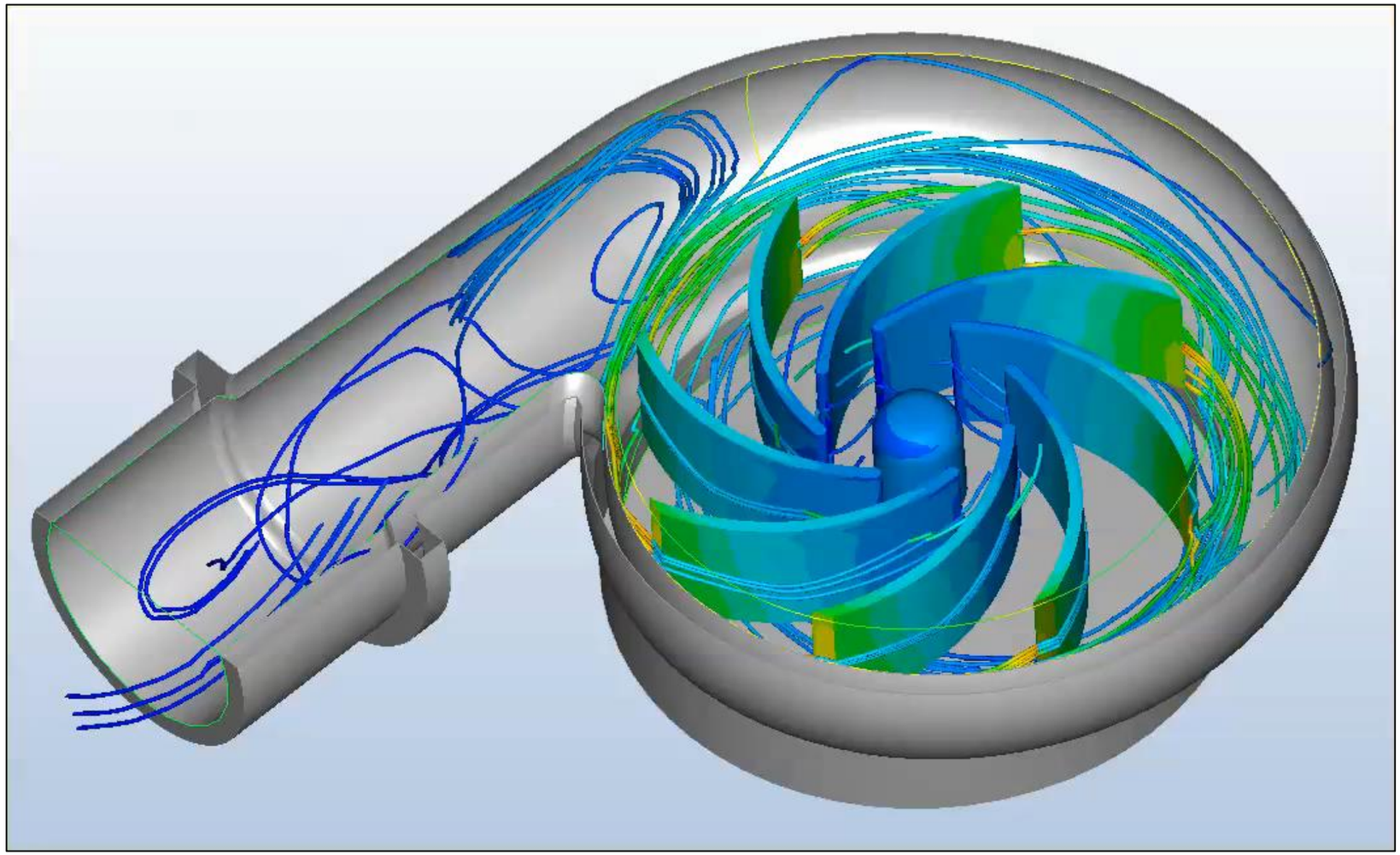




Centrifugal pump: How does it work?

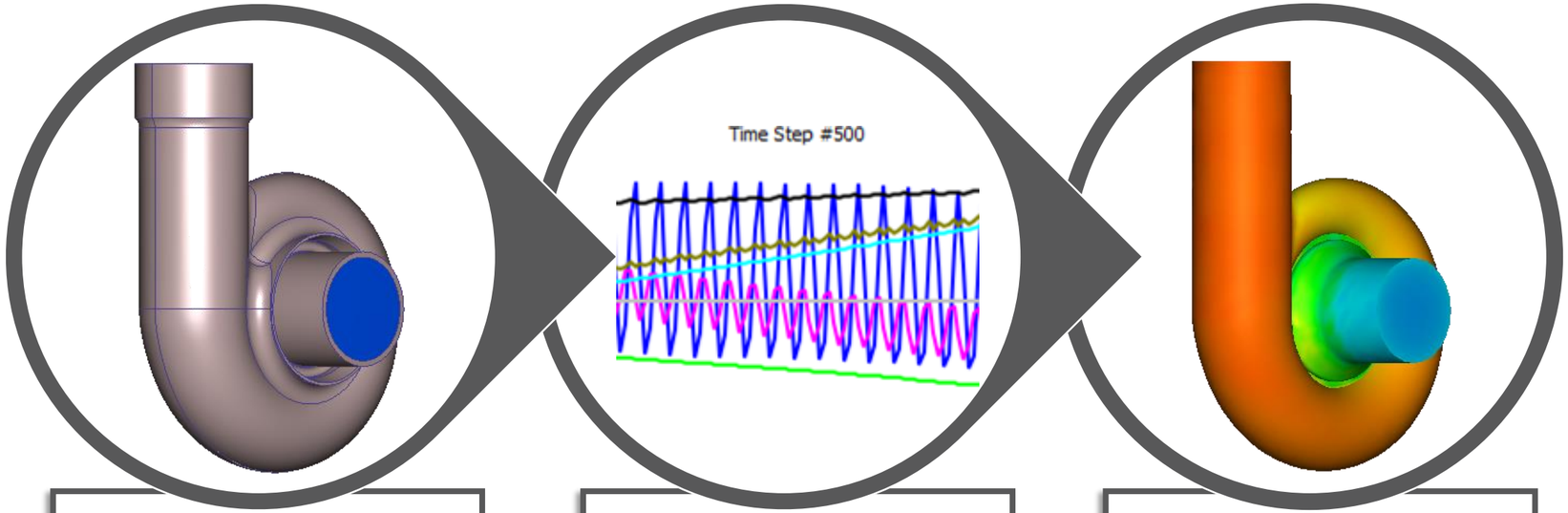
1. Impeller rotates (Electric motor)
2. It creates low pressure at the inlet
3. The low pressure helps suck fluid
4. The fluid is pushed radially from the impeller to the volute



Centrifugal pump: How does it work?



CFD Workflow - Steps

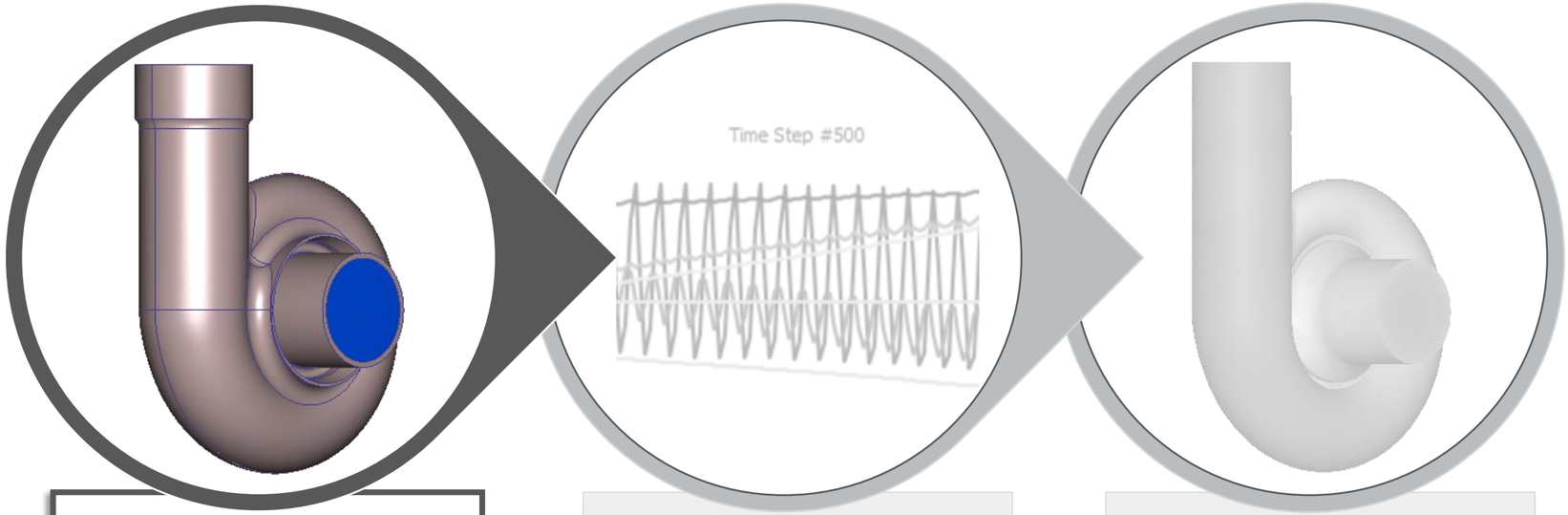


CAD
Materials
BC
Meshing

Solving

Results
Validation

Centrifugal pump: The CAD Model



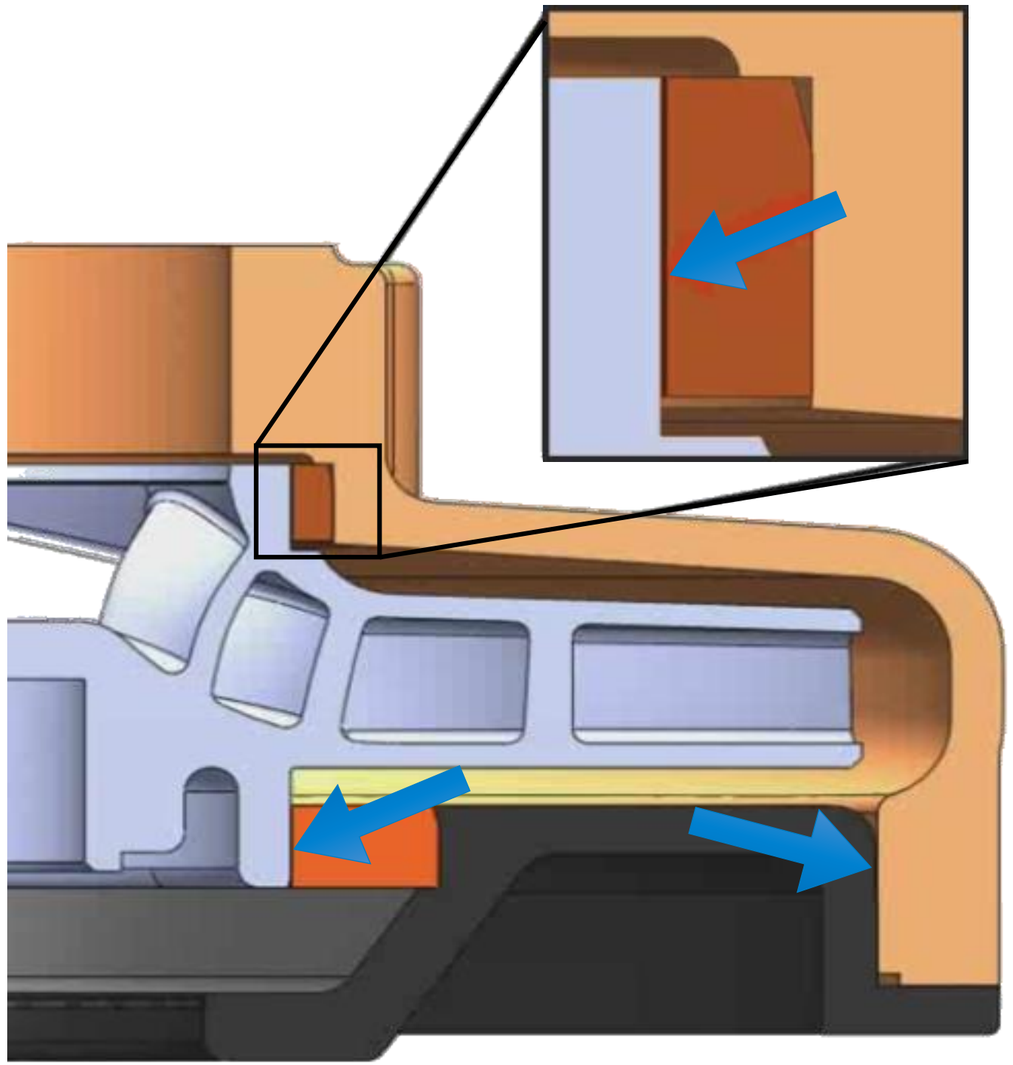
CAD
Materials
BC
Meshing

Solving

Results
Validation

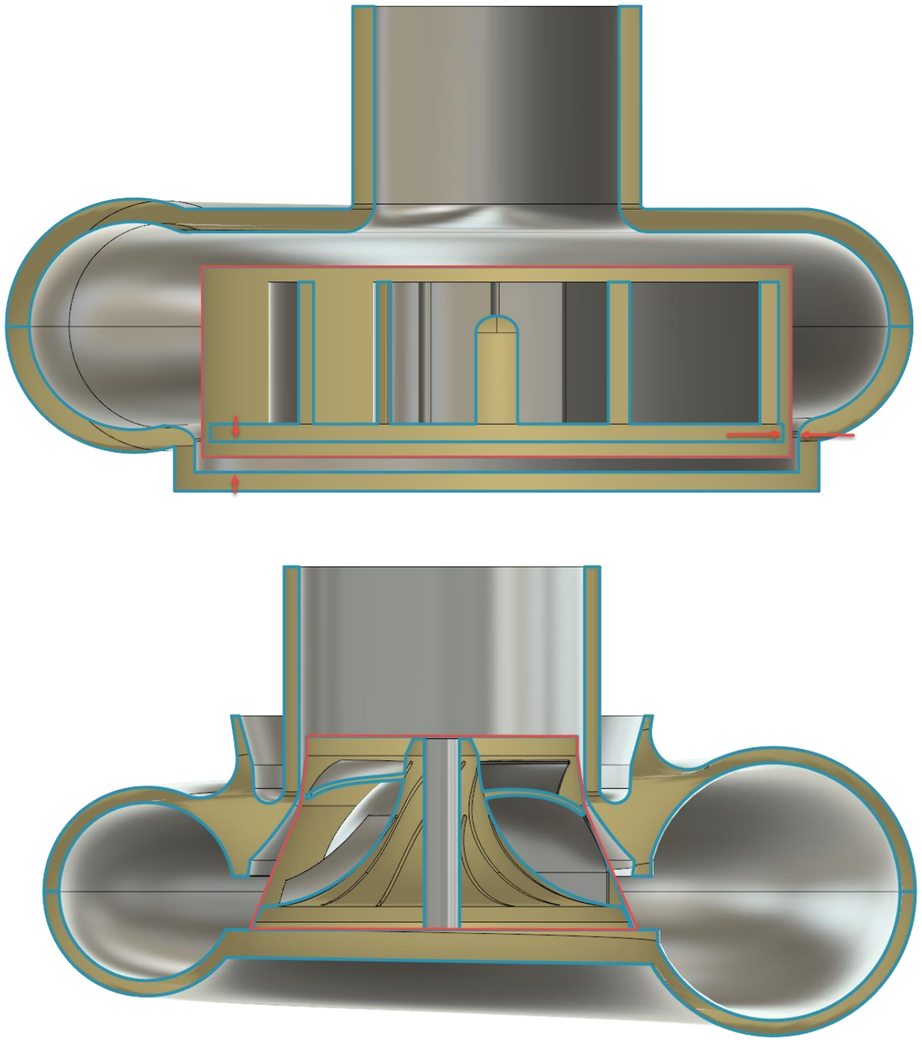
Centrifugal pump: The CAD Model

- **Simplifications**
  - Remove the shaft from the impeller
  - Fill in small gaps
  - Remove radii
  - Remove useless features



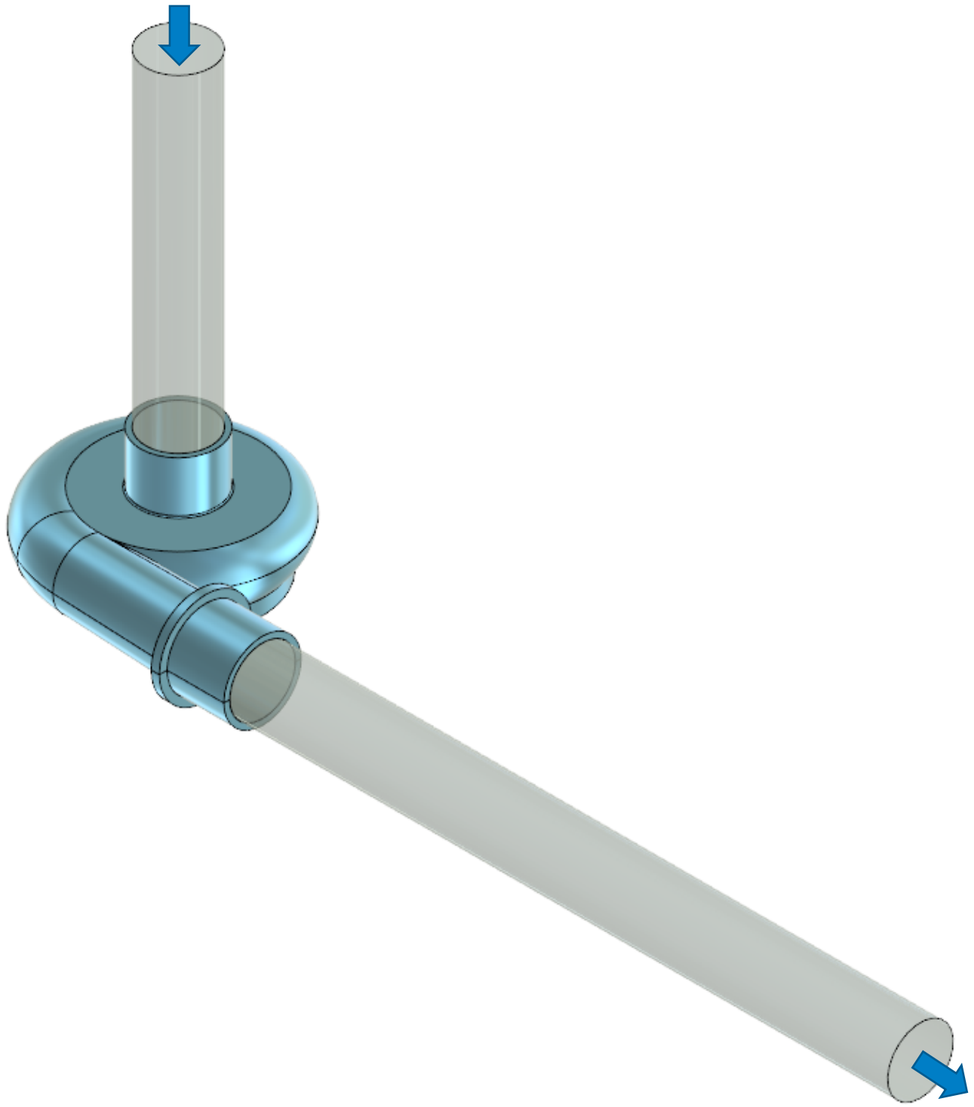
Centrifugal pump: The CAD Model

- **Rotating Region (RR)**
  - Positioned halfway between the impeller and the wall of the volute
  - Envelopes the impeller and small amount of fluid
  - Touches the wall if the impeller does the same (or is very close)

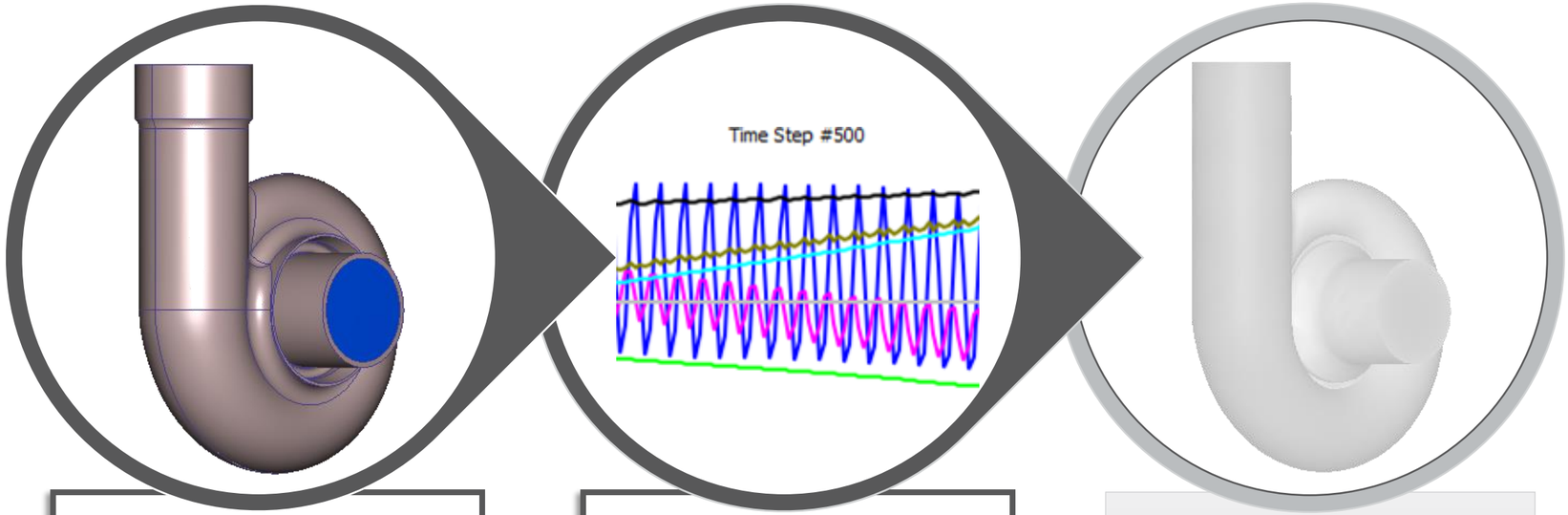


Centrifugal pump: The CAD Model

- **Openings extension**
  - Extend inlet and outlet
    - Inlet = 5X diameter in length
    - Outlet = 10X diameter in length



Centrifugal pump: Model Setup (Live Demo)



CAD
Materials
BC
Meshing

Solving

Results
Validation

AUTODESK® **CFD MOTION** 2018

Things to avoid

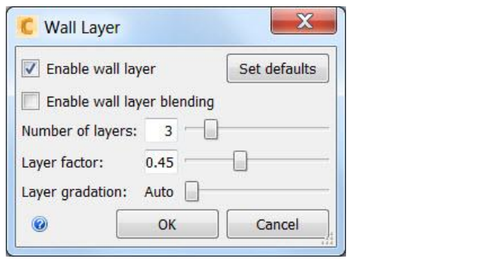
- **Rotating region material**
  The impulsive start-up: When the full rotational speed is specified from the beginning

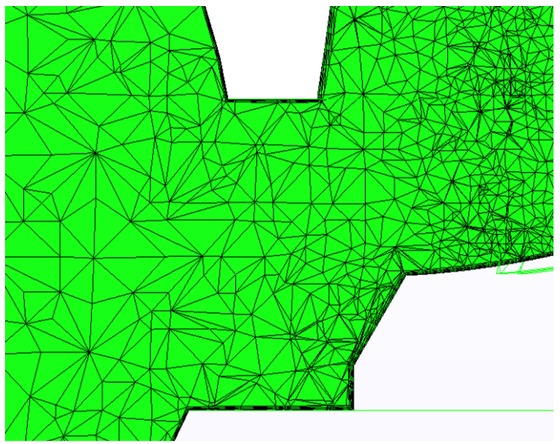
- **Boundary conditions (BC)**
  Direct application of non-zero pressure or flow rate at the discharge

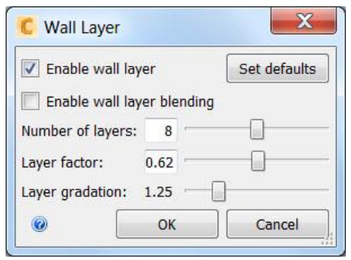
- **Meshing**
  Defining a non-adequate mesh. Rotating region analyses can be especially mesh sensitive

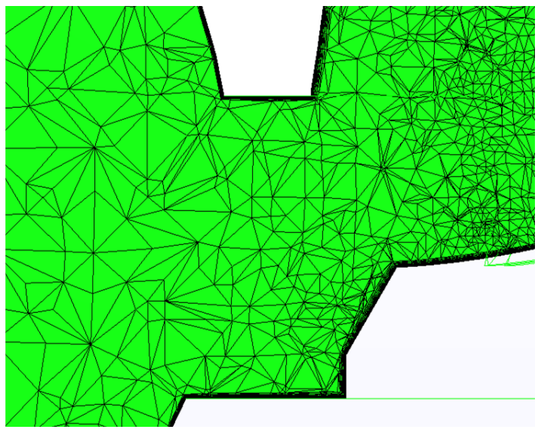
Tips & Tricks: Meshing

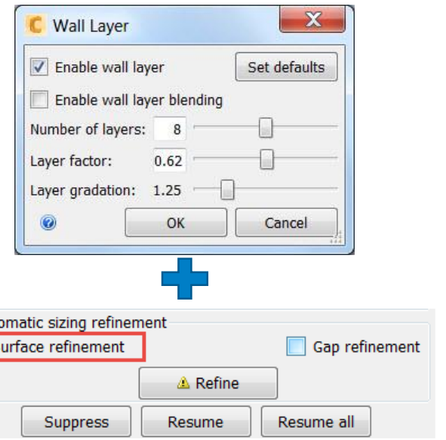
- Use the adequate wall layers settings that give:
  - Nodal Aspect Ratio < 100
  - Wall distance Y+ ~ 1 (SST k-w)

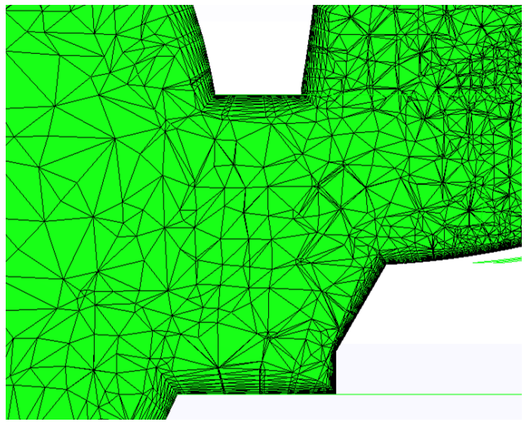






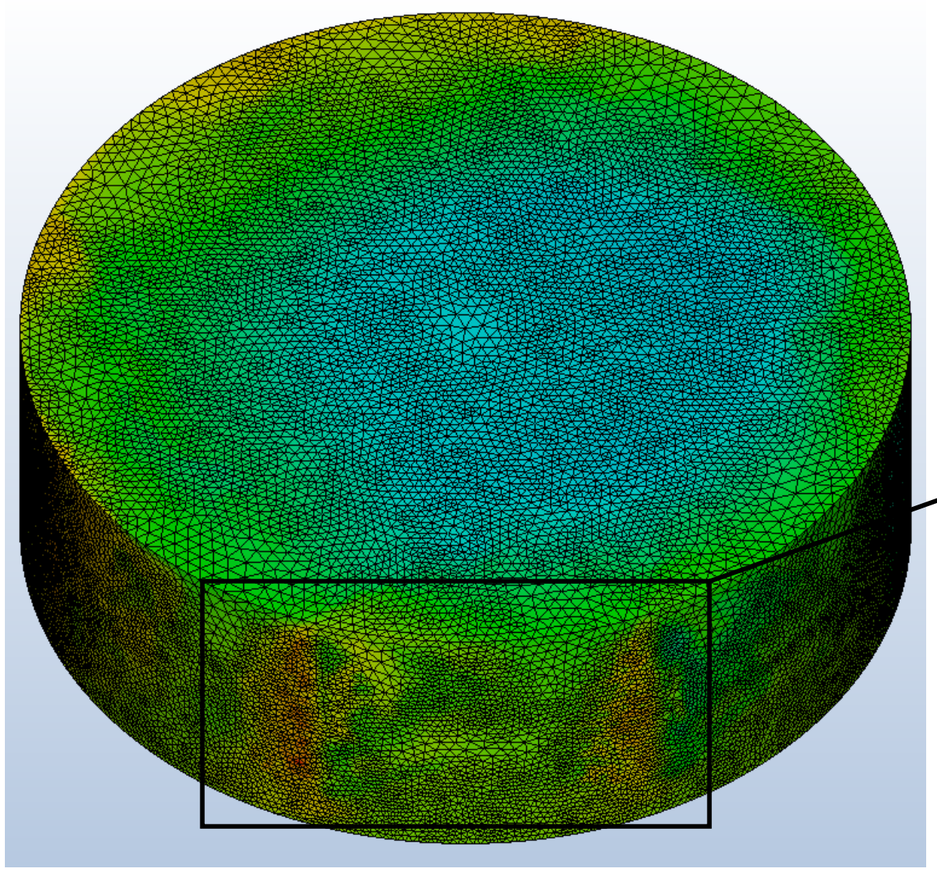




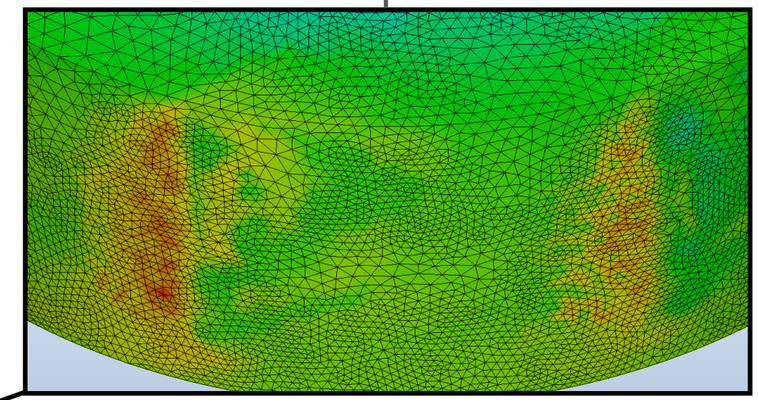


Tips & Tricks: Meshing

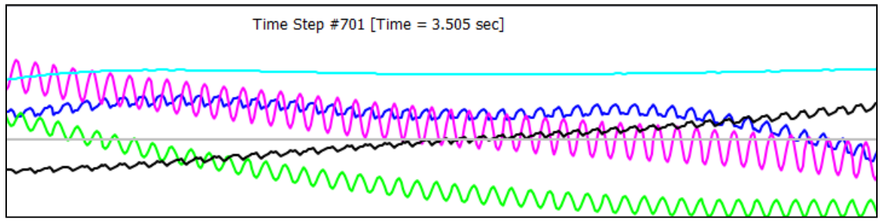
- Make sure to have a uniform mesh over the rotating region



hotspots



Convergence instability

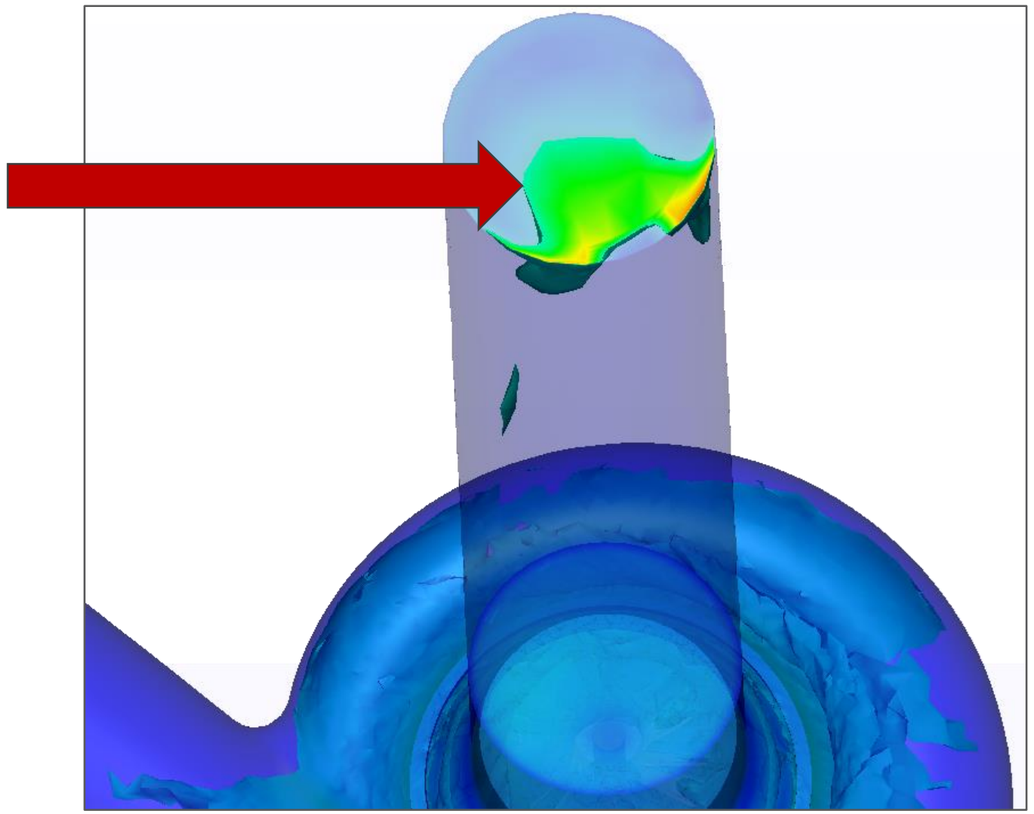


Tips & Tricks: Flag

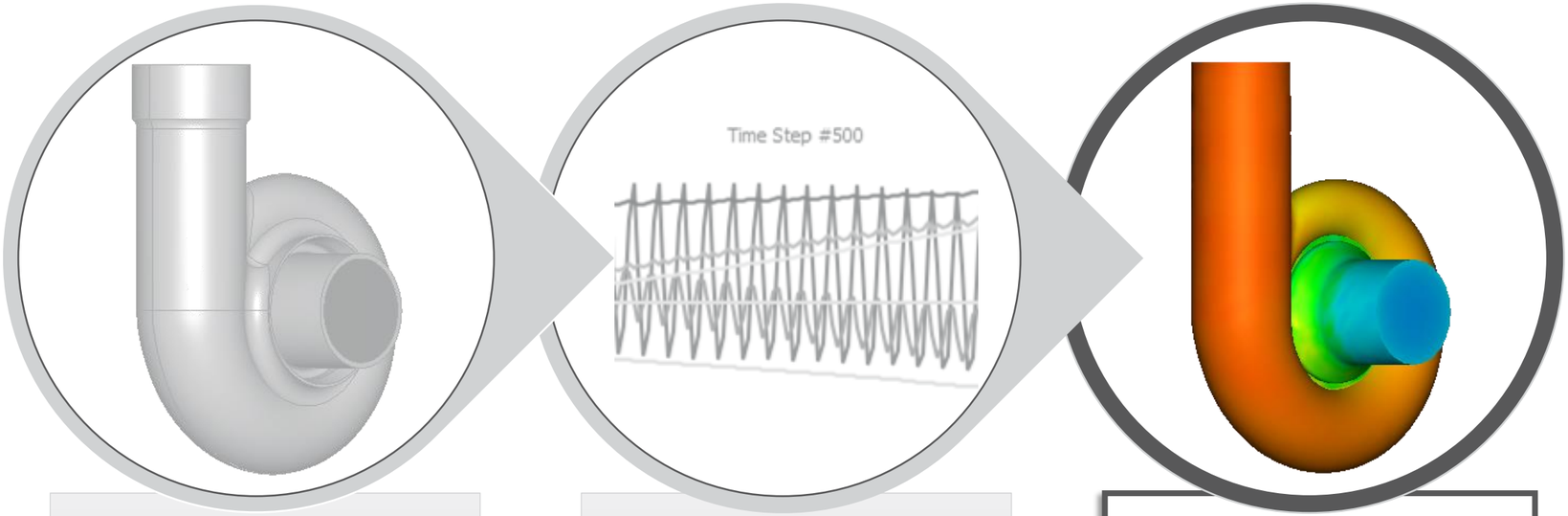
- Turn On (=1) the flag *adv5_no_dtime*

  Stability in transient analyses with a flowrate specified at the exit of the model

Instability at the inlet



Centrifugal pump: Results



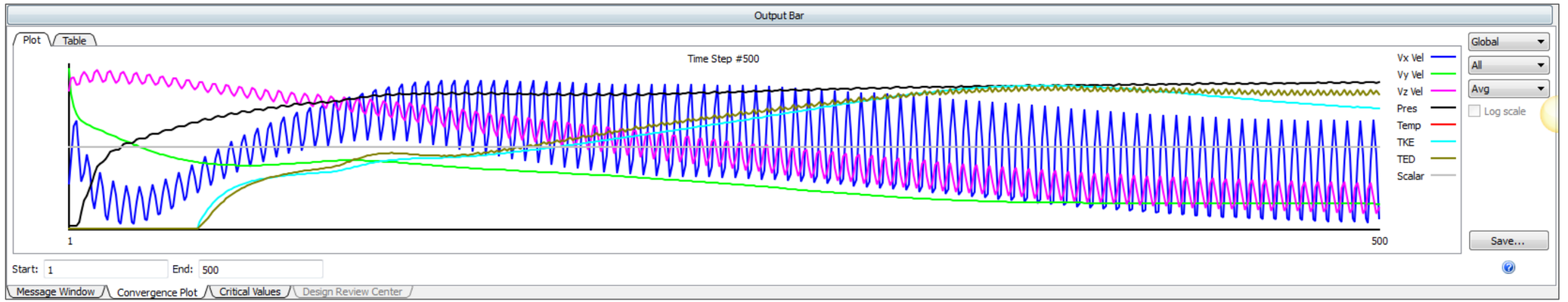
CAD
Materials
BC
Meshing

Solving

Results
Visualization

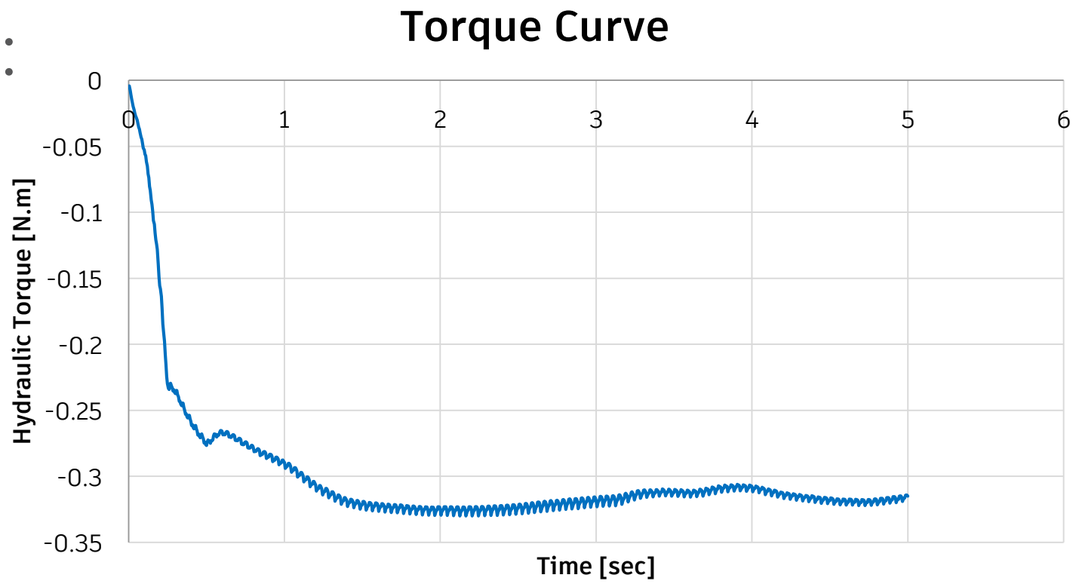
Centrifugal pump: Convergence assessment

- The convergence plot from a good Rotating region analysis will look as follows (Global results)



Centrifugal pump: Convergence assessment

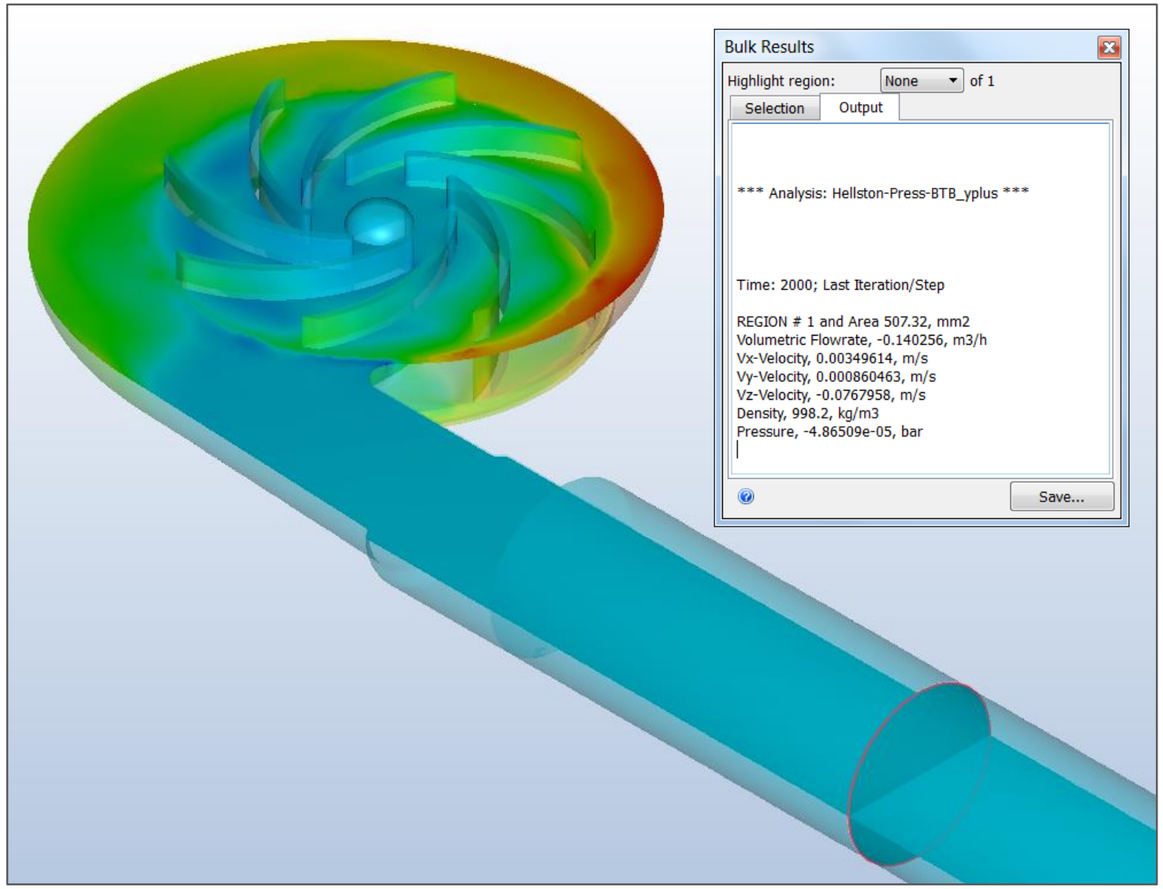
- Convergence can be assessed using:
  - The global results of pressure
  - Rotating Region results: Plot of the hydraulic torque
  - Monitor Points to track convergence of variables at specific points



Centrifugal pump: Results visualization

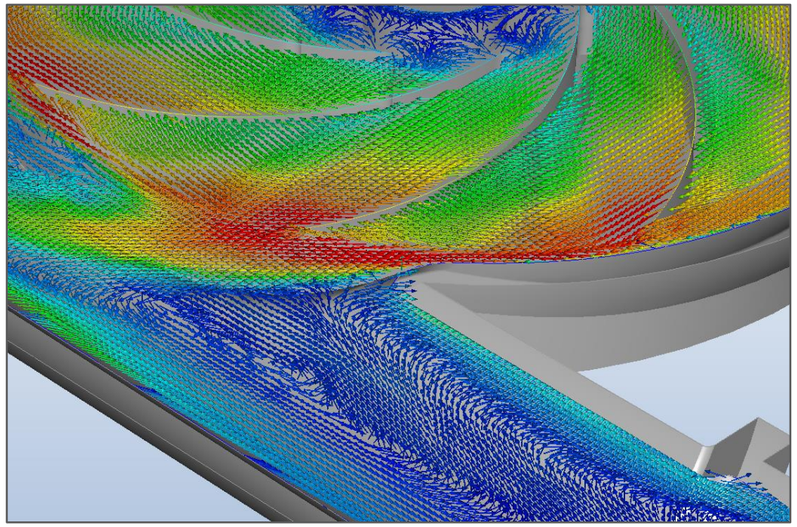
- The bulk-calculator tool calculates values of variables of interest over cut-planes

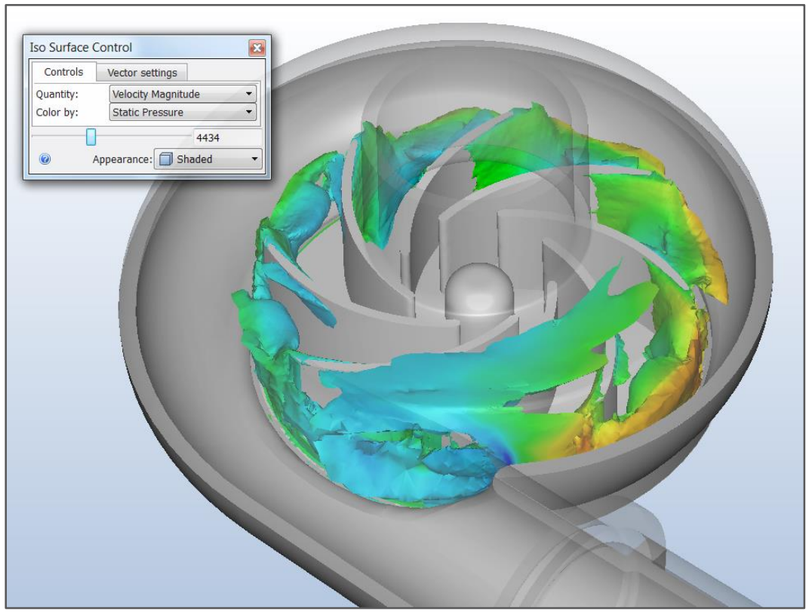
- The data extracted from the bulk calculator over the openings of the model + Hydraulic torque data help evaluate the **pump efficiency**



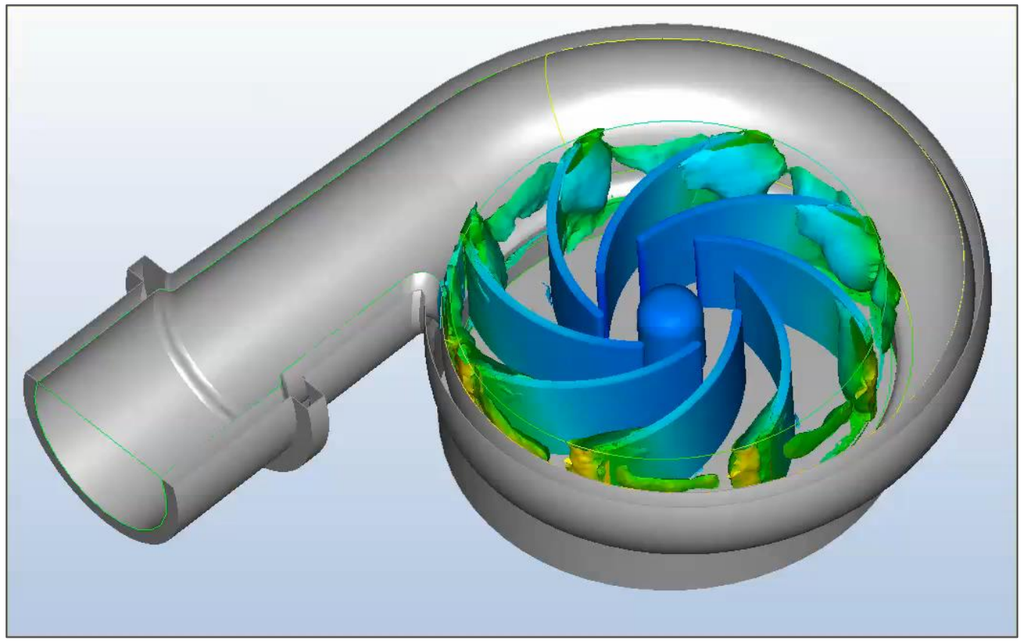
Centrifugal pump: Result visualization

- **Cut Planes** show high levels of detail – vectors are especially useful
- **ISO Surfaces** are useful for finding regions of interest
  - Highest flow
  - Cavitation
  - Nodal Aspect Ratio

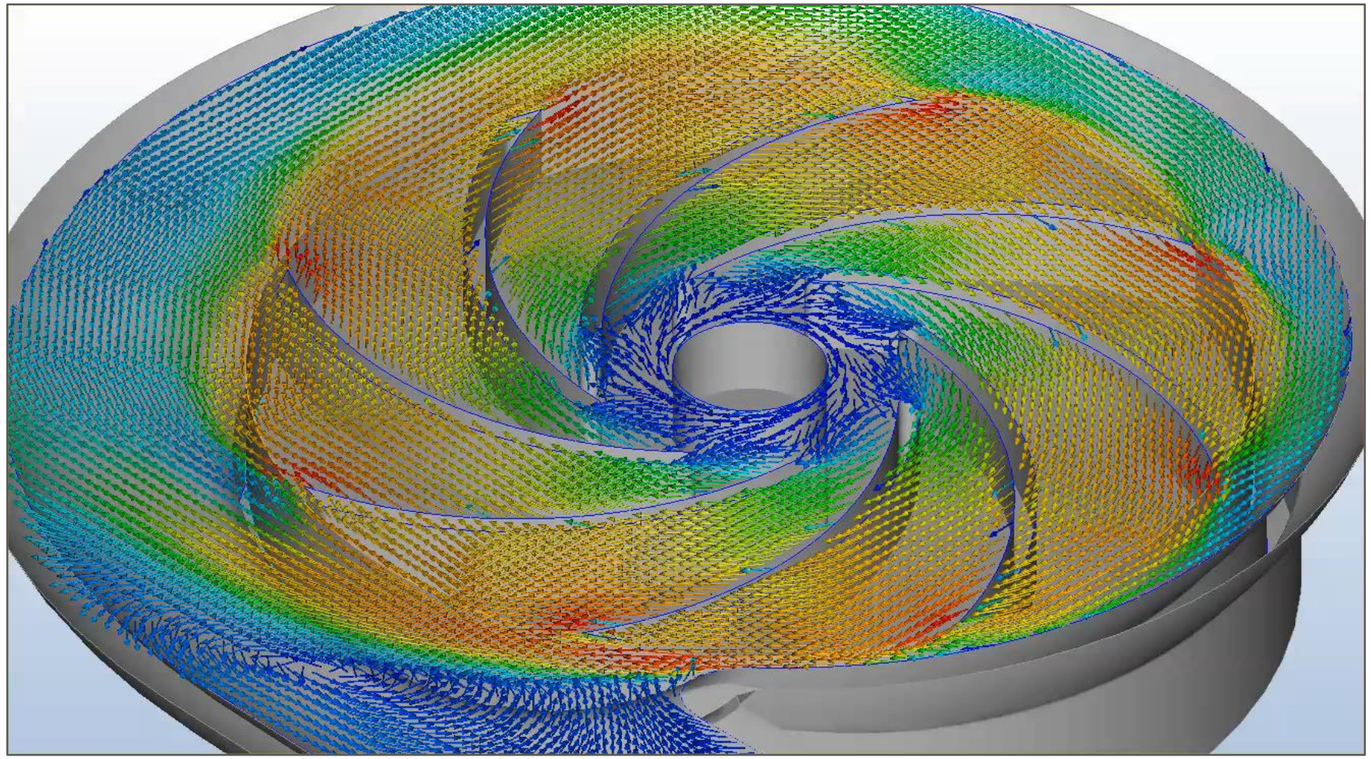




Centrifugal pump: Result visualization



Iso Surfaces



Cut-planes with vectors

Troubleshooting

- **Accuracy**
  - Verify that the geometry represents the actual geometry
  - Extend the openings when needed (avoid re-circulation)
  - Verify that the analysis settings (pressure, RPM, fluid) match test conditions
  - Refine the mesh throughout the model, and reduce time step size
  - Verify that torque, pressure, and flow have reached a steady-state solution (stopped changing). If not, run additional time steps

Questions & Answers

AUTODESK®

Make anything.