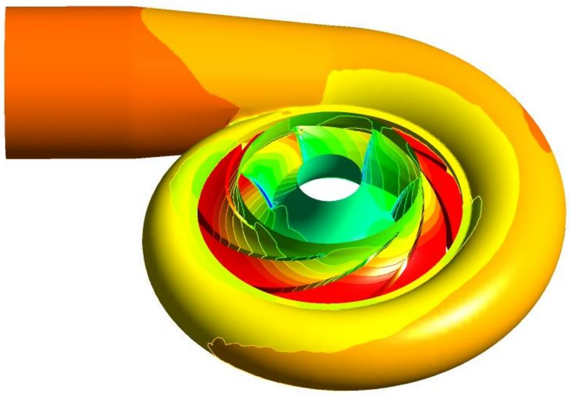
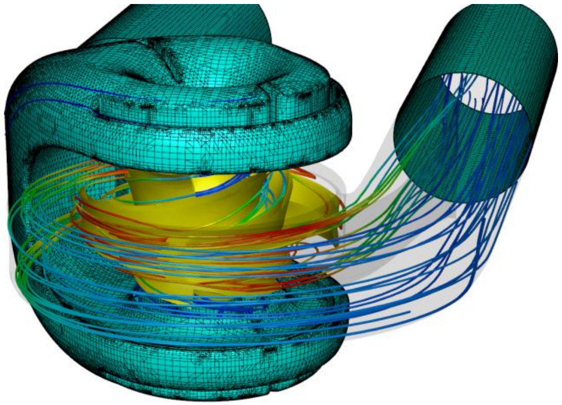
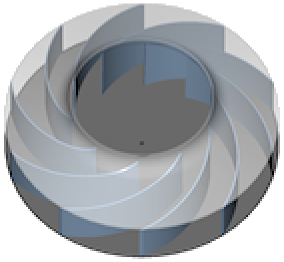
NAFEMS UK 2014 CONFERENCE

Oxford, UK
10-11 June





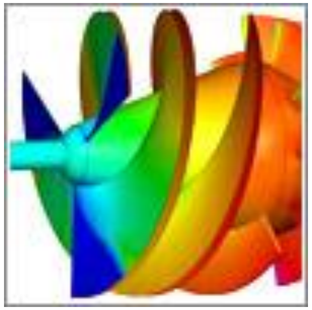
# Pushing the Boundaries of ‘UpFront CFD’



Sean Horgan and Mike Clapp
80/20 Engineering Ltd

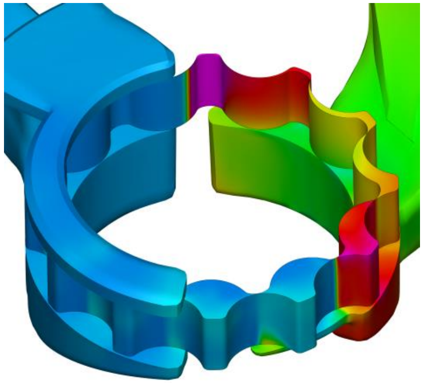
NAFEMS

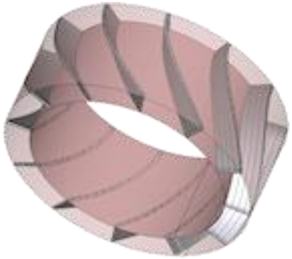
# 80/20 Engineering

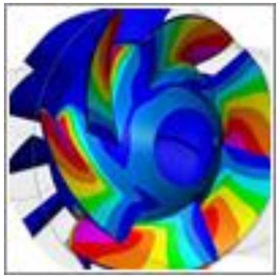


- ***AGENDA – 'NAFEMS UK - 10th June 2014'***

- Introductions
- Brief Company Overview
- Examples of Pushing the Boundaries of 'UpFront CFD'
  - Heat Exchanger and Gas Control Valve
- Turbo-Machinery Design and Verification
- Demonstration of CFturbo/PumpLinx
- Conclusions





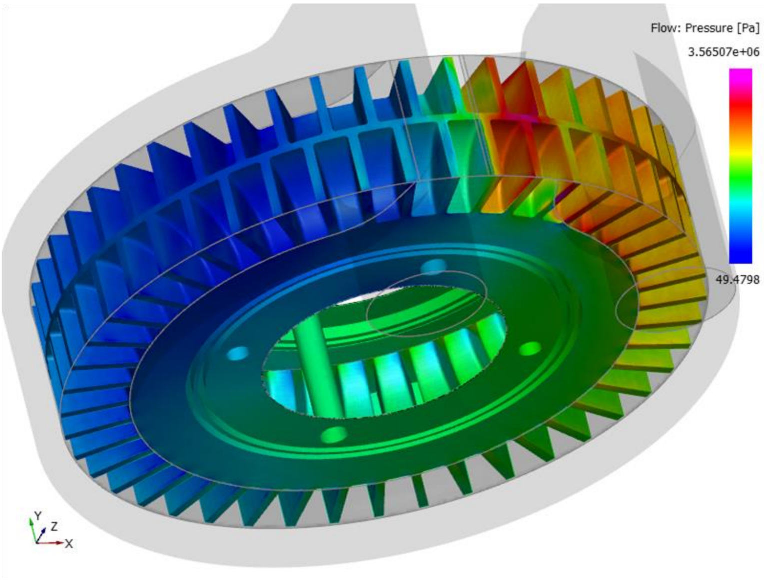


# 80/20 Engineering

- ***80/20 MISSION:***

  Deliver business value to companies in early stages of product/project development through the application of ‘best in class’ simulation technology in the field of:

  - Flow and Thermal,
  - Structural and FSI,
  - Dynamics,
  - Mechanisms,
  - Crash / Impact,
  - Multi Physics



# 80/20 Engineering

- ***The Value 80/20 Engineering delivers:***

  Design Engineers need to evaluate many concepts quickly and easily, so need a real alternative to traditional physical testing…

  - ***<u>CFD/CAE tools for Product Development</u>***

  Some Flow/Thermal/Structural applications are very complex in nature and often companies just don’t possess the resources and/or competencies to perform in-house simulation….

  - ***<u>High Level Simulation Consultancy</u>***

NAFEMS

NAFEMS UK 2014 CONFERENCE

Oxford, UK

10-11 June

# Flow and Thermal Simulation of an Electric Heat Exchanger

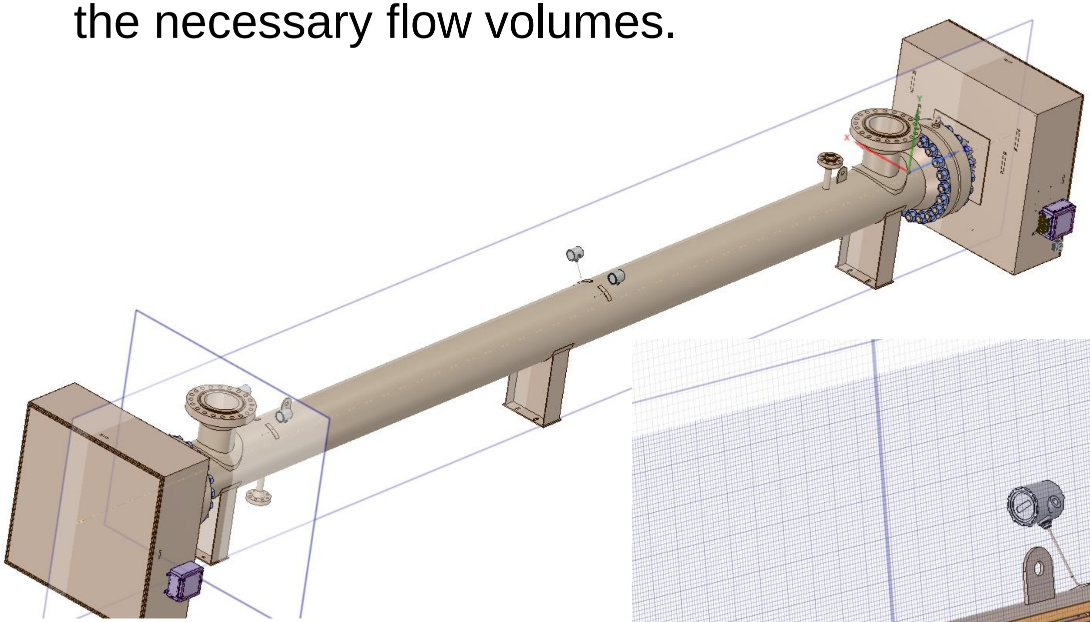
Michael Clapp

# Introduction

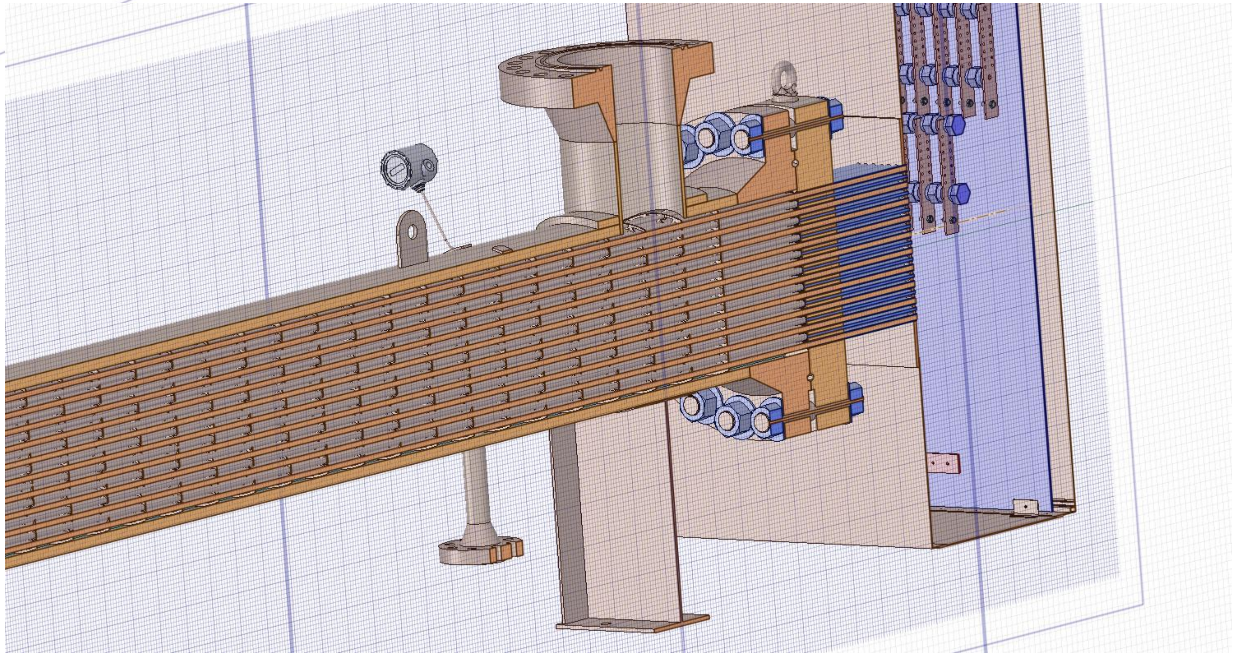
- Vulcanic UK Limited commissioned 80/20 Engineering Limited to provide a flow and heat transfer simulation of a segment of their Type 2006 Electric Heat Exchanger.
- The aim of this simulation work was to assess the Fluid Flow and Thermal performance of a new design of Electric Heat Exchanger subject to a combined set of loading conditions.
- For the purposes of this initial simulation the model is scaled down to the last 8 baffles closest to the outlet. This enables an understanding of any potential hotspots within this region based on the assumed inlet flow operating temperatures.

# Geometry

The geometry was supplied by Vulcanic as a SolidWorks Assembly. This was initially read into SpaceClaim in order to simplify the geometry and prepare the necessary flow volumes.



Region of Interest

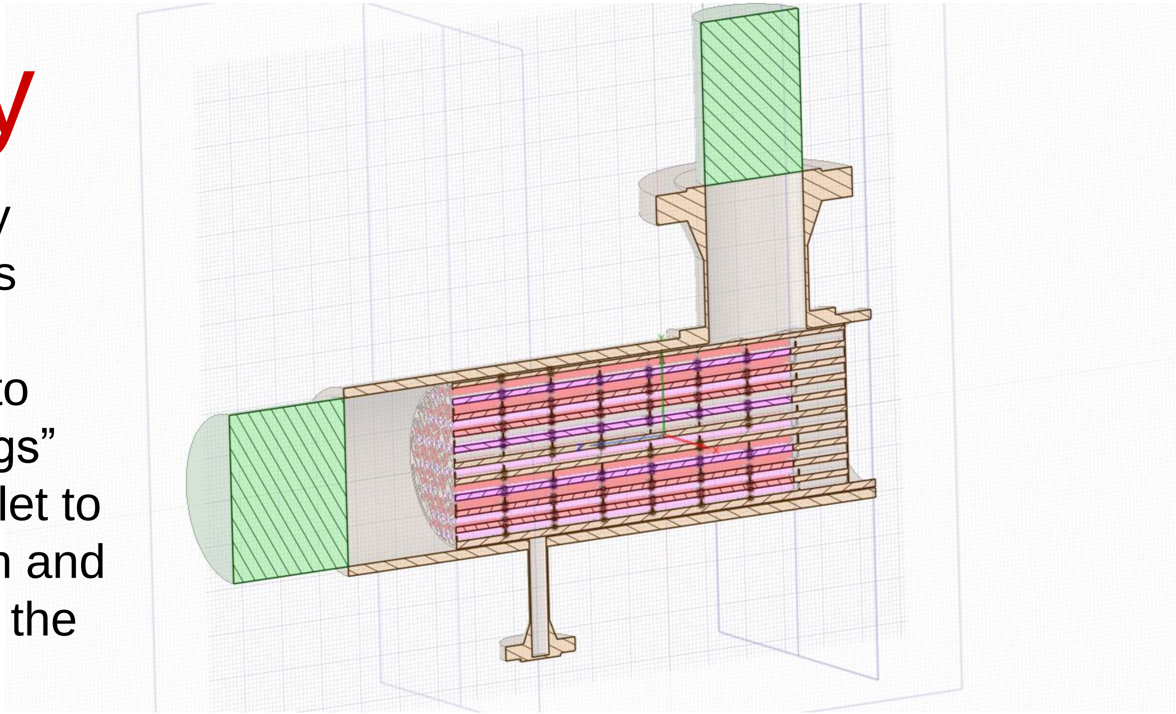


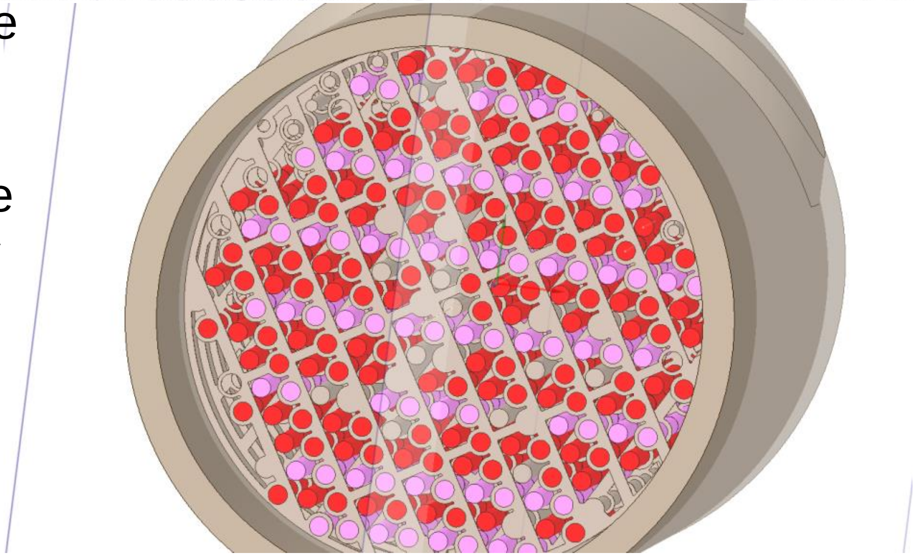
# Geometry

The geometry was simplified by removing the small components around the outside of the heat exchanger and cutting it down to just the region of interest. “Plugs” were added to the inlet and outlet to close off the internal flow region and add surfaces on which to apply the boundary conditions.

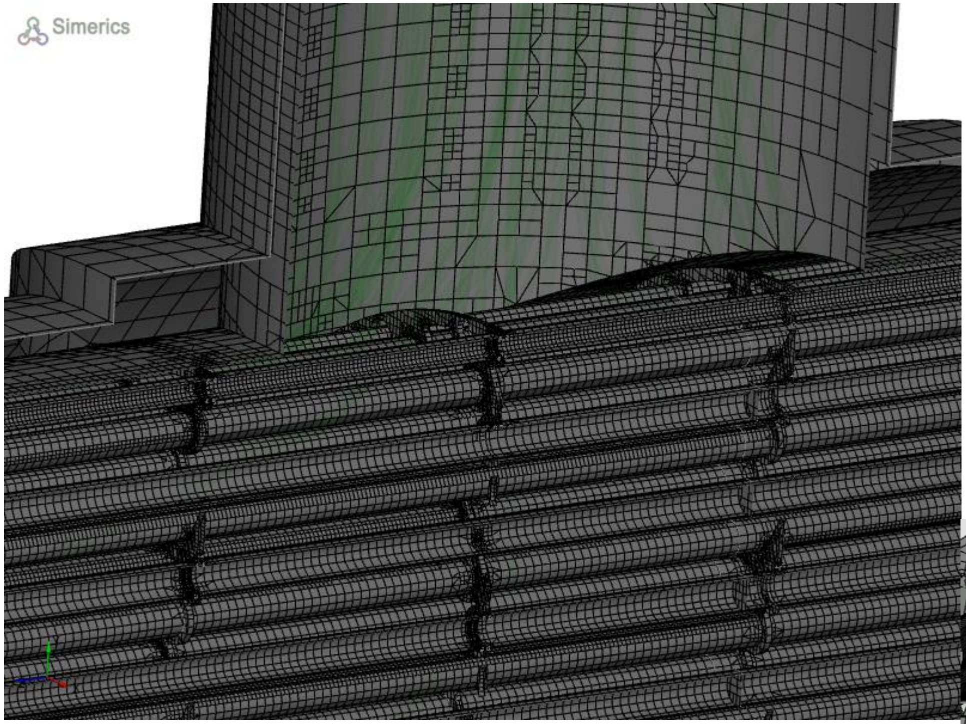
Many of the small clearances were closed to reduce the size of the mesh.

The rods were grouped so that the heating elements could be quickly identified in the Simerics CFD software.

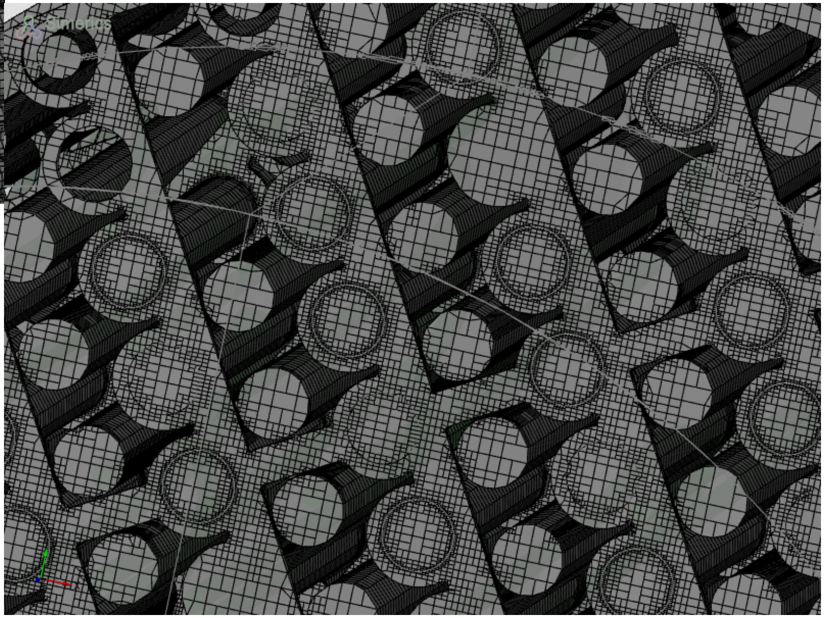




# Mesh



The completed mesh contained 10,677,151 cells with 36,243,104 faces.



# Fluid Properties

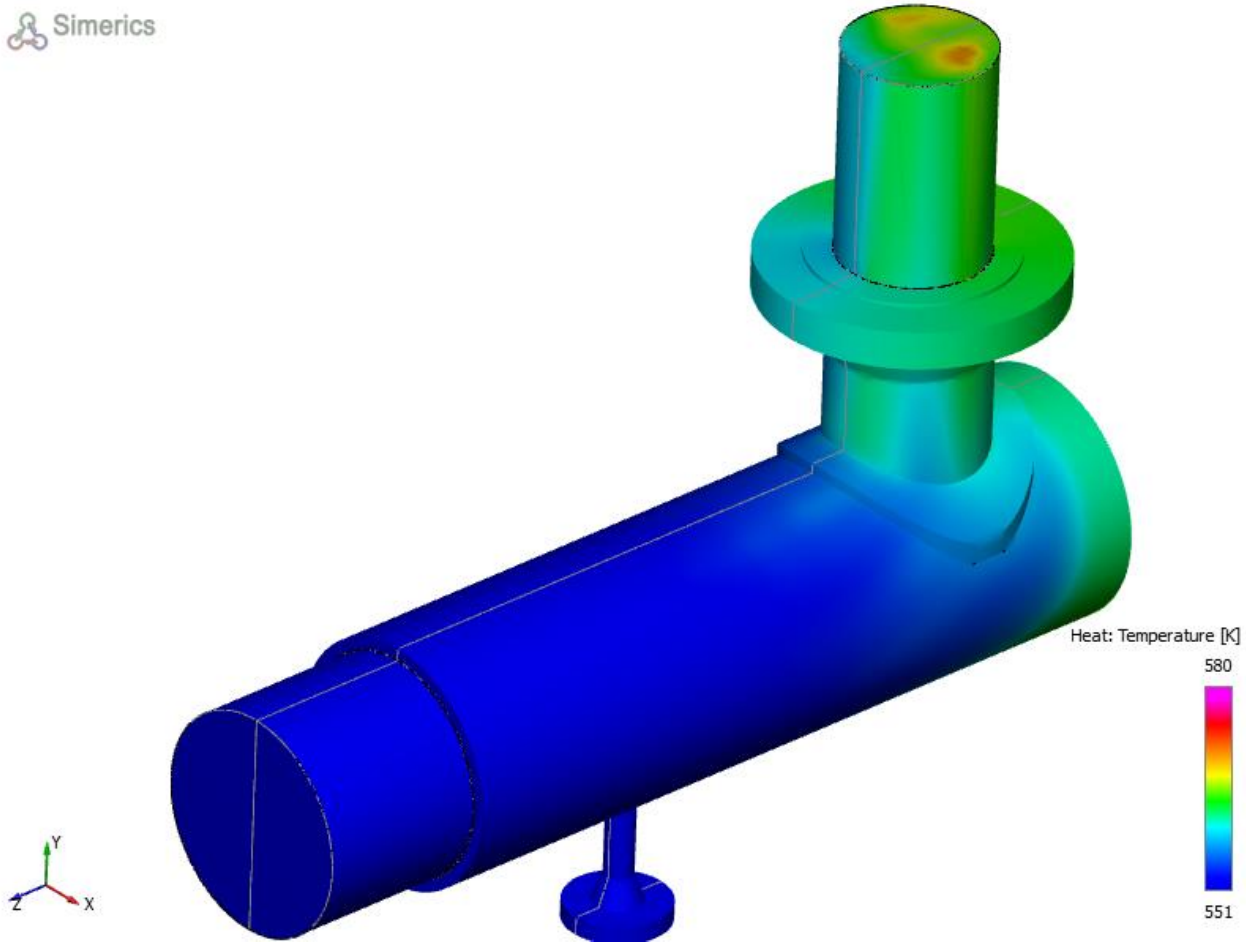
Temperature dependent fluid properties were applied to the three fluid regions using the data supplied by Vulcanic from their TASC calculation. This used the following tabular data.

| Temperature (Kelvin) | Density ($kg/m^3$) | Viscosity (pa.sec) | Thermal Conductivity (W/m.K) | Specific Heat (J/kg.K) |
|---|---|---|---|---|
| 0 | 50 | 1.9e-5 | 0.07 | 2900 |
| 273 | 40 | 1.9e-5 | 0.07 | 2900 |
| 539 | 28.838 | 1.928e-5 | 0.07471 | 2959 |
| 552 | 28.193 | 1.955e-5 | 0.07675 | 2989 |
| 568 | 27.389 | 1.999e-5 | 0.0793 | 3028 |
| 6000 | 22 | 2.0e-5 | 0.08 | 3200 |

# Solution

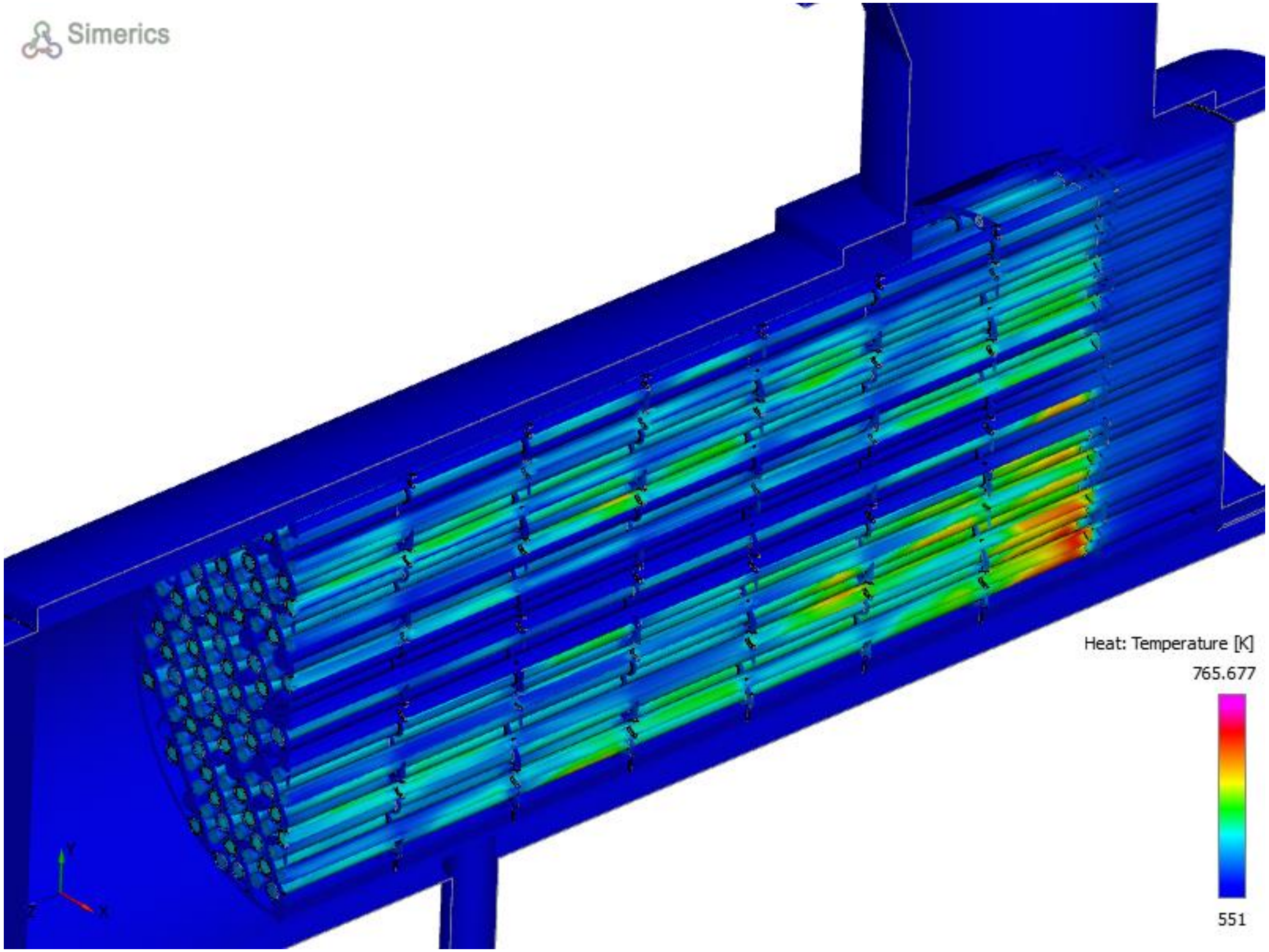
- The model was run as a steady state. All the default Simerics solution settings were used. The flow and heat modules were used with the standard K-epsilon turbulence model. The solution stopped after 1000 iterations. After that some relaxation controls were added which caused the solver to reach its default convergence criteria after a further 98 iterations.
- The full solution sequence took about 7 hours to run on a Dell Precision T1650 workstation with a 3.4 GHz Xeon processor.

# Exterior Temperatures



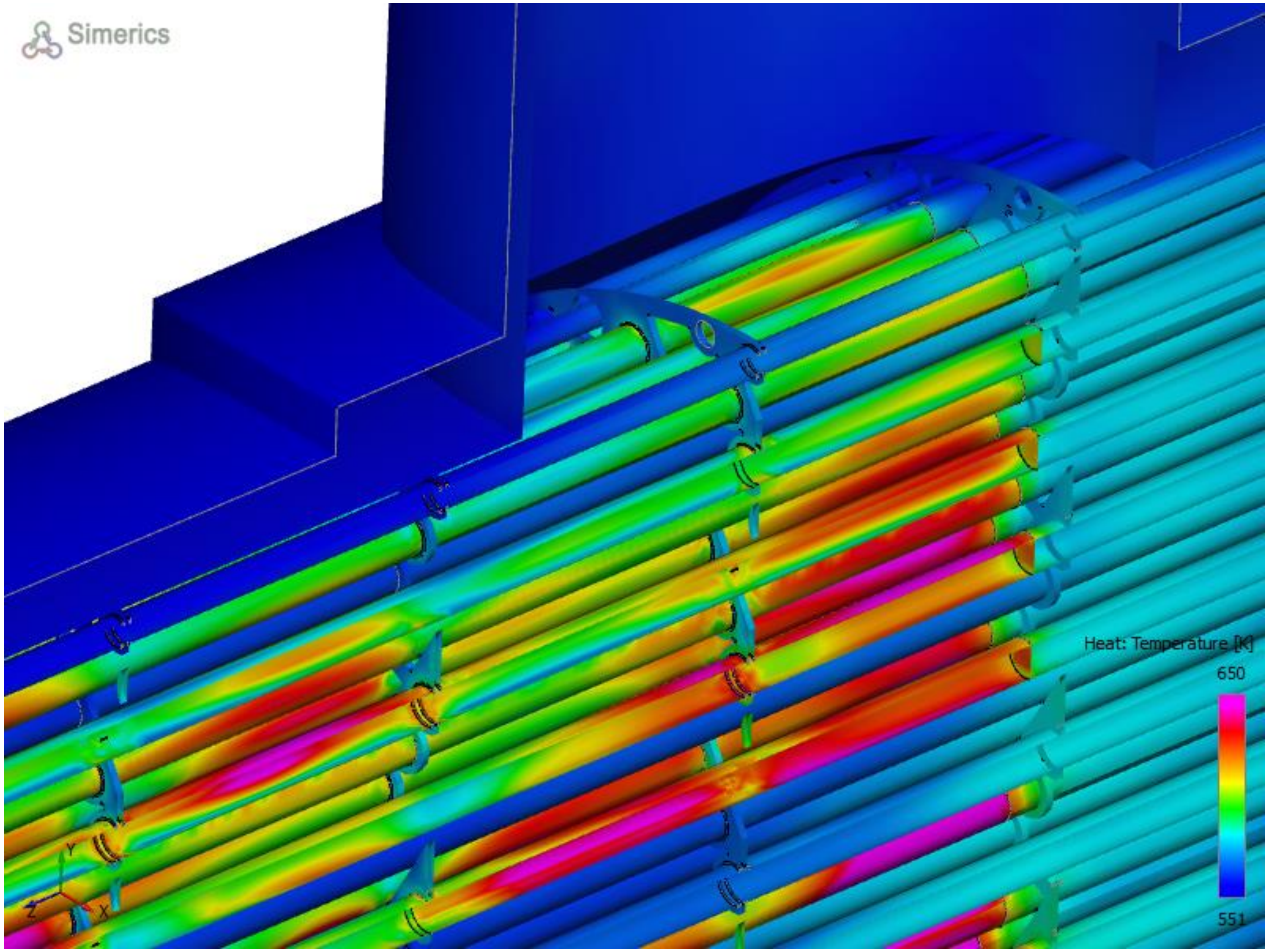

Temperature contours on exterior surface – note that the temperature scale is limited

# Internal Temperatures



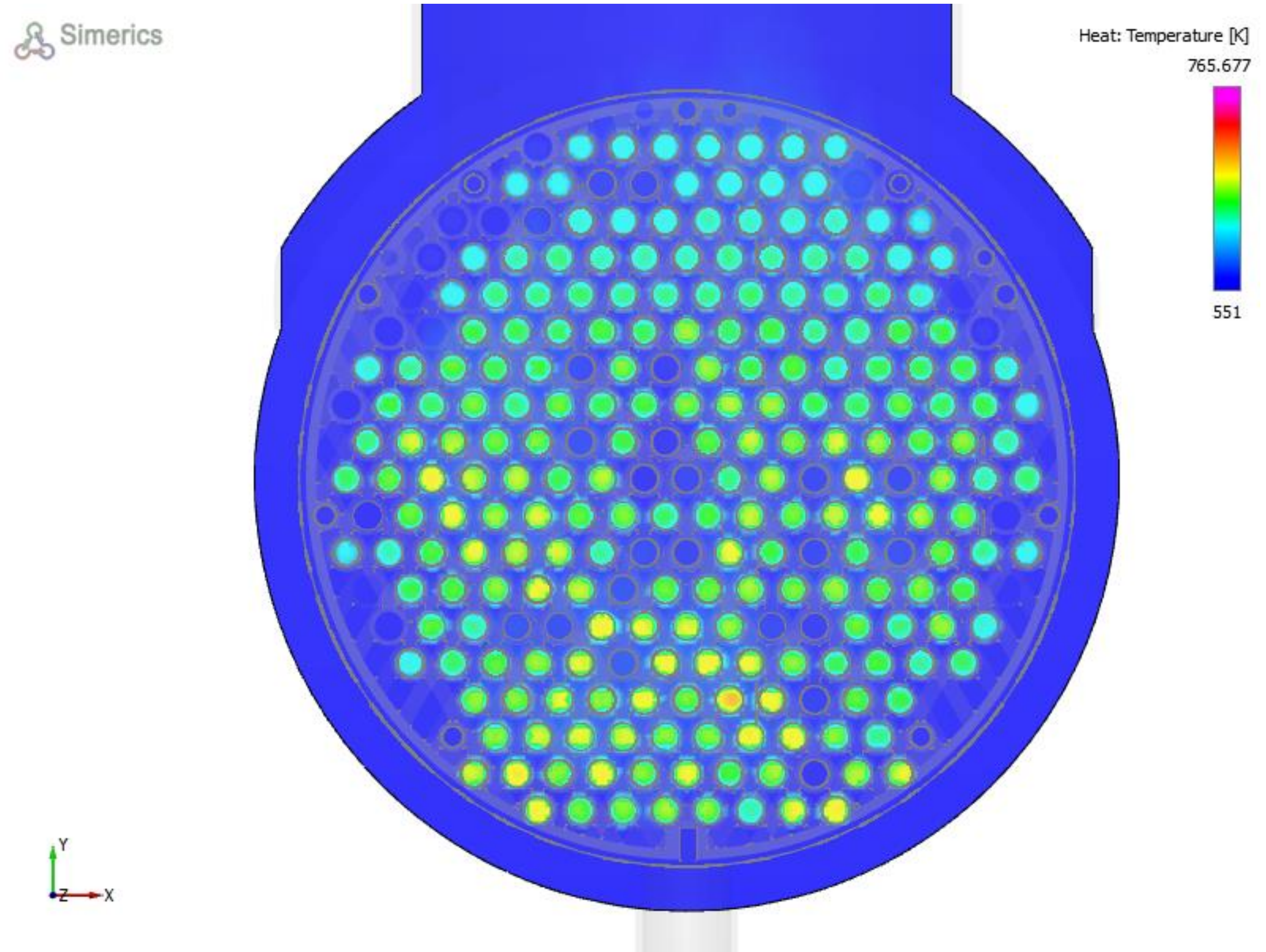

Temperature contours on clipped view of tube bundle

# Internal Temperatures



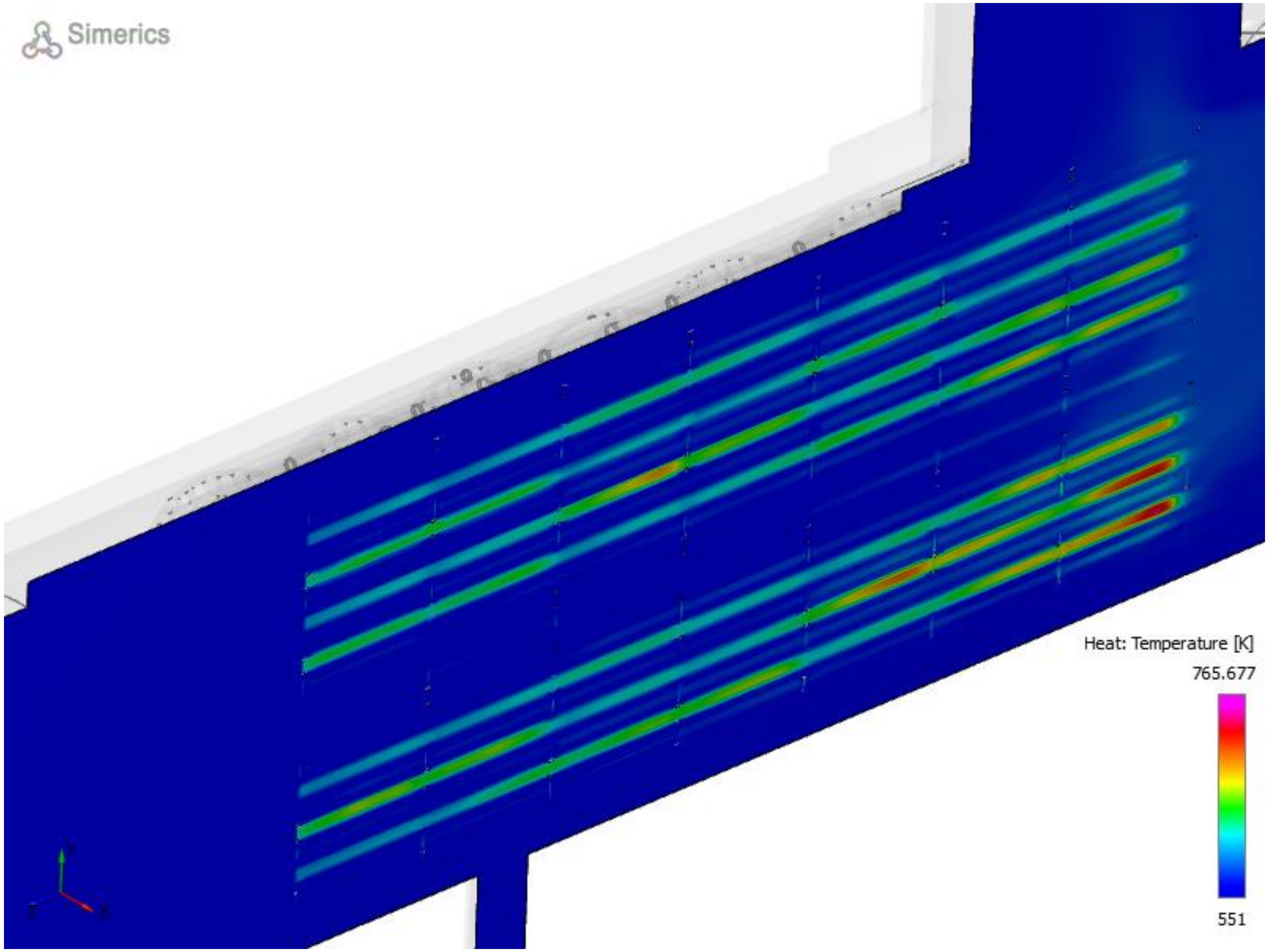

Detail of temperature contours on clipped view of tube bundle – note that the temperature scale is limited

# Cutting Plane Temperatures

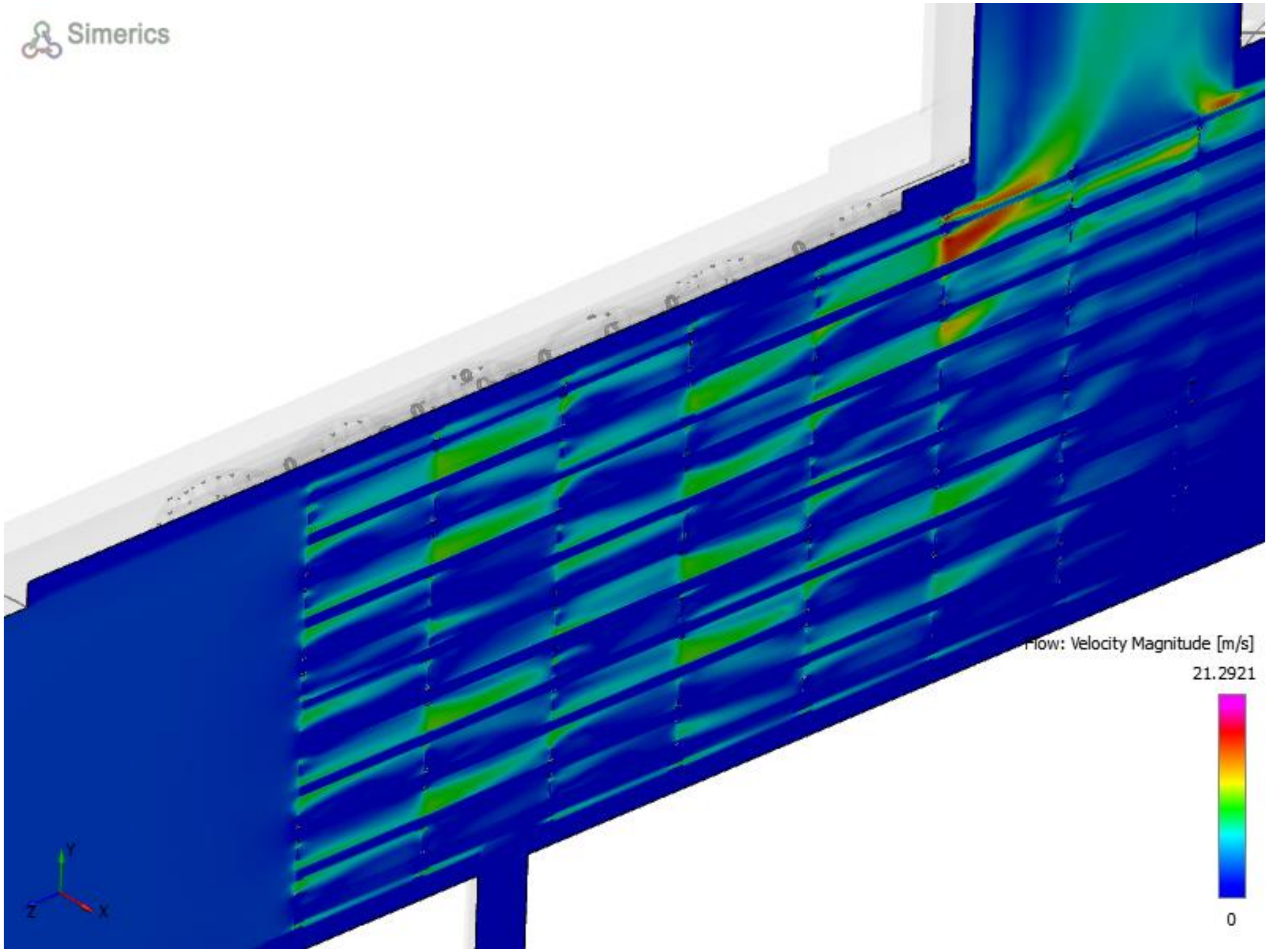



Temperature contours on cutting plane through tube bundle

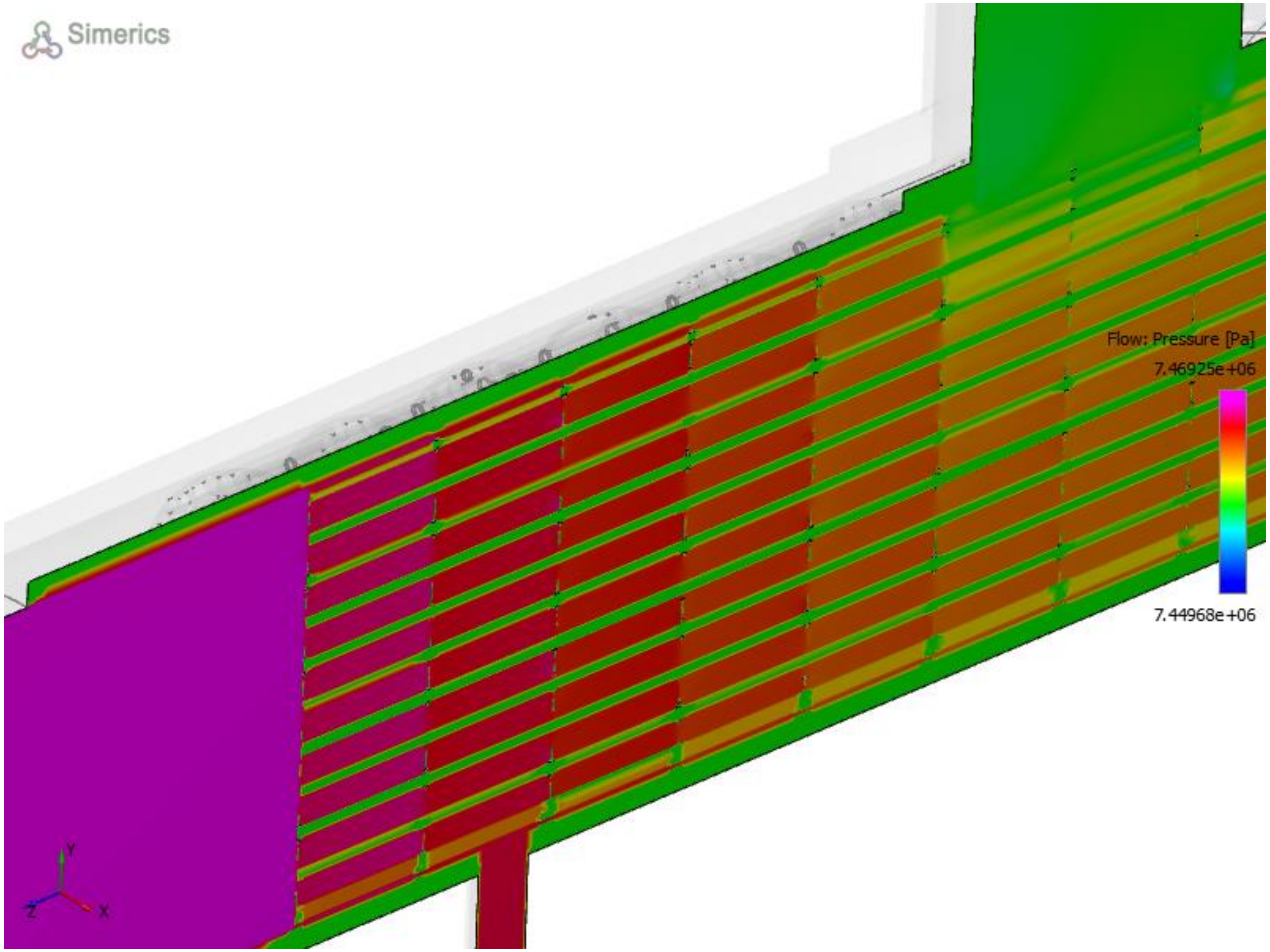
# Cutting Plane Temperatures



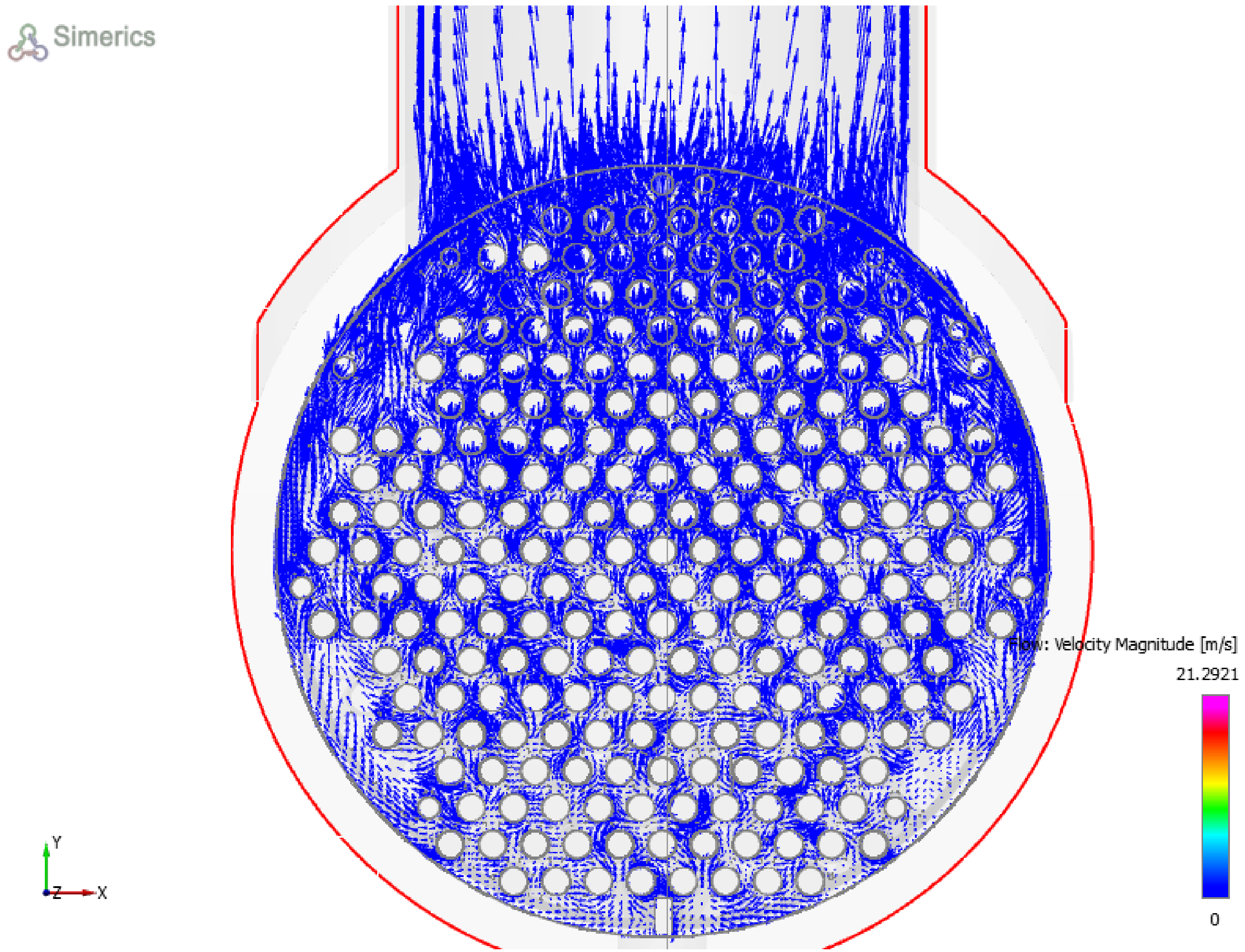
# Cutting Plane Velocities



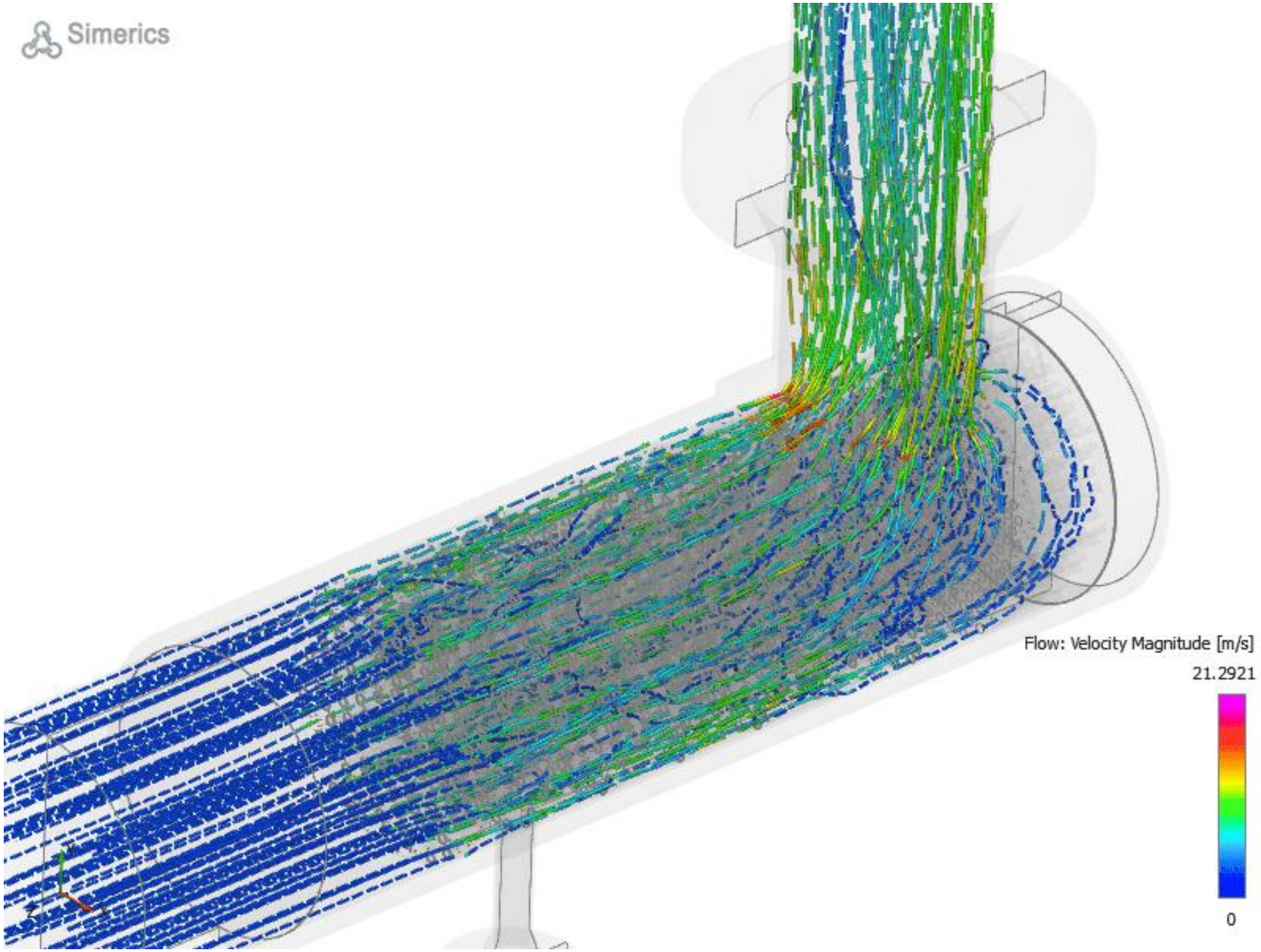
# Cutting Plane Pressures



# Velocity Vectors



# Streamline Traces




Mass less particle traces coloured by velocity magnitude.

# Flow Simulation of an Oxygen Control Valve for BPR Medical

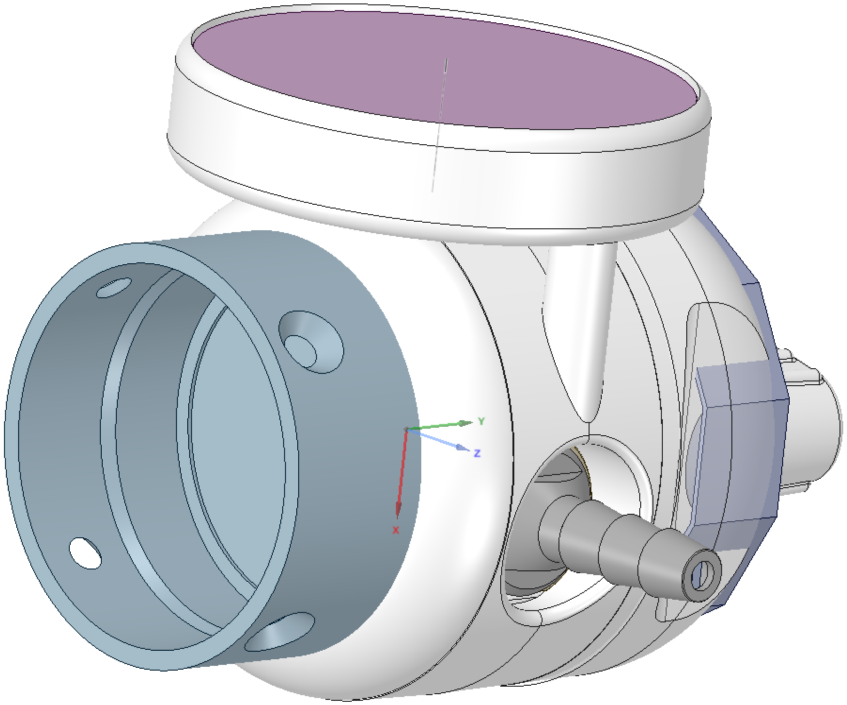
Michael Clapp

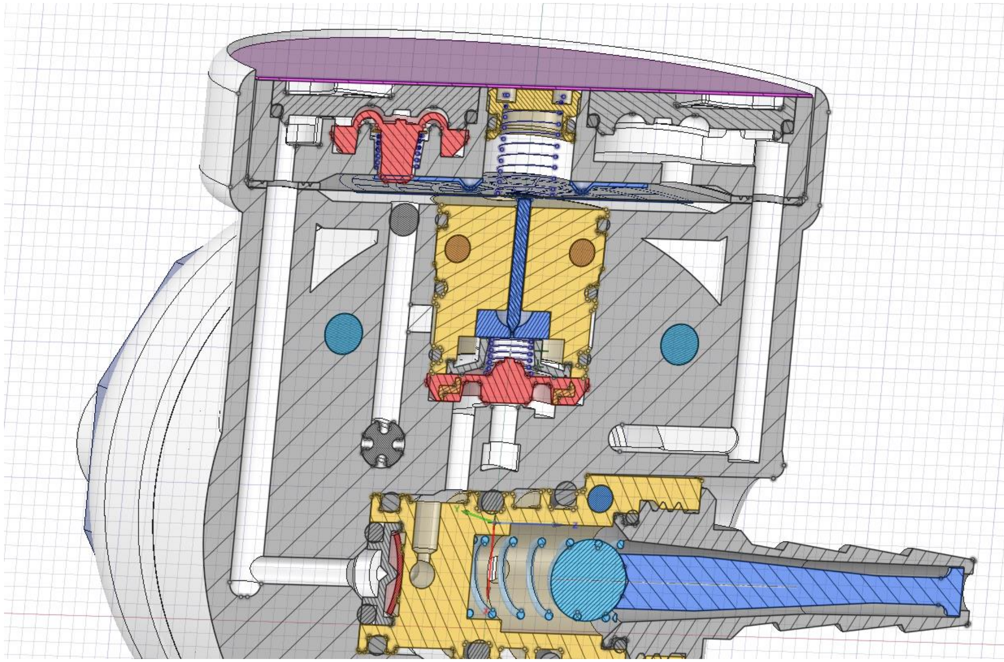
# Introduction

- BPR Medical commissioned 80/20 Engineering Limited to provide a transient flow simulation of an Oxygen Control Diaphragm Valve.
- The aim of this CFD study was to assess the performance of a Gas Control Valve in terms of flow characteristics.

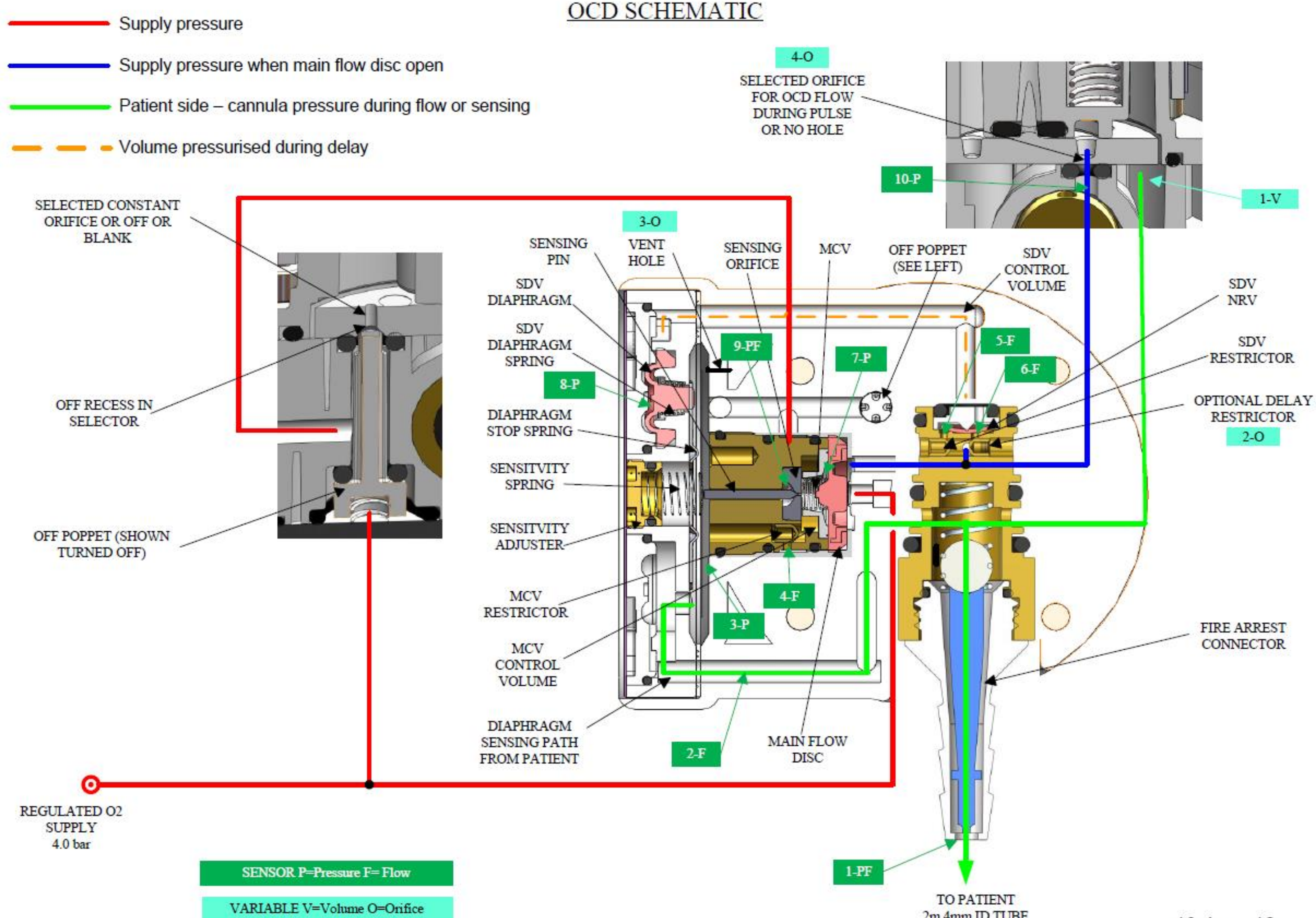
# Geometry

The geometry was supplied by BPR Medical as a SolidWorks Assembly. This was initially read into SpaceClaim in order to prepare the necessary flow volumes.



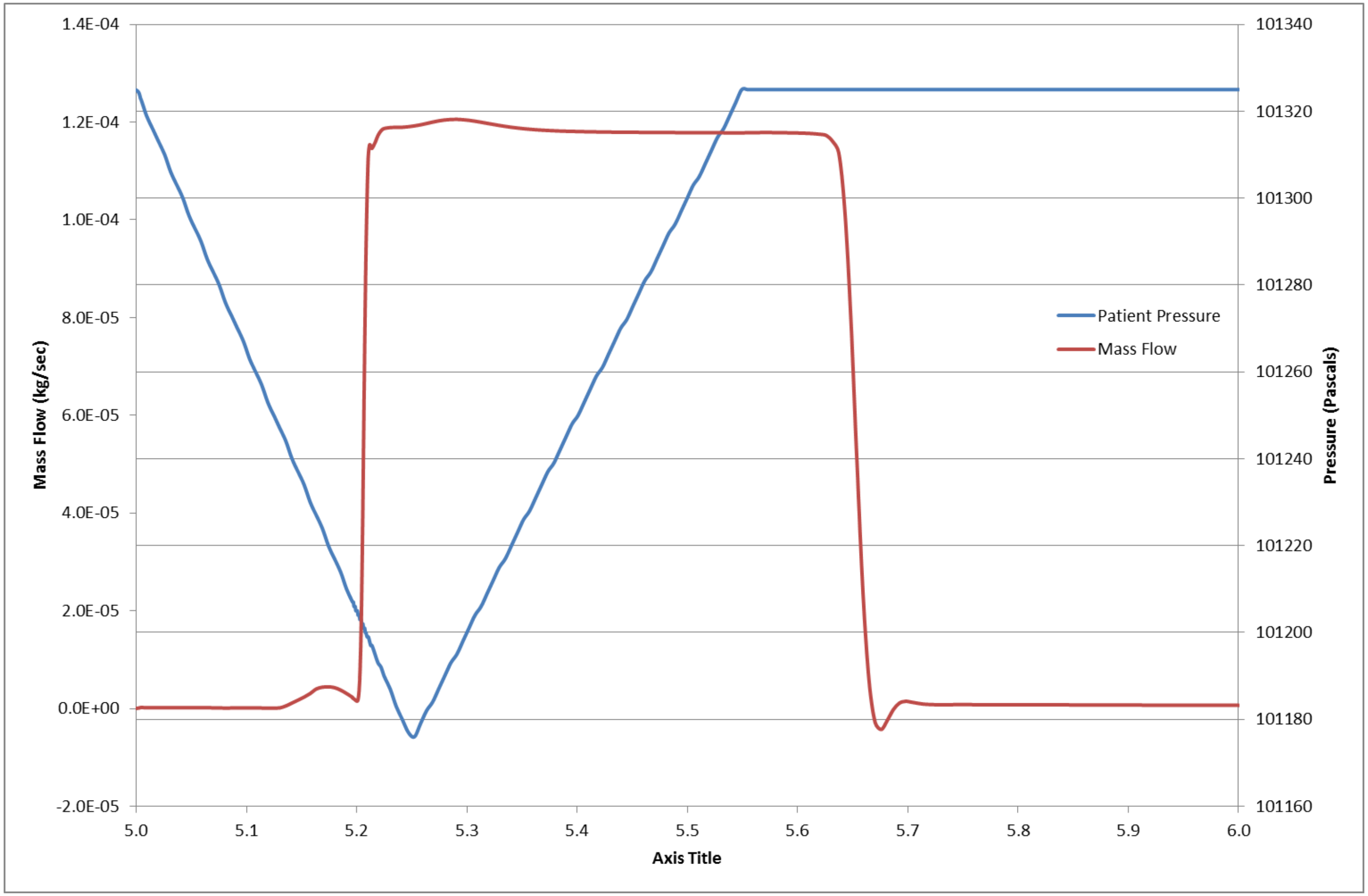


# OCD SCHEMATIC




18 June 12

# Patient Outlet

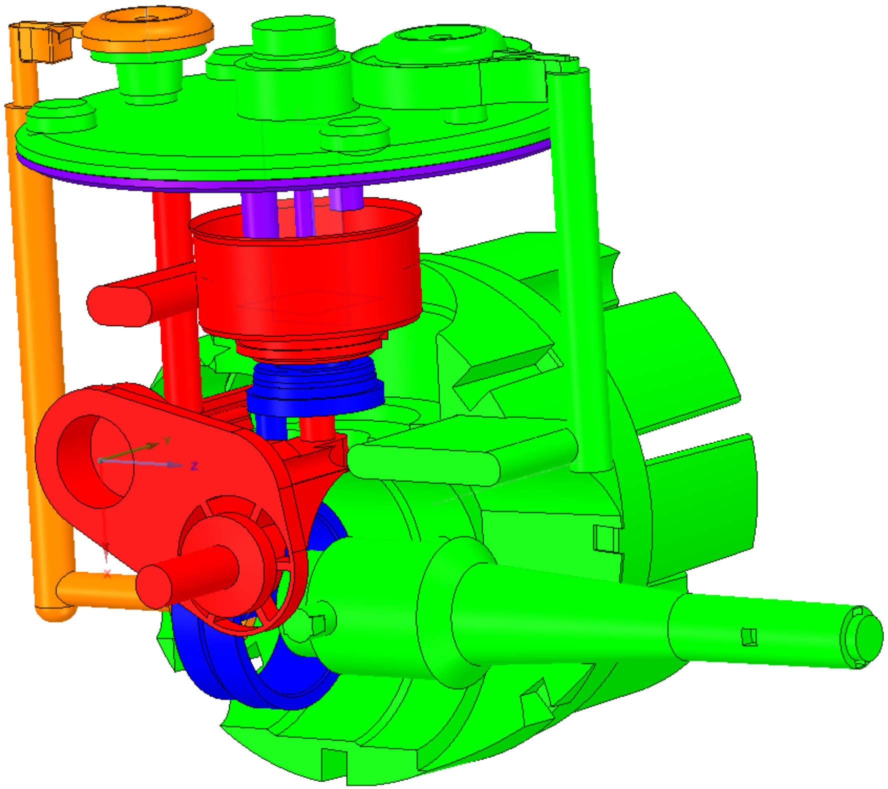



Pressure pulse from patient and Oxygen Flow to patient over first 1 second of cycle

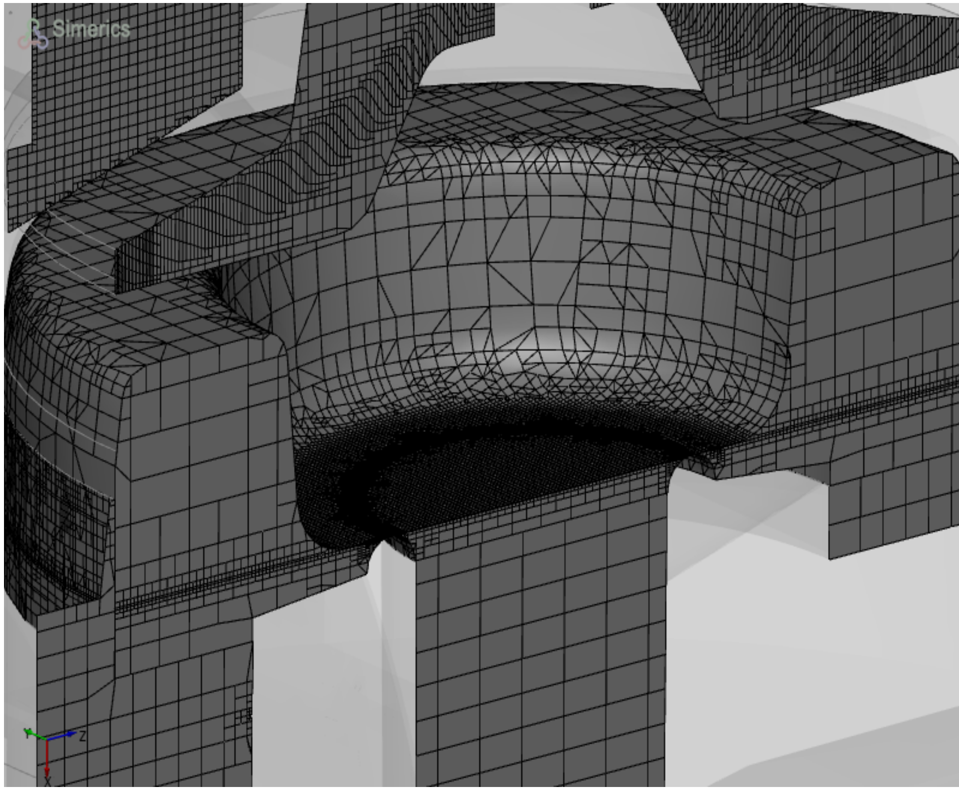
# Geometry

The flow volume was simplified by removing the details of the springs and some of the fillets. It was then split into 17 different volumes which were grouped into the different coloured regions shown below. These volumes were exported as individual STL files which were read into the Simerics Software.

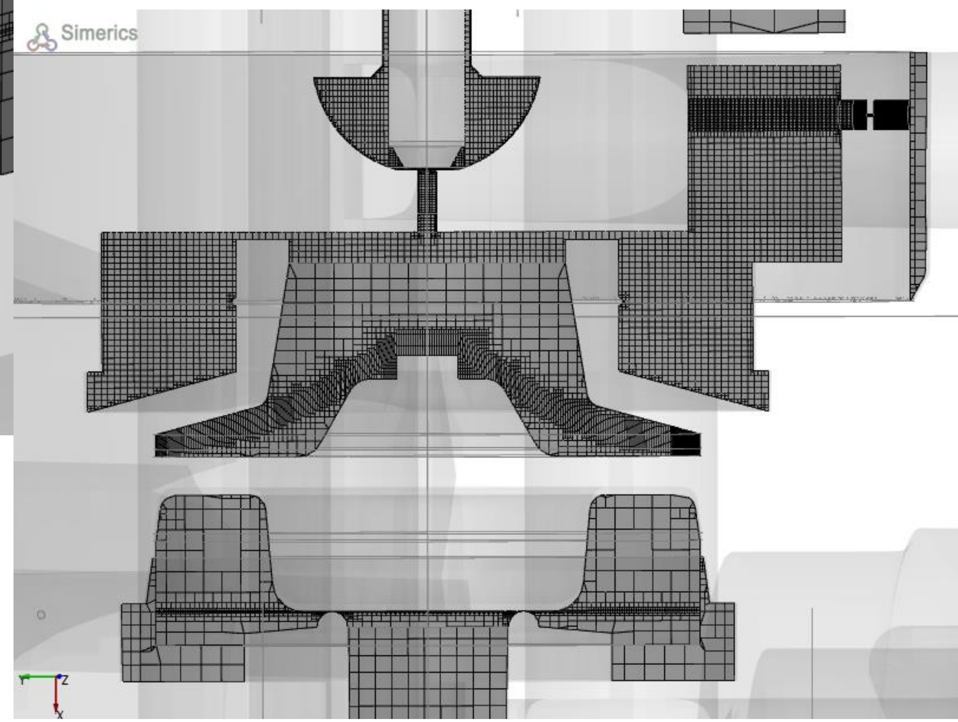
Green – Patient Side
Red – Supply Side
Orange – Delay Circuit
Blue – Supply when Main Valve is open
Purple – Needle Valve and Diaphragm Discharge.



# Mesh



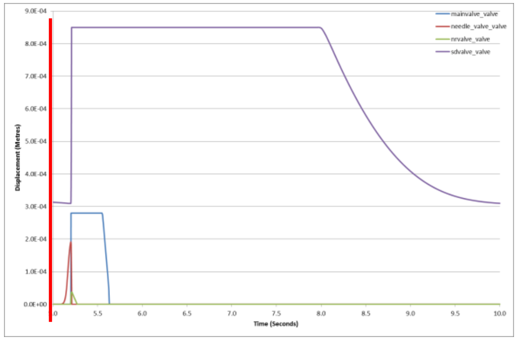

The completed mesh contained 2,616,171 cells with 9,523,125 faces.



# Solution

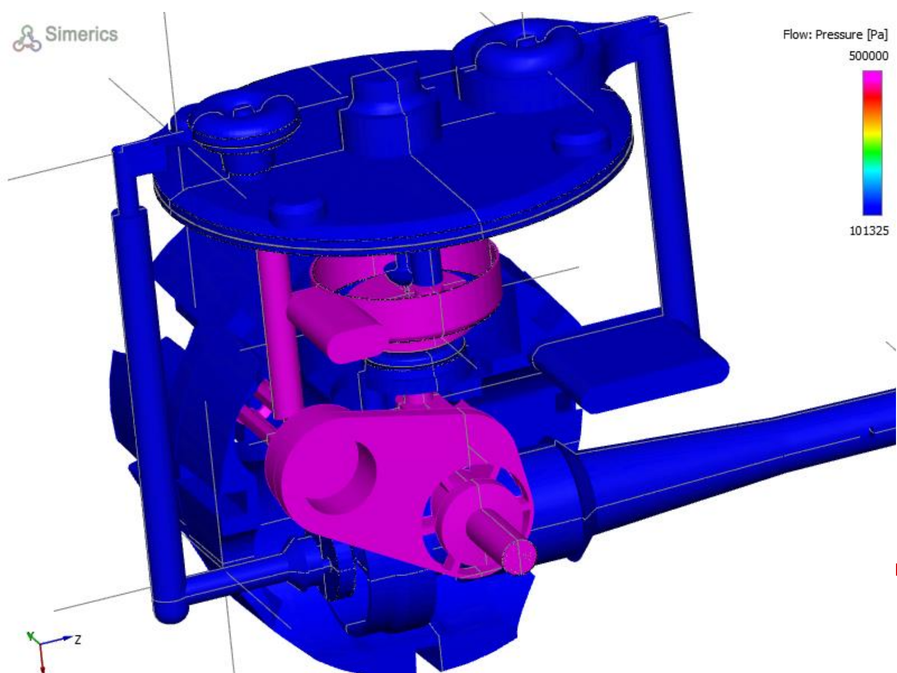
- An initial steady state solution was run to obtain a flow solution with the valves in the positions defined in the previous slide. Once this was converged a transient calculation was started to move the valves into their positions for the operating transient. This third stage was initiated by the start of the ramped pressure load on the patient outlet after 5 seconds.
- The entire solution sequence took 60 hours to run on a Dell Precision T1650 workstation with a 3.4 GHz Xeon processor.

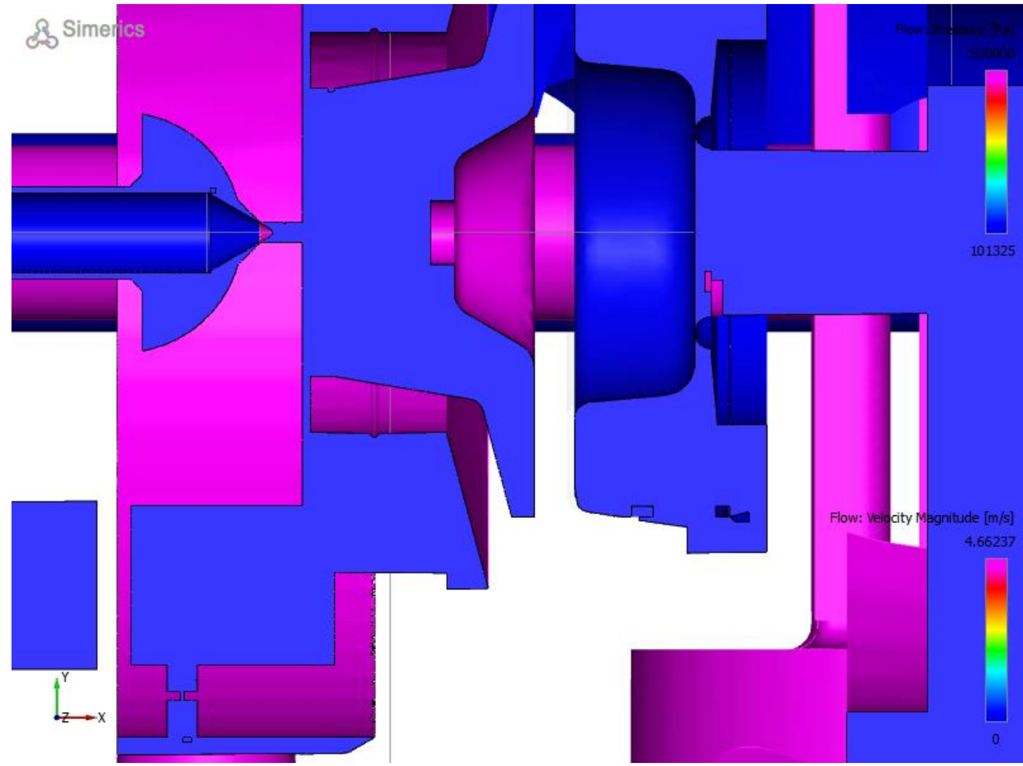
# Step 1: Initial Positions




Time = 5 Secs

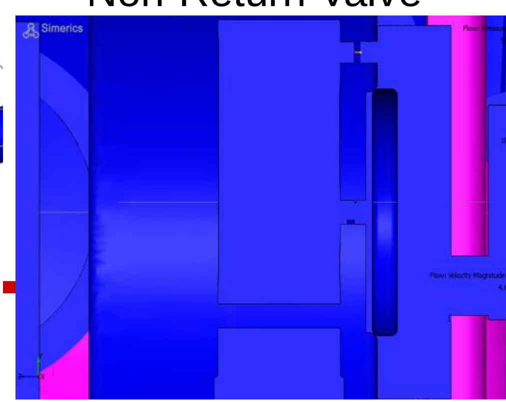
All valves initialised in closed position – high pressure only on supply side



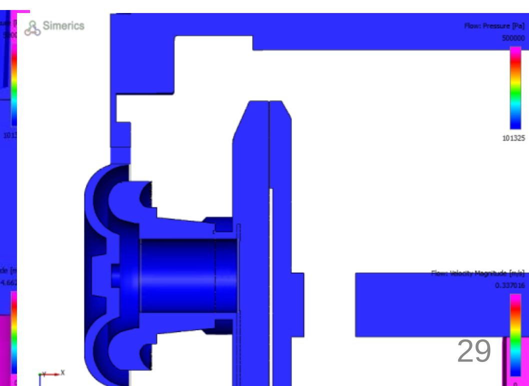




Main and Needle Valves

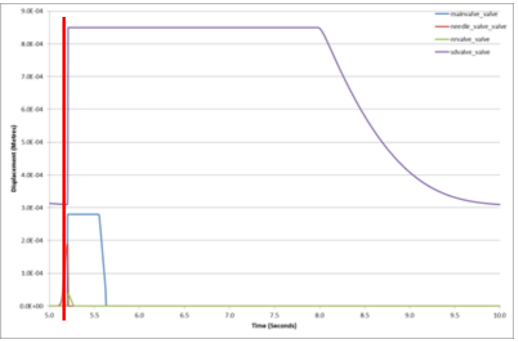
Non-Return Valve




SDV Valve

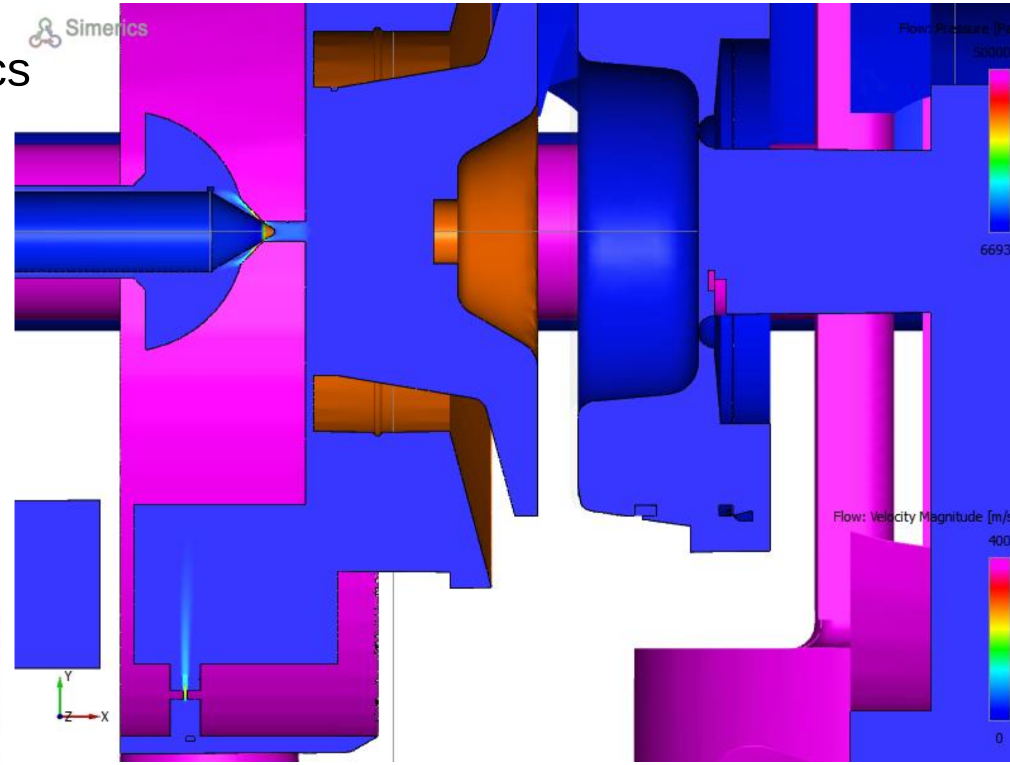


# Step 2: Needle Valve Triggered

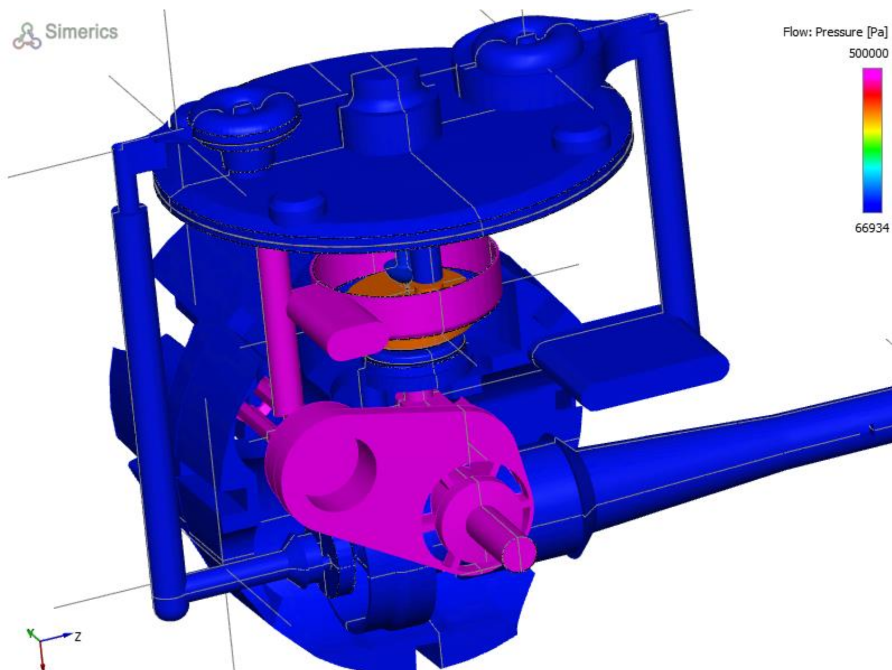


Time = 5.141 secs

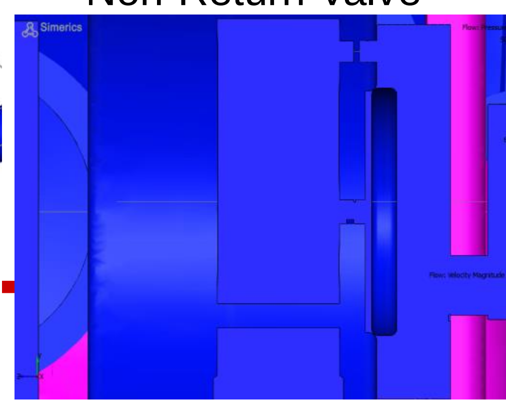
Needle Valve opens due to suction pressure pulse on Diaphragm



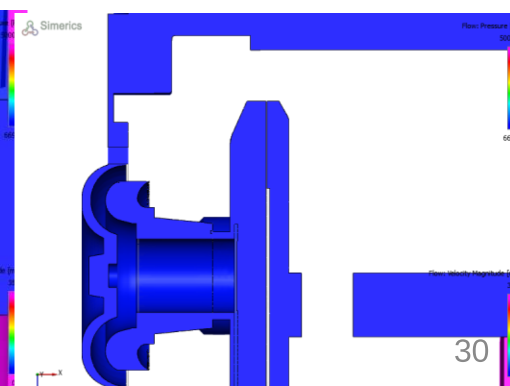

Main and Needle Valves



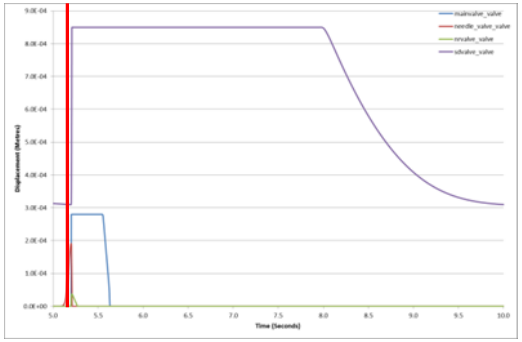

Non-Return Valve




SDV Valve

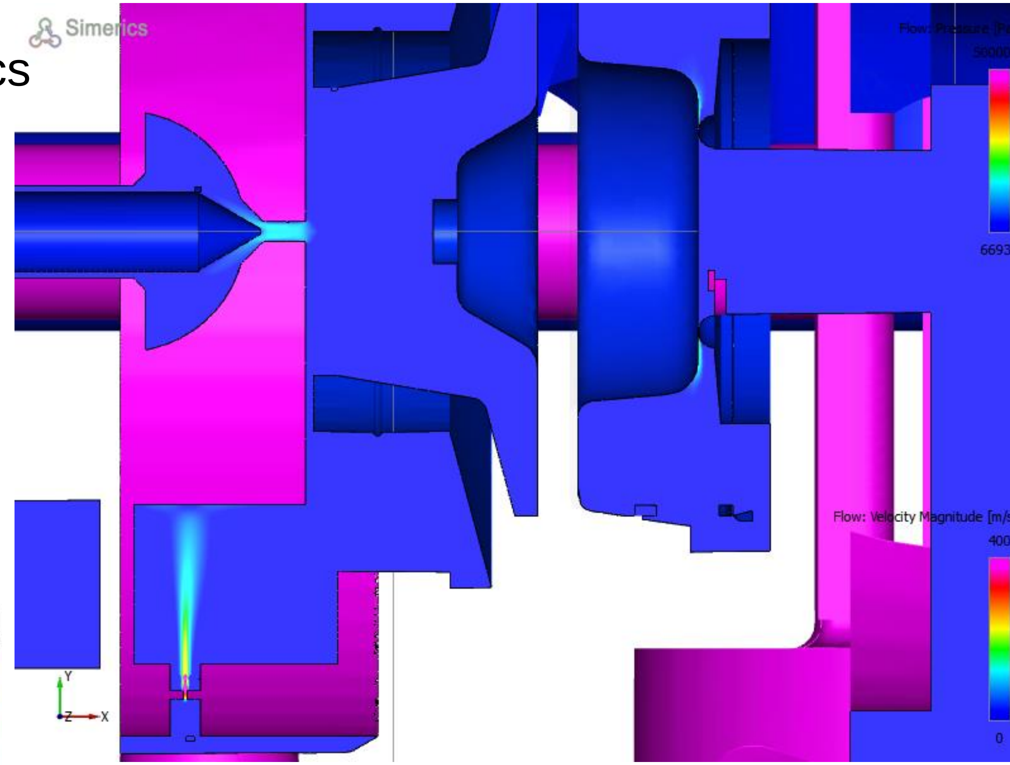


# Step 3: Main Valve Opens

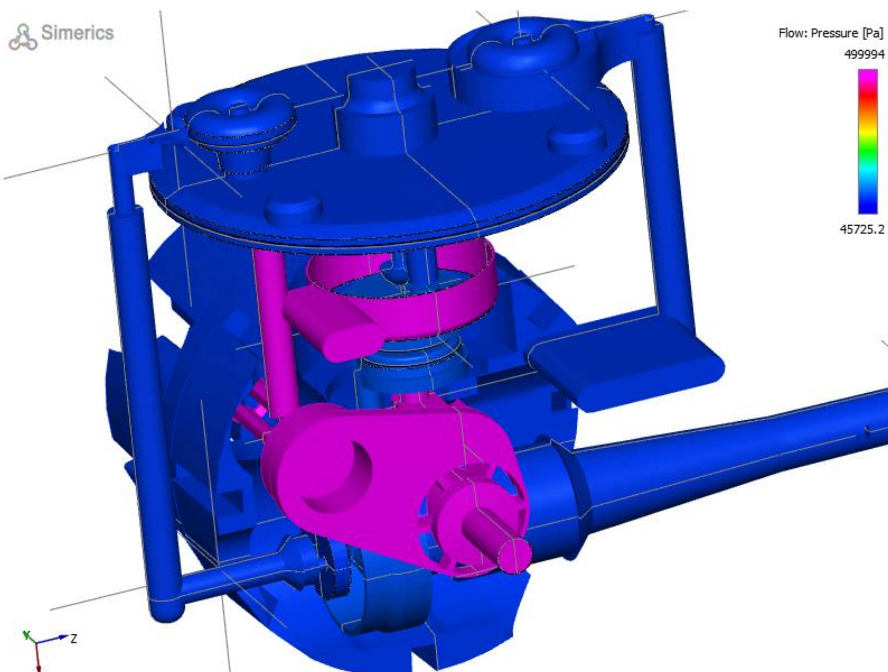


Time = 5.199 secs

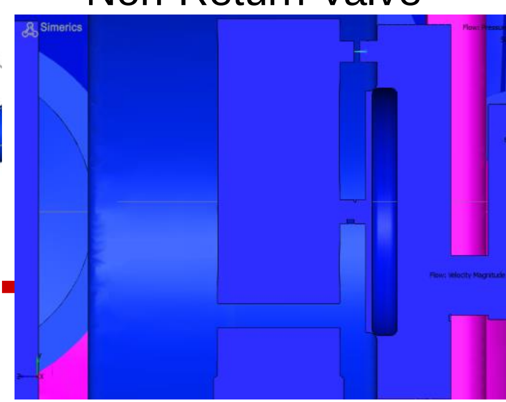
Main Valve starts to open due to pressure loss through needle valve



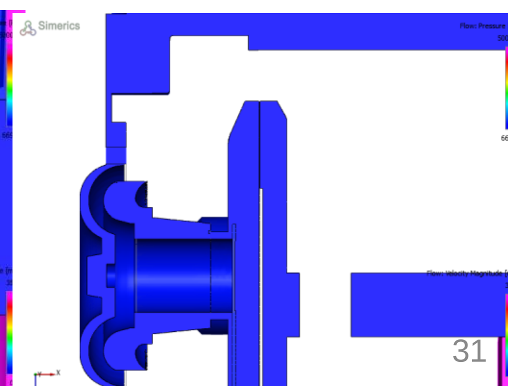

Main and Needle Valves



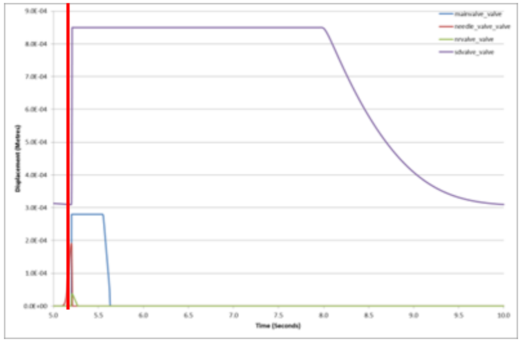

Non-Return Valve




SDV Valve

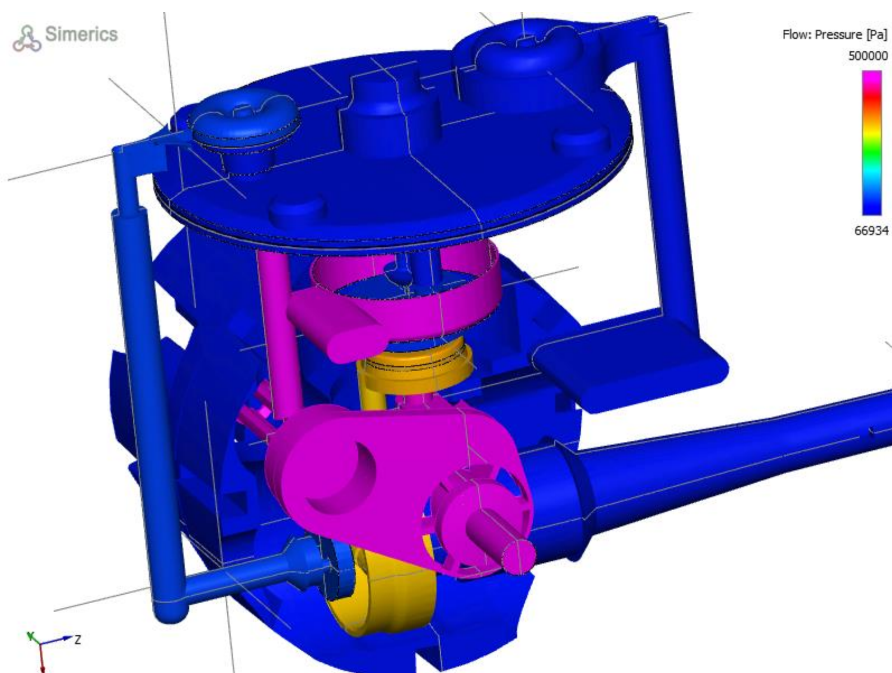


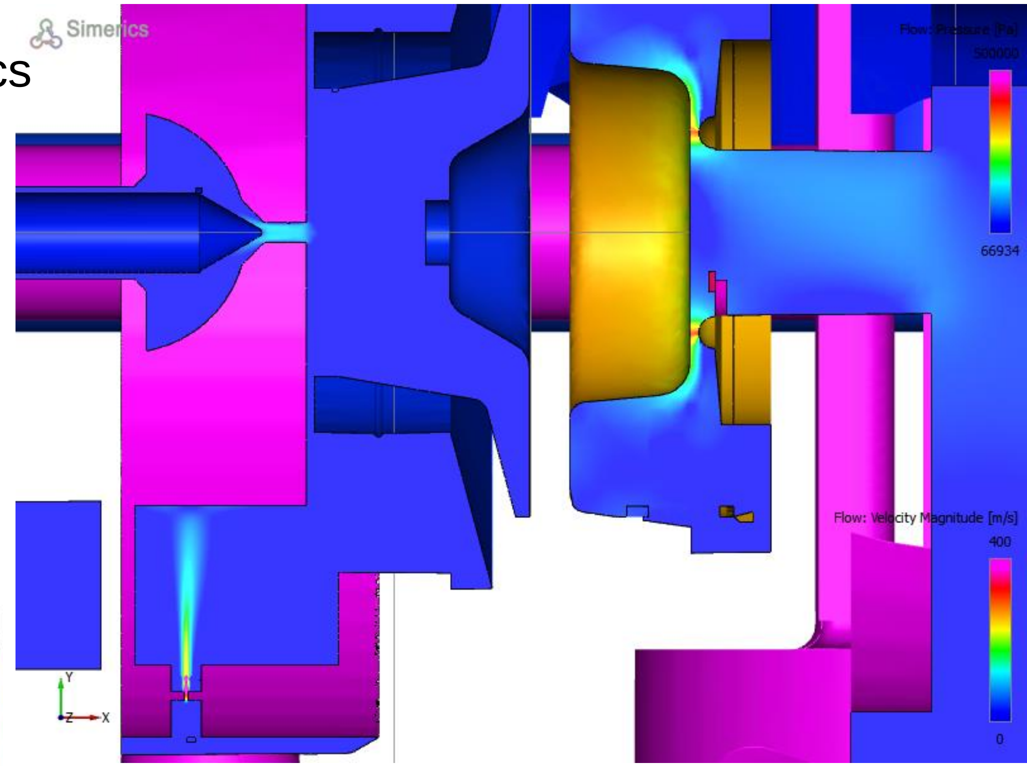
# Step 4: Main Valve Opening



Time = 5.202 secs

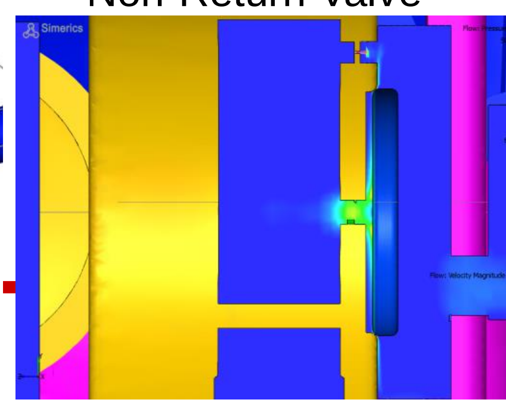
Main Valve Moving and pressurising "Blue Region"



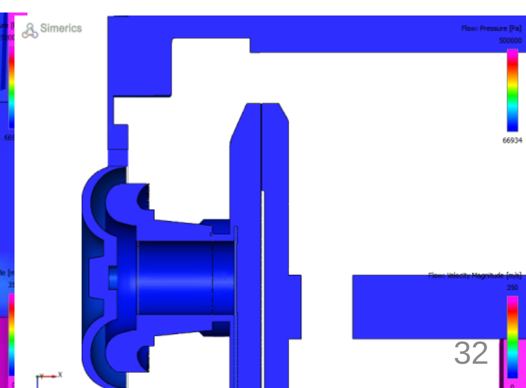




Main and Needle Valves

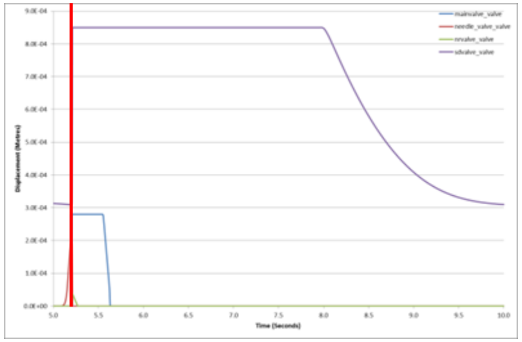
Non-Return Valve




SDV Valve

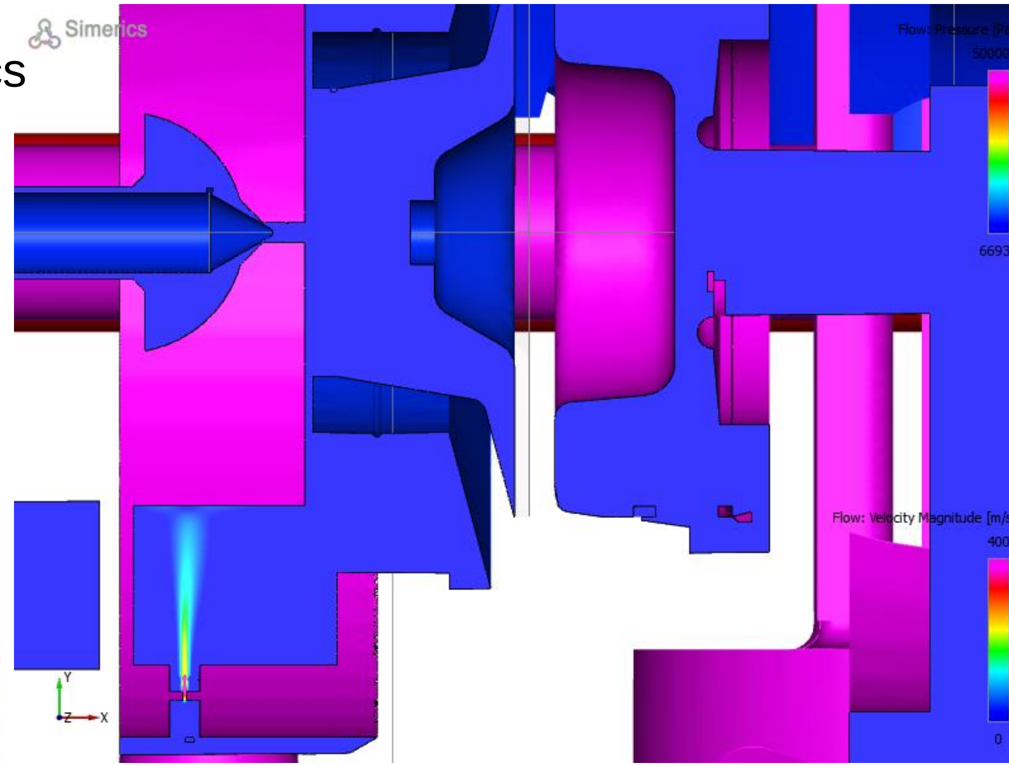


# Step 5: Needle Valve Closes

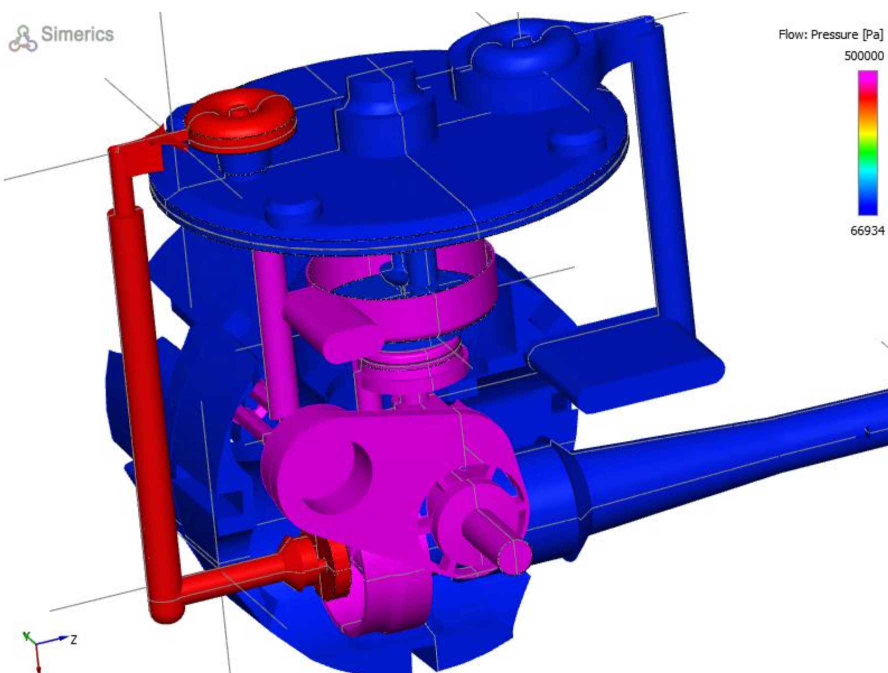


Time = 5.211 secs

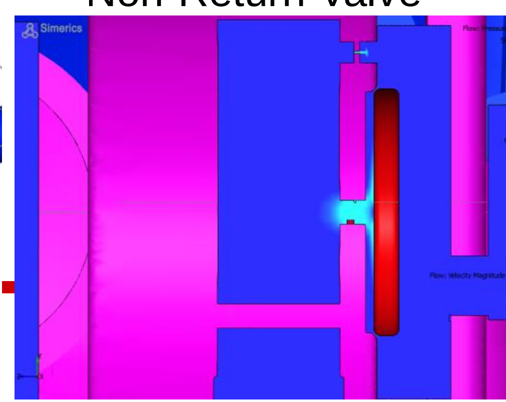
Main Valve open and pressure activates delay circuit closing needle valve.



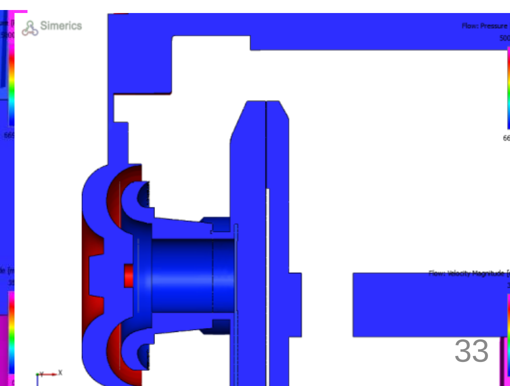

Main and Needle Valves



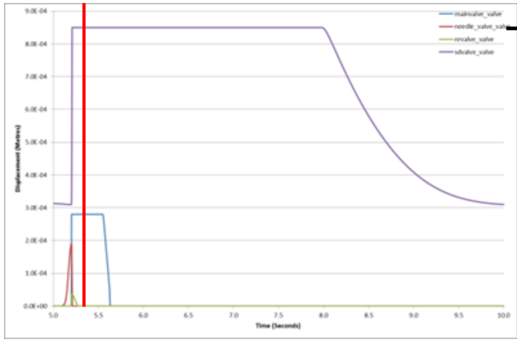

Non-Return Valve




SDV Valve

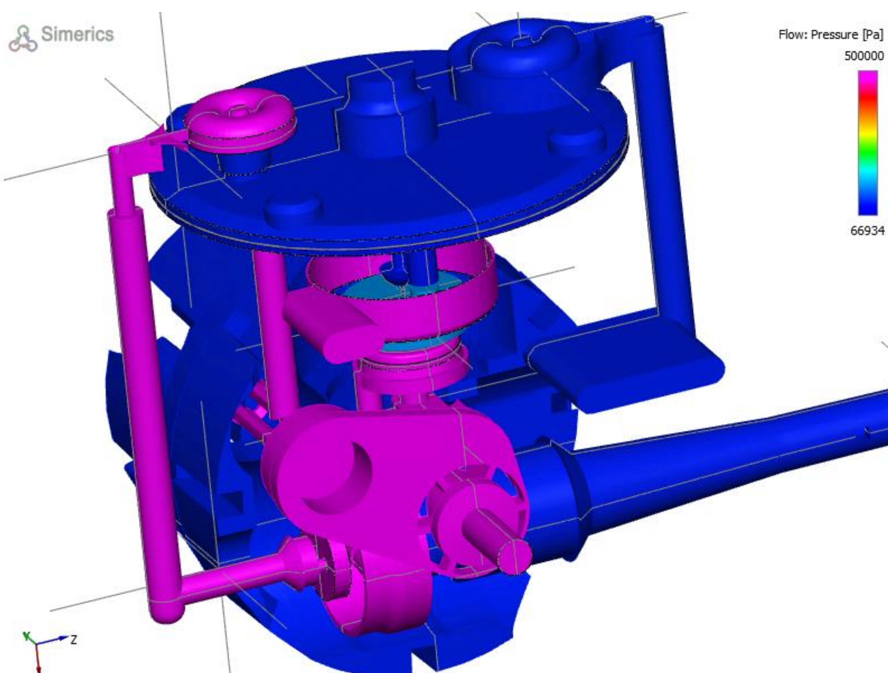


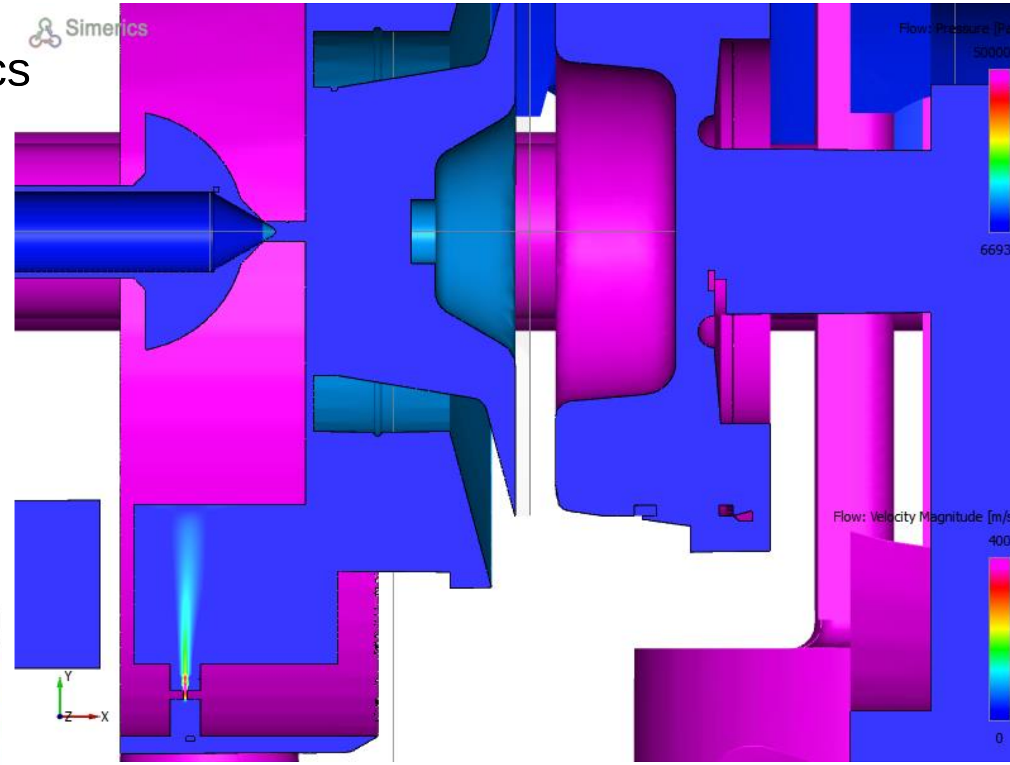
# Step 6: NRV Closure



Time = 5.253 secs

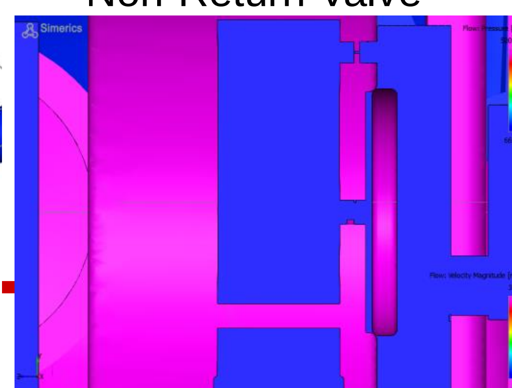
Non Return Valve closes and pressure rises in main valve upper chamber.



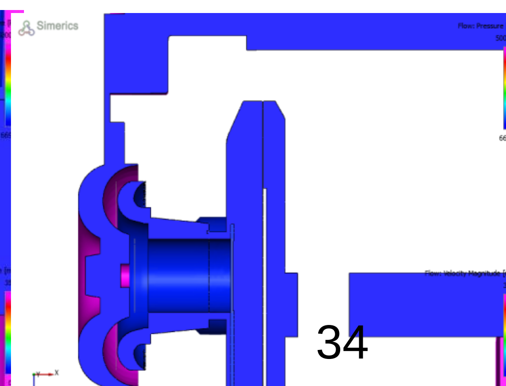




Main and Needle Valves

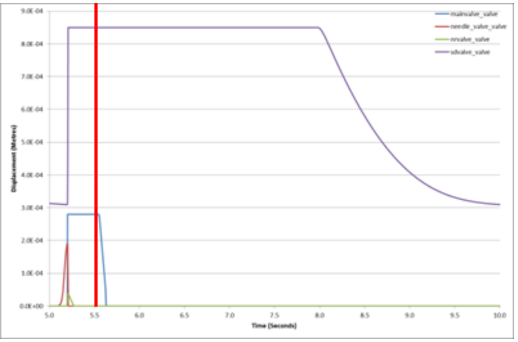
Non-Return Valve



SDV Valve

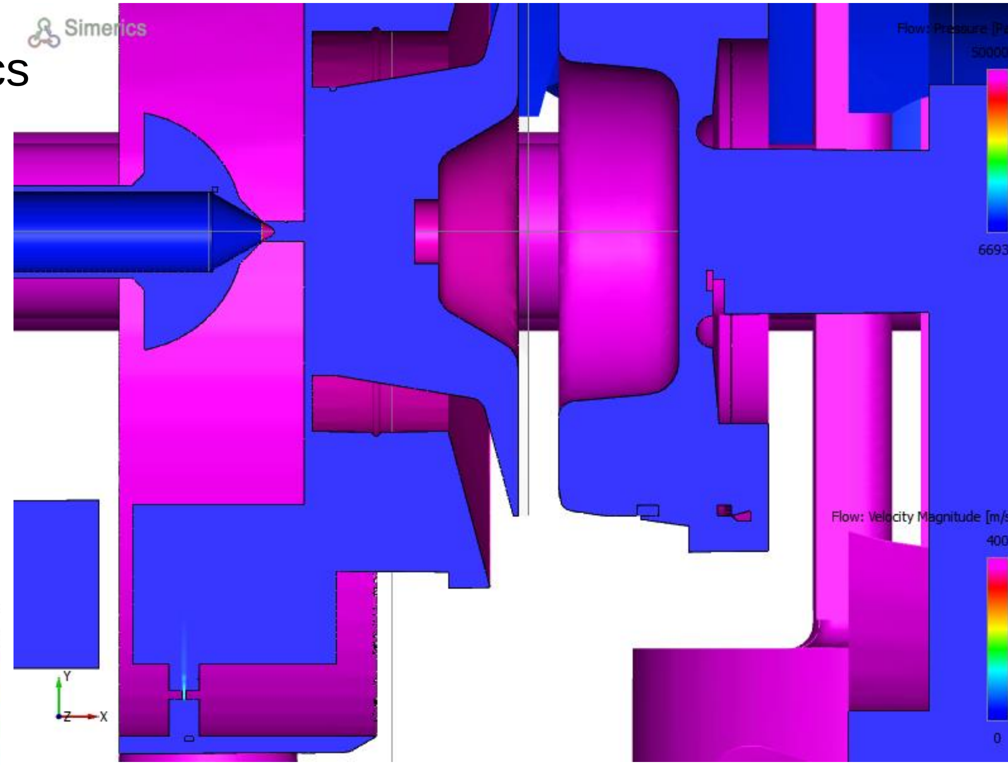


# Step 7: Main Valve Closure Start

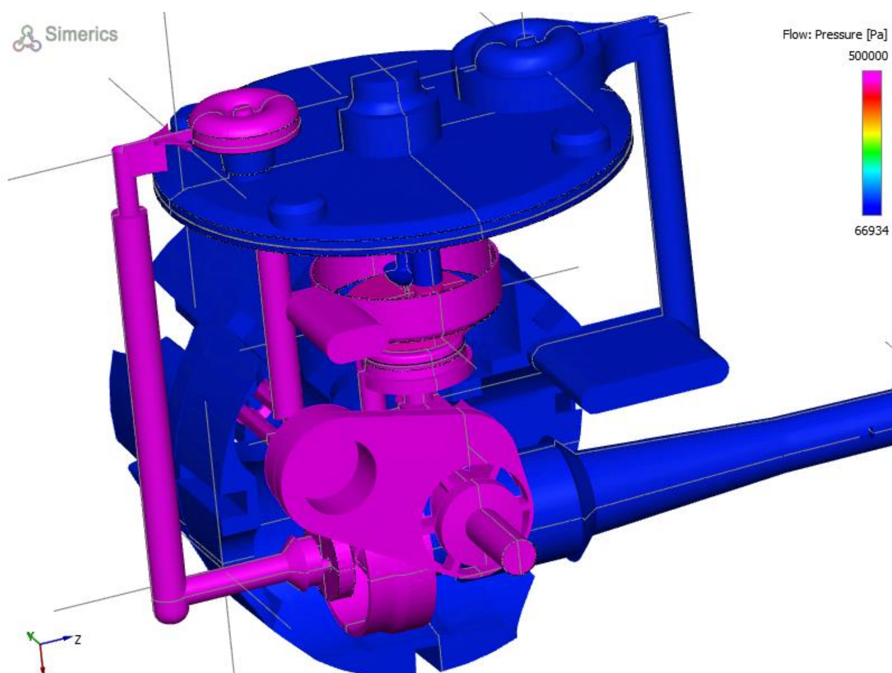



Time = 5.571 secs

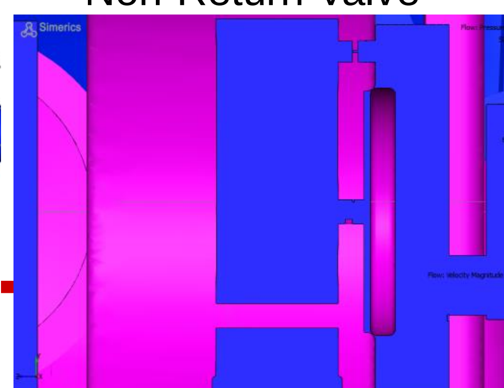
Upper Chamber reaches pressure and starts moving Main Valve



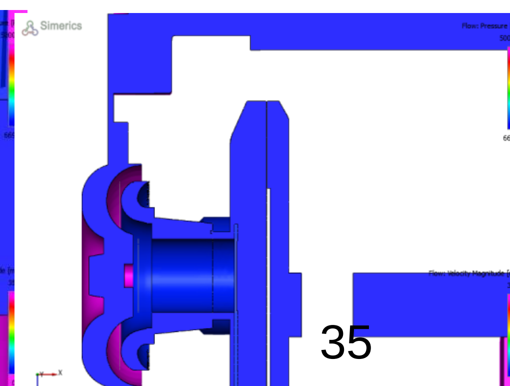

Main and Needle Valves



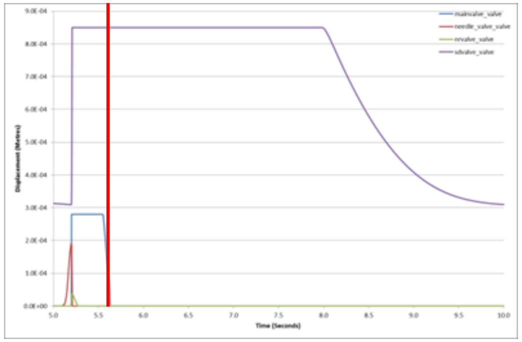

Non-Return Valve




SDV Valve

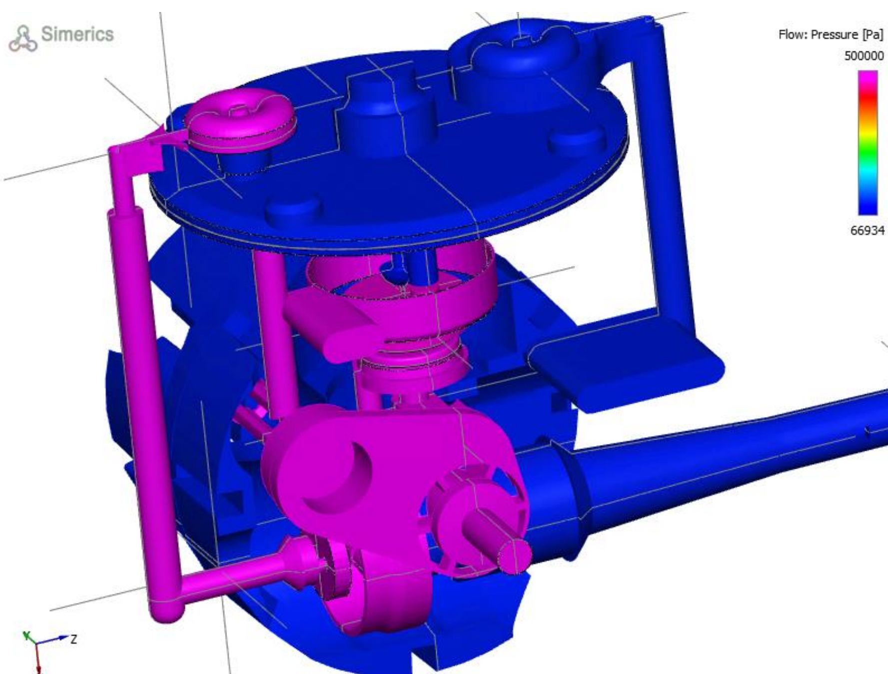


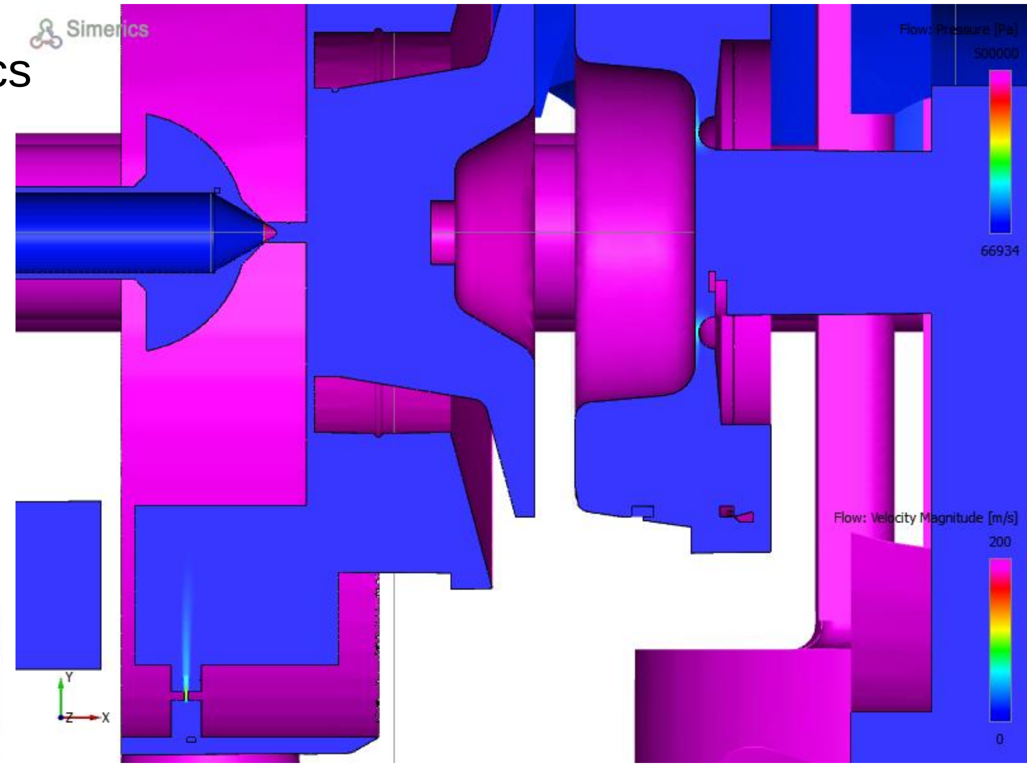
# Step 8: Main Valve Closure End



Time = 5.626 secs

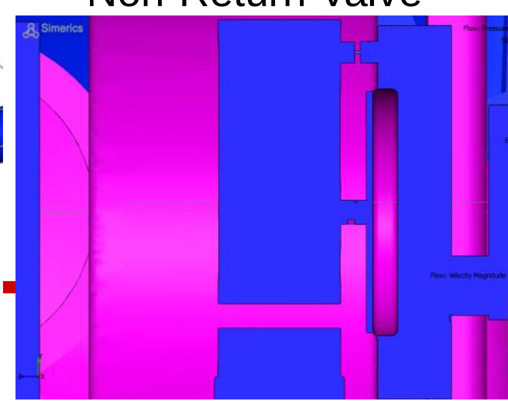
Rapid final movement of main valve due to acceleration of flow.



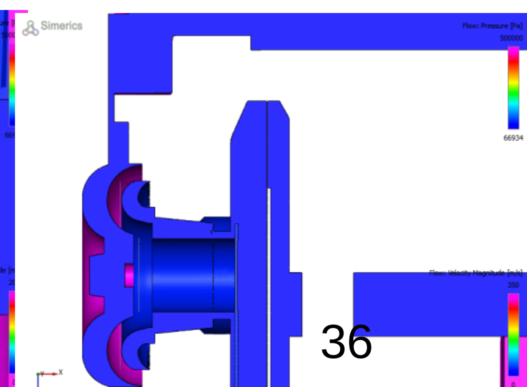




Main and Needle Valves

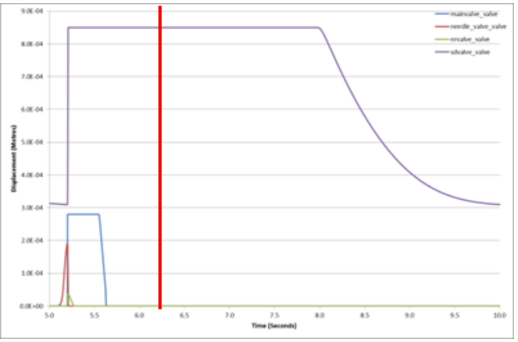
Non-Return Valve




SDV Valve

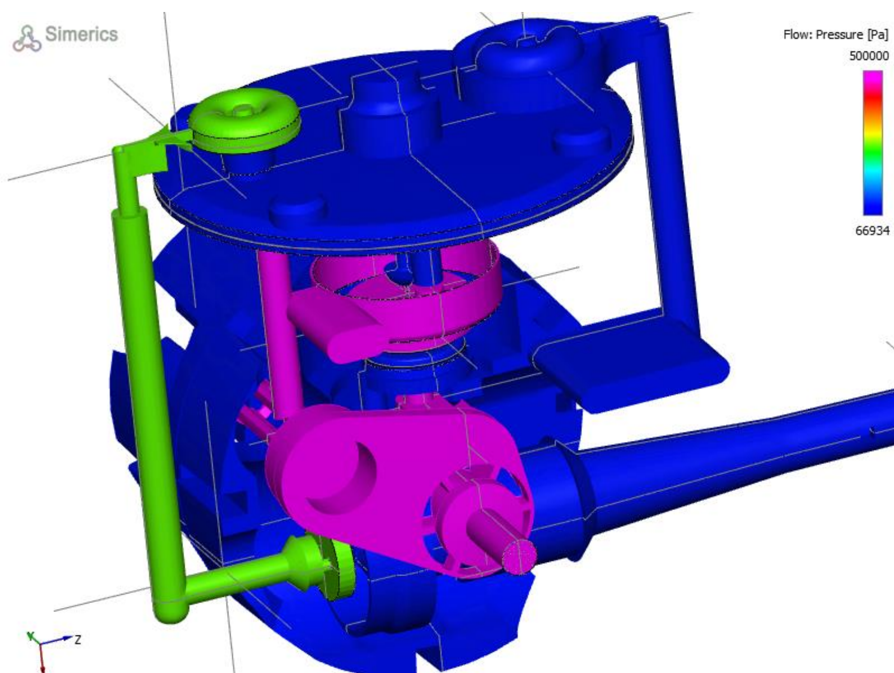


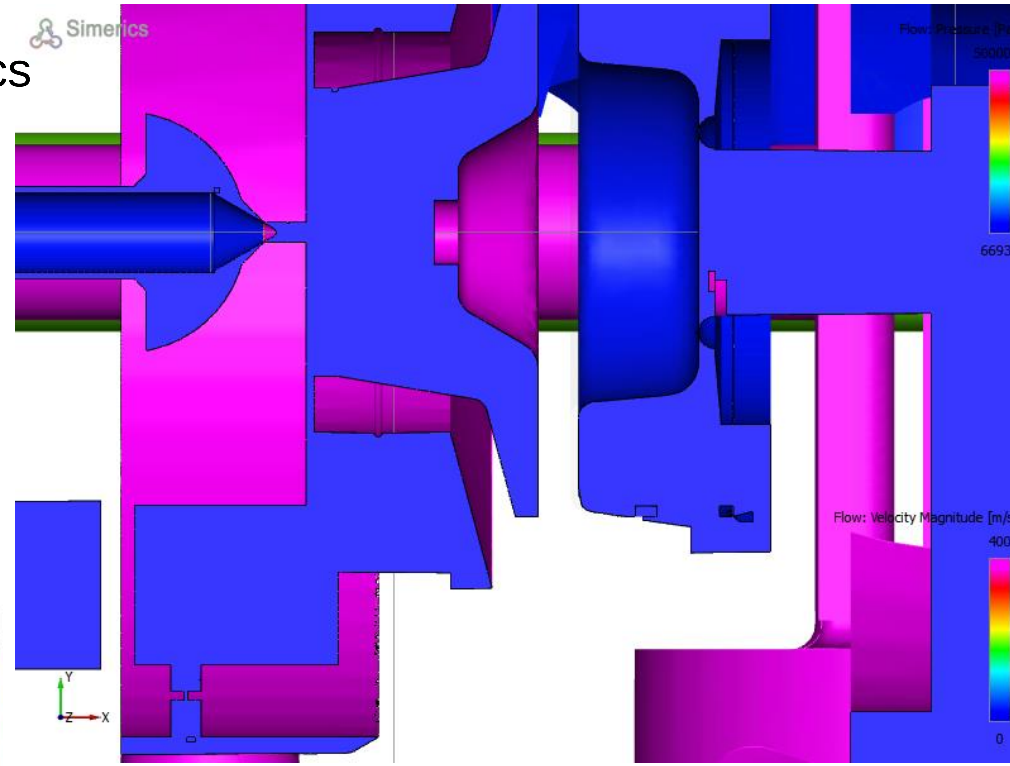
# Step 9: Main Valve Closed




Time = 6.418 secs

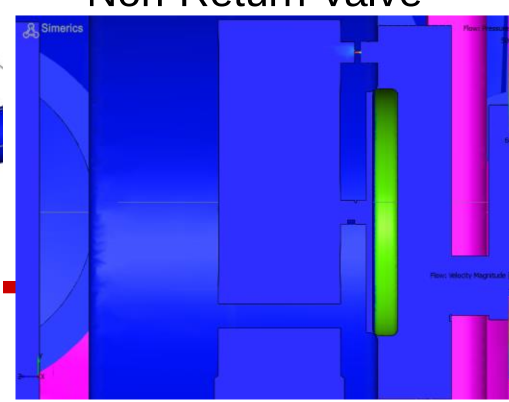
Slow leakage from Delay circuit stops any activation of Diaphragm



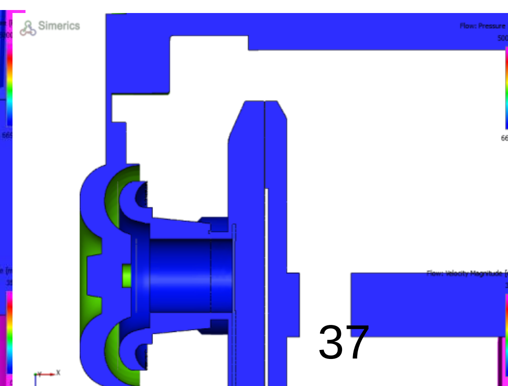




Main and Needle Valves

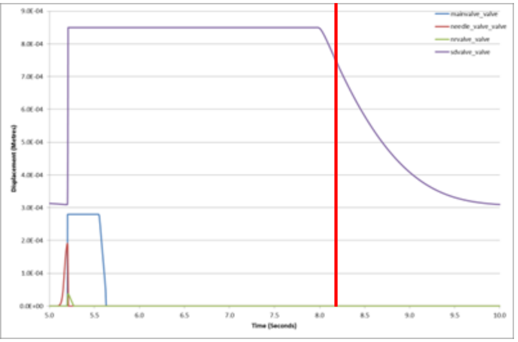
Non-Return Valve




SDV Valve

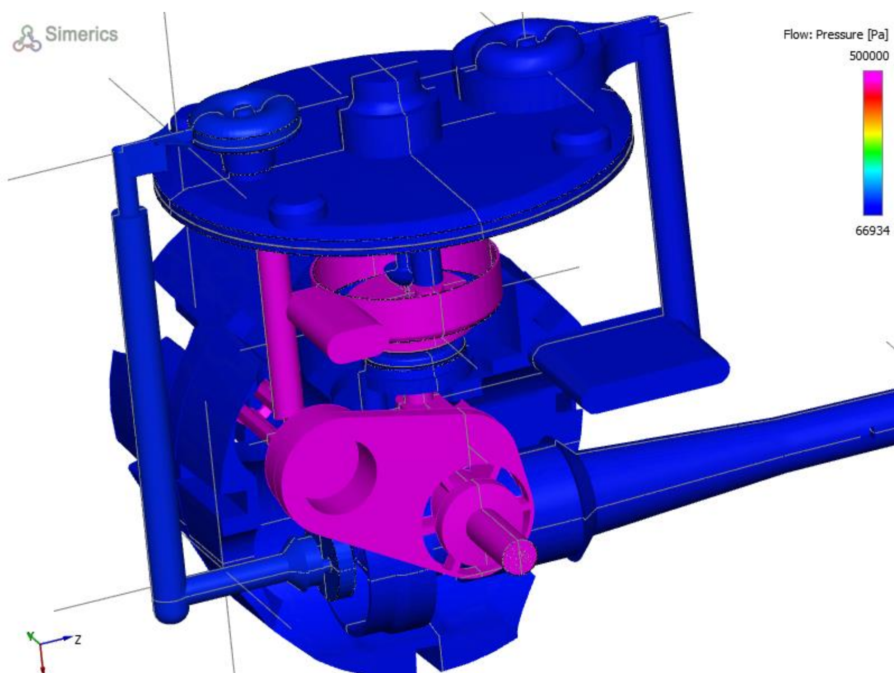


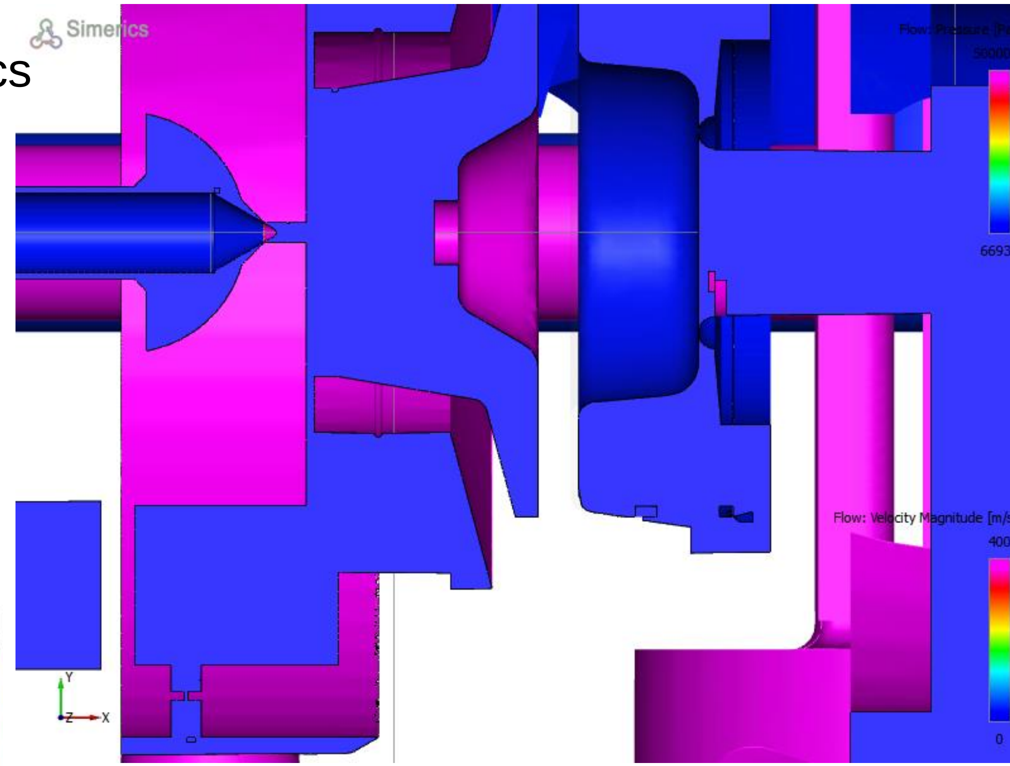
# Step 10: SDV Valve Starts Move



Time = 8.178 secs

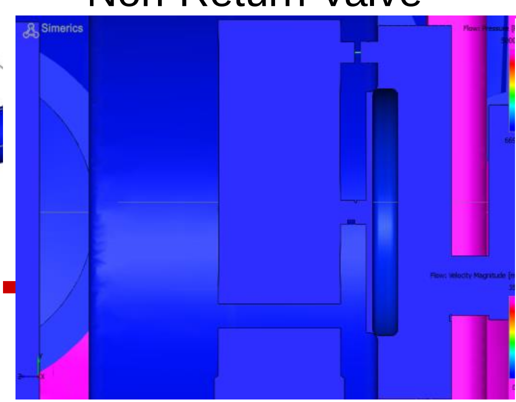
SDV valve starts closing due to reduction of pressure in Delay circuit



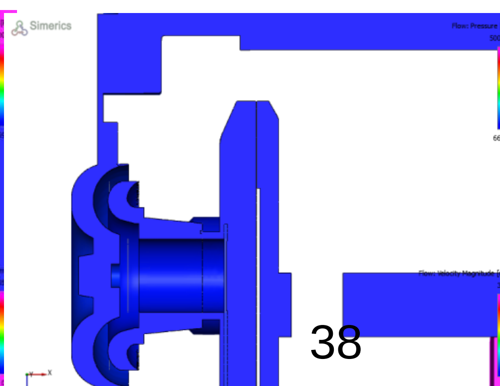




Main and Needle Valves

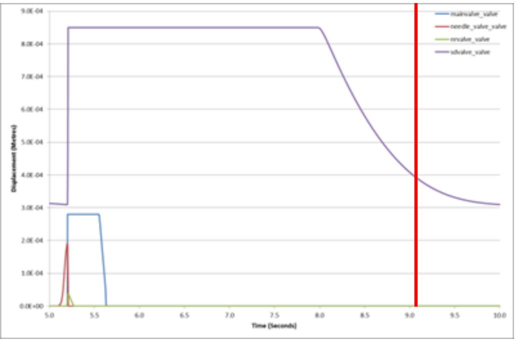
Non-Return Valve




SDV Valve

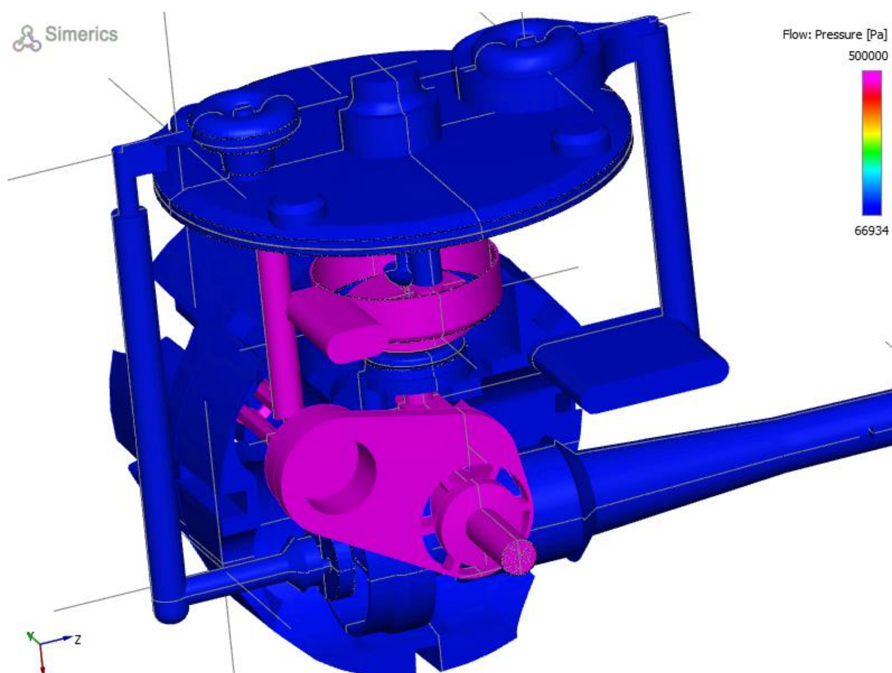


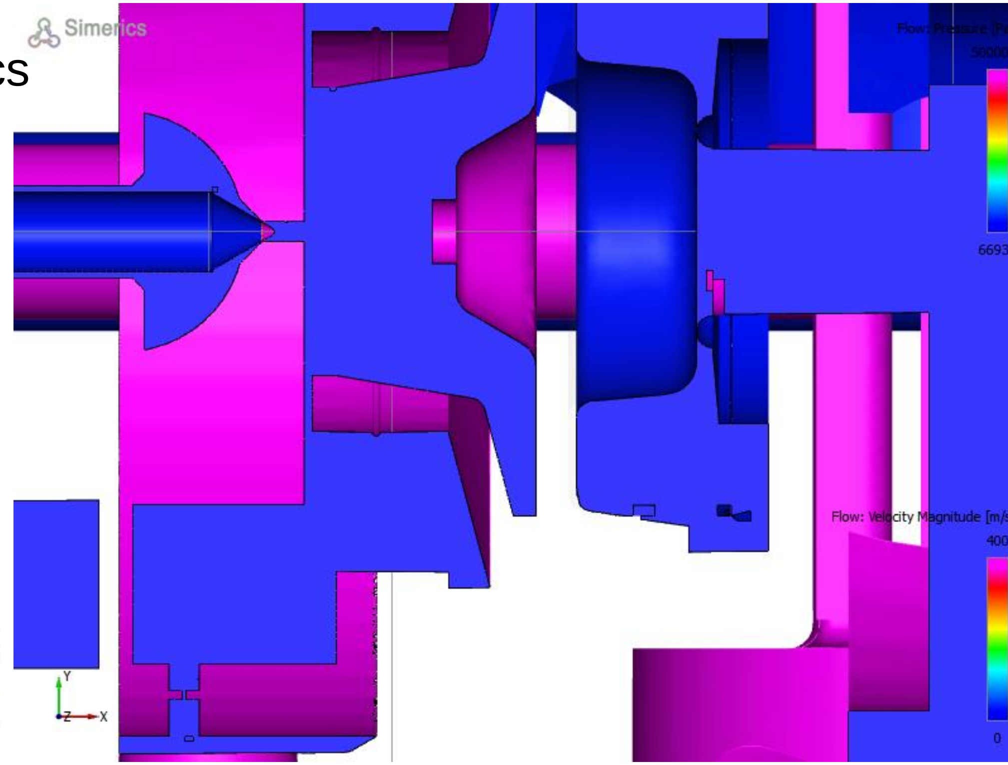
# Step 11: SDV Valve Disengage




Time = 9.059 secs

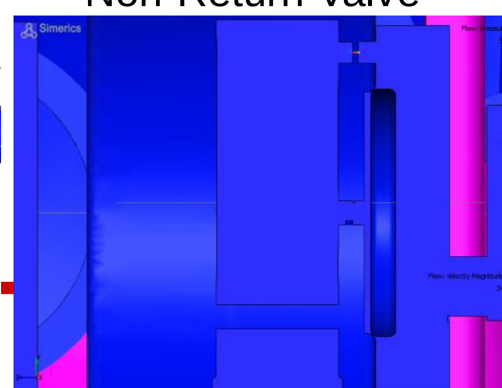
SDV Valve disengages from Diaphragm allowing next cycle to start.



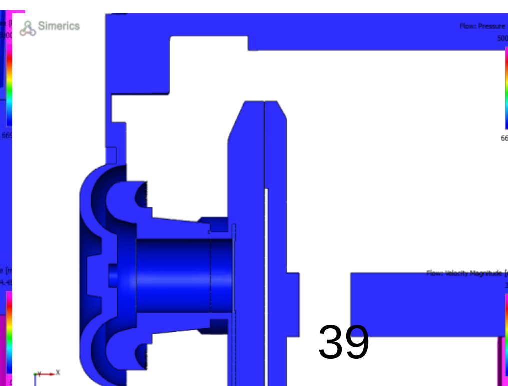



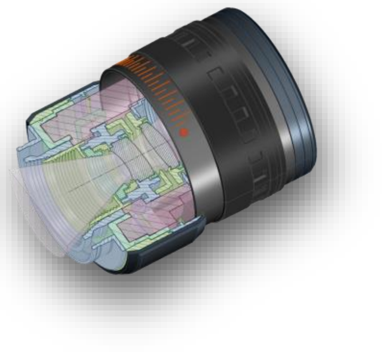

Main and Needle Valves

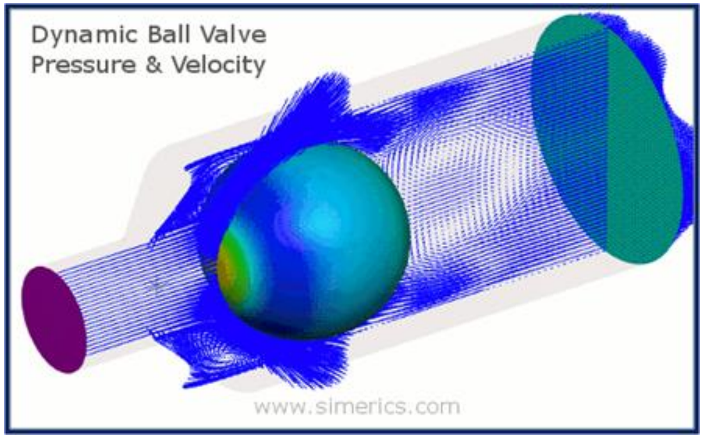
Non-Return Valve

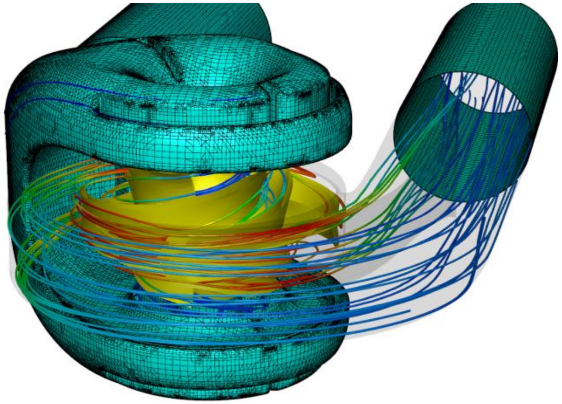



SDV Valve







# What is 8020CFD?

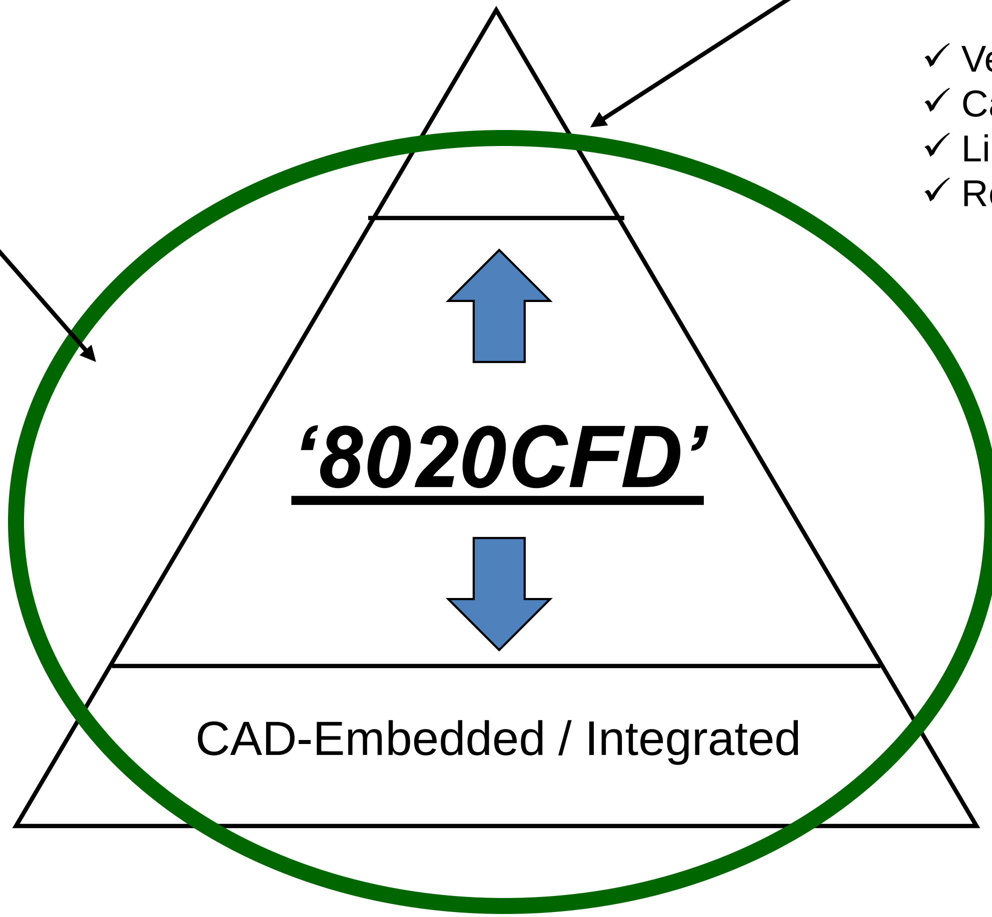
Sean Horgan

# CFD – Application/Implementation Success

CFD Analysts / Specialists

- ✓ Very Specialized
- ✓ Career in CFD
- ✓ Limited impact on design decisions
- ✓ Research based problems

Upfront CFD Engineers

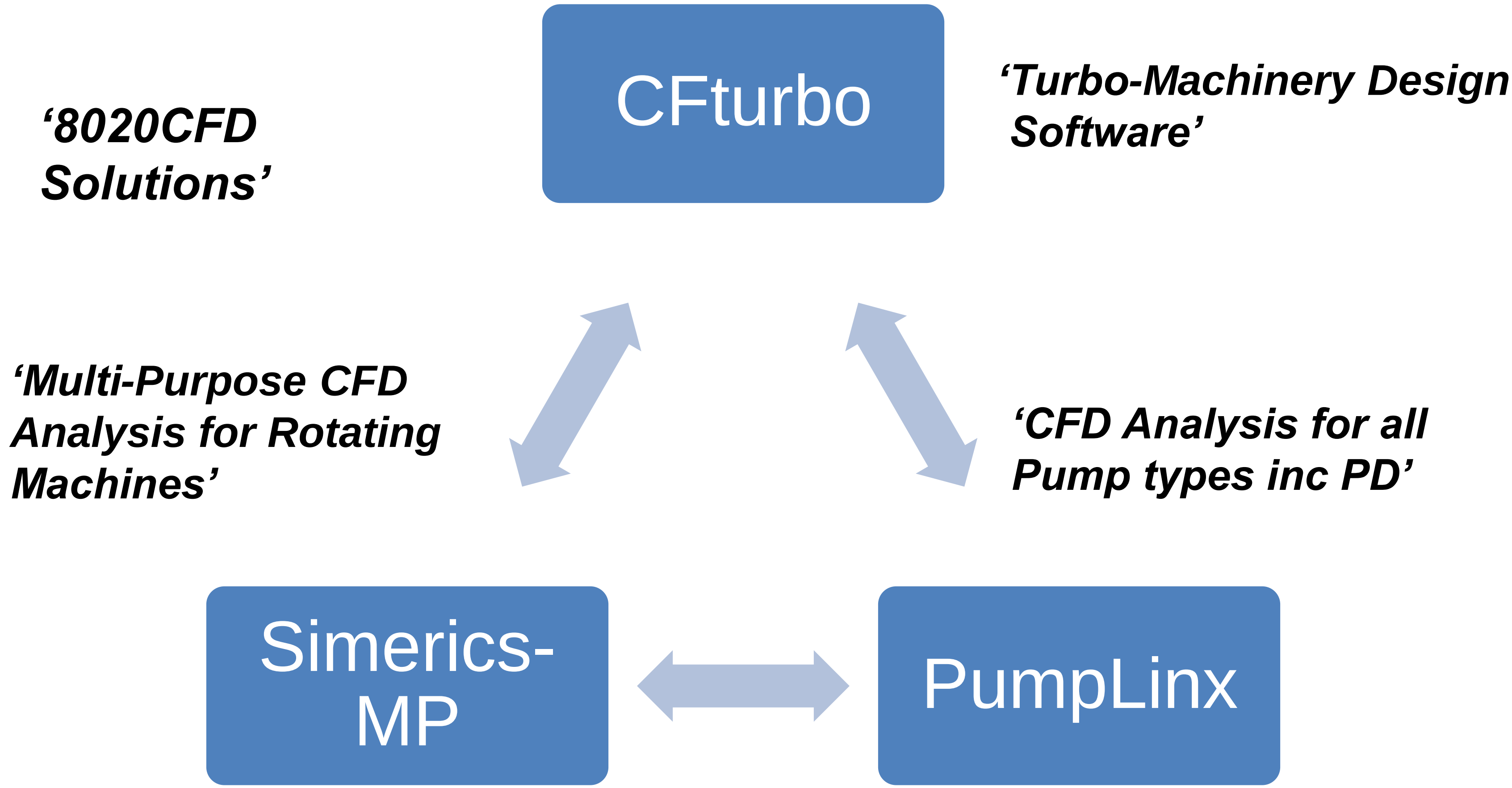



8020CFD

- ✓ Reliable and Accurate
- ✓ Tool for Product Development
- ✓ Fast Design Decisions
- ✓ Don’t want to make a career of driving CFD software

# 80/20 CFD – Simulation Experience

| Industry | Applications/Challenge |
|---|---|
| Aerospace, Auto, Propulsion | High Speed, External Aero, FSI Combustion, Acoustics, Multiphase |
| **Turbo-Machinery; Pumps, Compressors, Fans, turbo-chargers, blowers…** | **Positive Displacement Pumps, Cavitation, Flow Regimes** |
| Flow Control, Valves; Industrial Machines; Ovens, Refrigeration, Food Production… | Product Development, Cost versus Test, CAD Driven |
| Electronics, AEC (Buildings/Consumer) | Component Libraries, Vertical Output Variables, Customization |
| Process Industry, Pharm, Oil & Gas, | Material behaviour, Multiphase, Explosions, Free-surface |
| Energy, Power Production | Solar, FSI, Tidal, Heat Recovery, Fuel Cells, Wind |
| Marine, Ship Hydrodynamics | Free-Surface, Stability, Wakes |

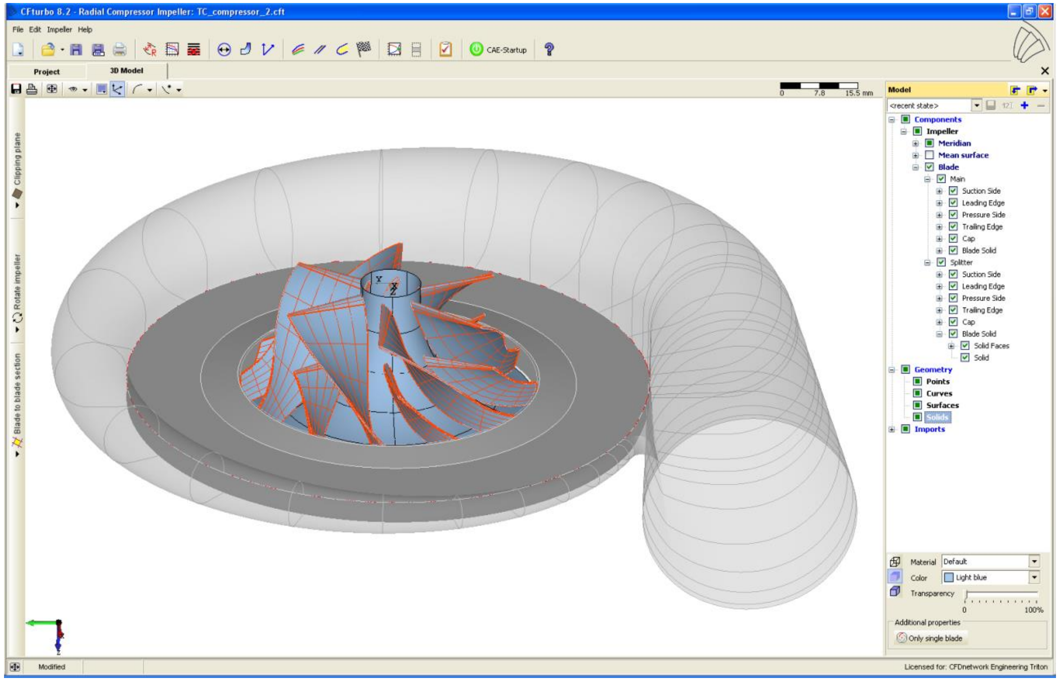

CFturbo
‘8020CFD Solutions’
‘Turbo-Machinery Design Software’
‘Multi-Purpose CFD Analysis for Rotating Machines’
‘CFD Analysis for all Pump types inc PD’
Simerics-MP
PumpLinx

# CFturbo®

- Origin at Dresden University of Technology
- Professional software development started 2003
- On market since 2005
- ~ 70 clients worldwide
- Available modules:
  - Impeller
  - Stator
  - Volute

  to design

  - Pumps
  - Blowers/Ventilators
  - Compressors
  - Turbines



# CFturbo - General Introduction

| Design of radial and mixed-flow | Available modules |
|---|---|
| ▸ Pump | ✓ Impeller |
| ▸ Ventilator and Blower | ✓ Stator |
| ▸ Compressor | ✓ Volute |
| ▸ Turbine | |

- Combination of established design theory with latest empirical knowledge
- Frequently proven results for practical use
- Easy to use
- Direct interfaces for many CAE-software packages

- Runs on Windows XP/ Vista/ 7
- Node locked or floating license
- User interface in English language
- Online-Help in German and English language

# Simerics Inc

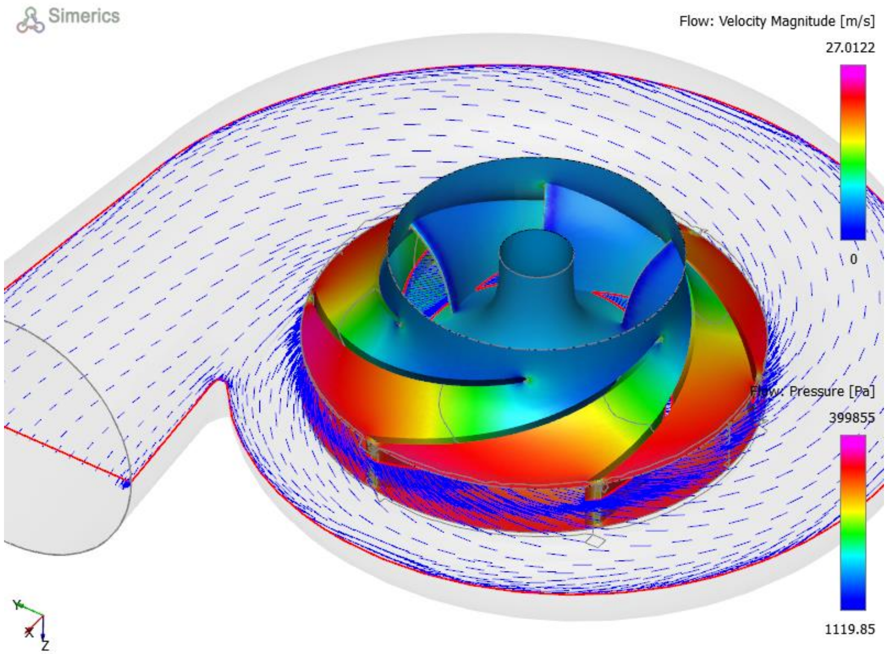
## Design Better Pumps or Rotating Devices

**Mission: to provide designers the most effective simulation tool possible (**predicting **cavitation, pressures and flow)**

- **Produces/Services:**
  - **PumpLinx: Pump Design Tool**
  - **Simerics-MP: Multi-Purpose CFD Tool**

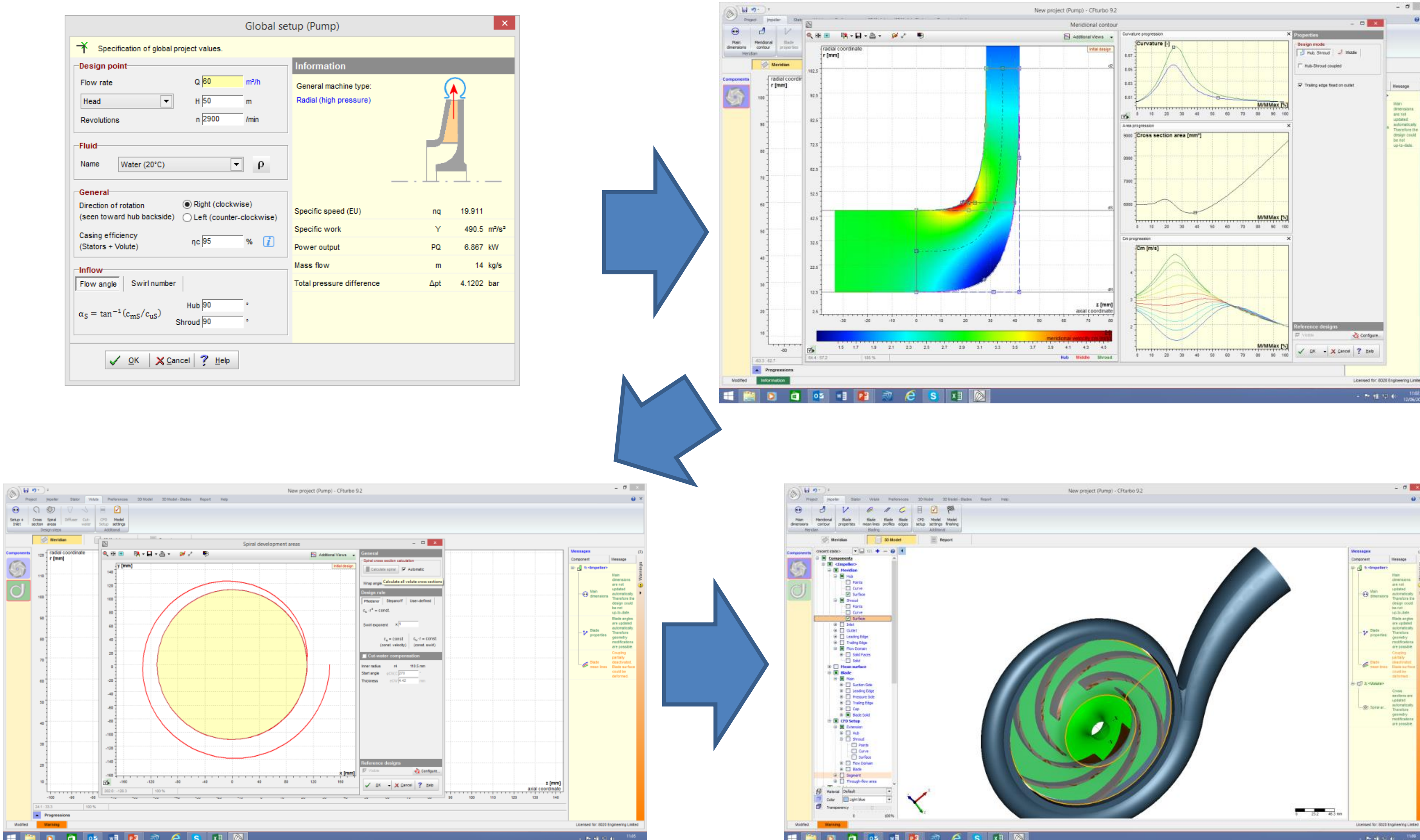
**Simerics Inc - Headquarters in Huntsville, Alabama**

# *'Turbo-Machinery Process Demonstration' CFturbo – PumpLinx – Centrifugal Pump*

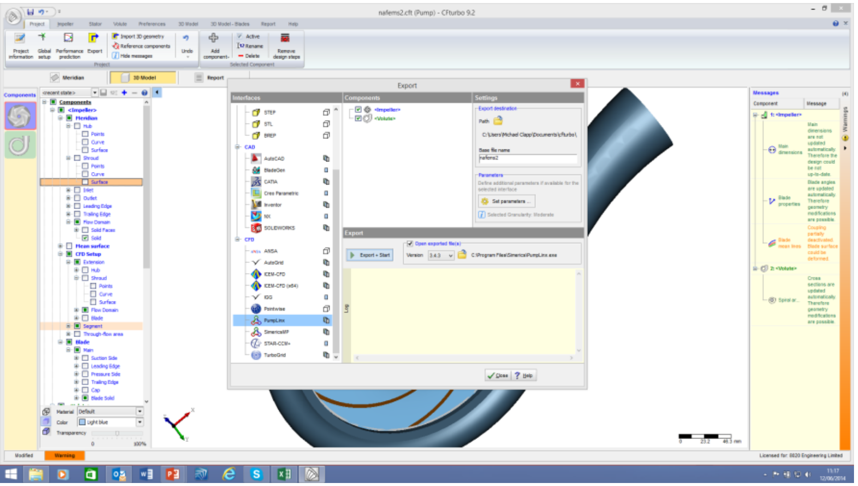


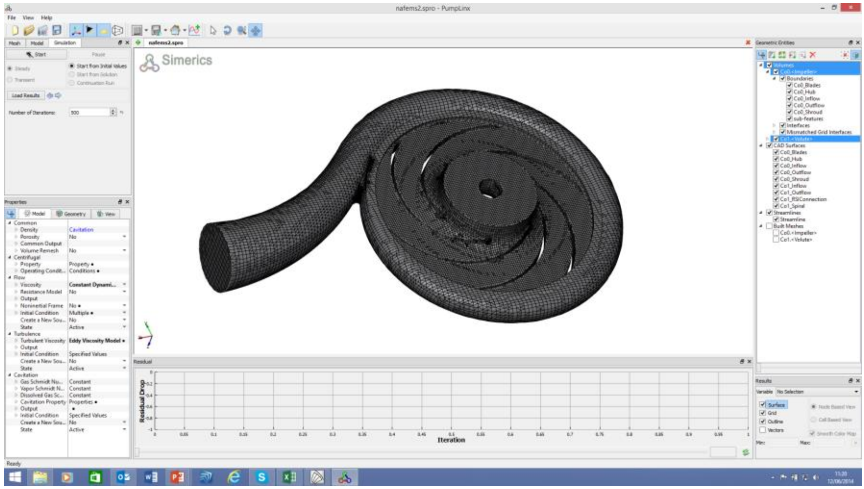

Mike Clapp

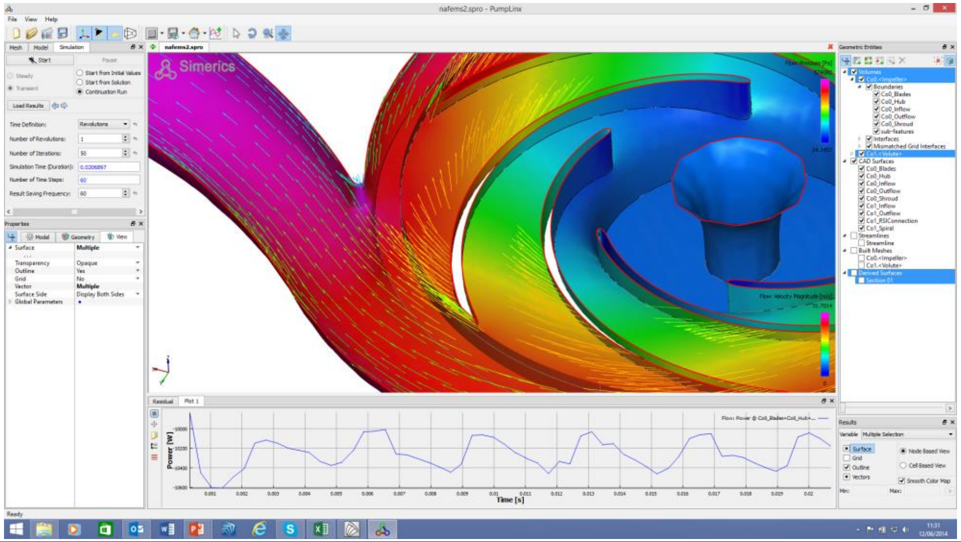
# From Design Point to 3D Geometry



# … to CFD Simulation




In a 10 minute live demonstration

# 80/20 Engineering

- *The Pareto Principle, often called "**The 80/20 Rule**", means that in anything, the few (20 percent) is usually responsible for the many (80 percent).*

- Application to the Engineering 'Simulation' Process

  80% of business value delivered by 20% of analysis resources deployed for simulation.

  - ***Critical that simulation resources are used in an optimum way***

- Focus on Product Development

  80% of product performance, or project success is defined by 20% of the development life-cycle.

  - ***Critical that performance information is available at the earliest opportunity.***

# Pushing the Boundaries of 'UpFront CFD'

Sean Horgan, Mike Clapp
80/20 Engineering Ltd
+44 (0) 1494 616020
info@8020Engineering.com
support@8020Engineering.com
www.8020Engineering.com