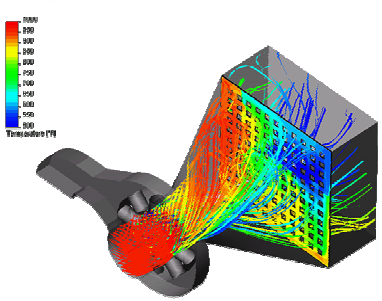
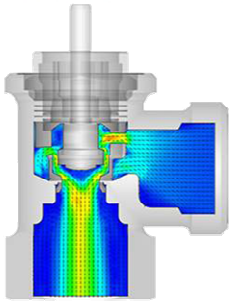
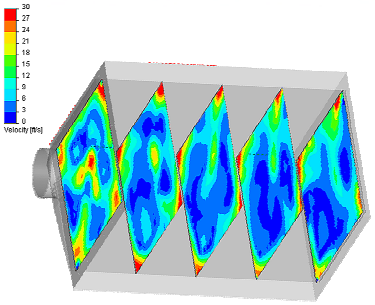
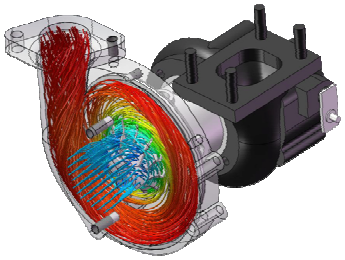
# Mentor Graphics' CFD Software Capabilities for HVAC Equipment Design & Manufacturing







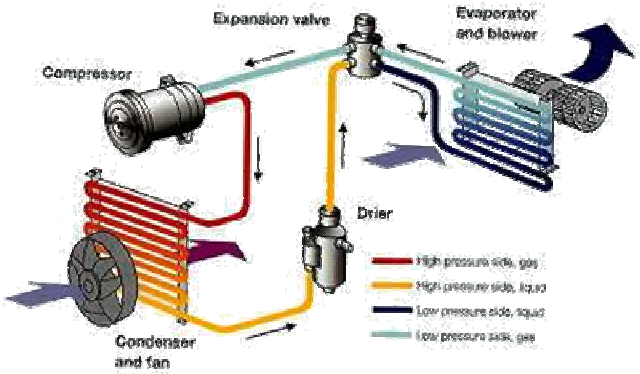


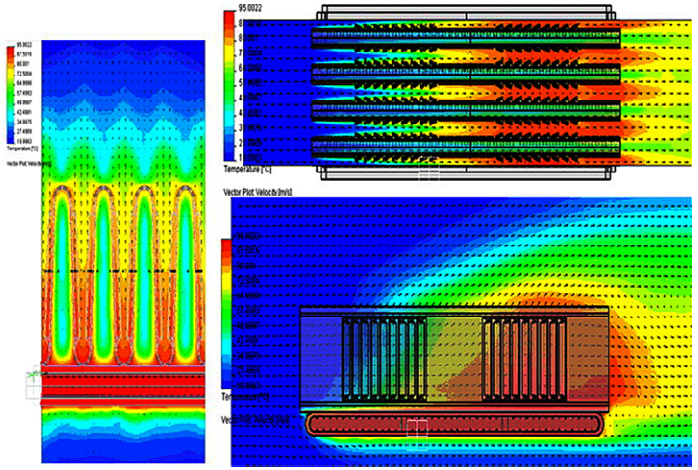
- Simulation of airflow and heat transfer using 3D CAD models directly with no need for data translations or copies
- Solving airflow/thermal problems in complex devices including rotating geometry, curvilinear geometry, and thin shells
- Analyzing and solving advanced physics including phase change and transient processes
- Analyzing and solving both internal and external flows
- Develop "green" equipment that consumes less energy and maximizes cooling/heating efficiency
- Optimize temperature and humidity control
- Predict movement and distribution of airborne particulates
- Reduce equipment noise

Mentor Graphics

# Air Conditioners

- **Primary goal** - analyze and improve the efficiency of equipment (as delivered and in situ):
  - Air intakes
  - External flow of exhaust gas
  - In a building or room
  - Heat, cool, humidify and dehumidify air as well as filter particles.

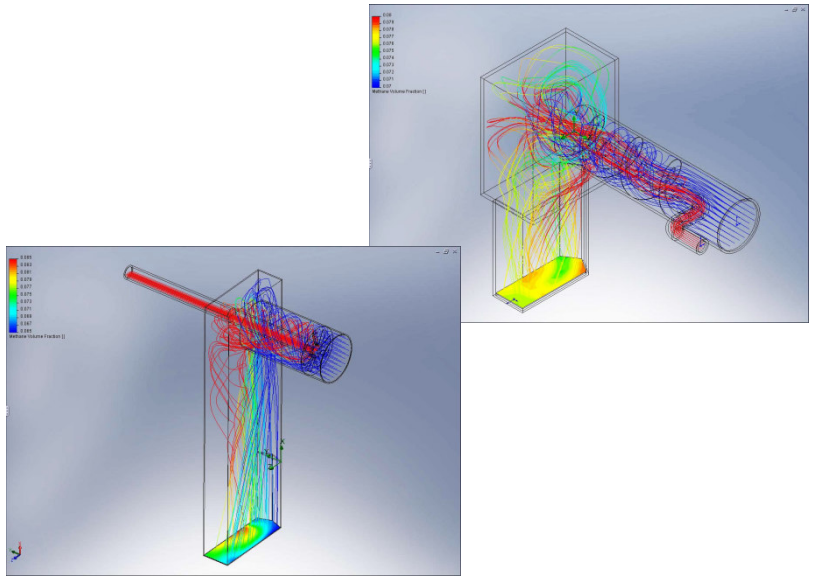


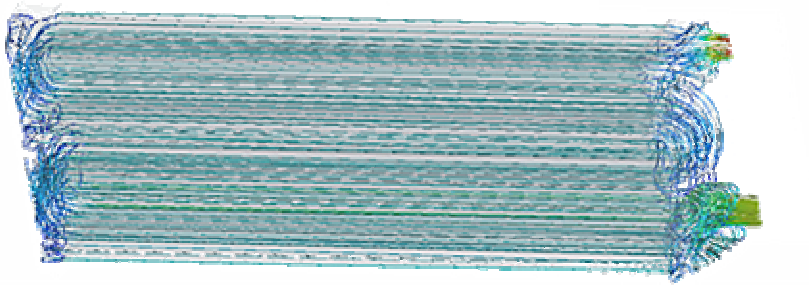



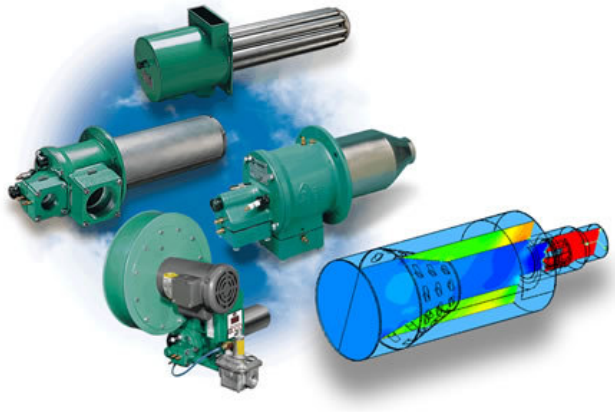
*An air-conditioning heat exchanger thermal analysis*

# Boilers, Burners & Heaters

- **Primary goal** - analyze and improve the efficiency of boiler components & units:
  - Air intakes
  - Combustion & heat transfer
  - Exhaust gas flows
  - Heat exchanger pipes & pipework

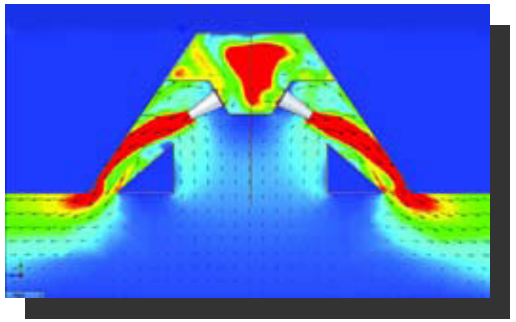


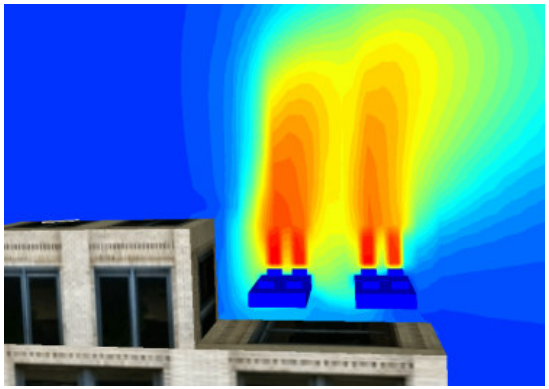




# Chillers & Condensers

- **Primary goal** - analyze and improve the efficiency of equipment (as delivered and in situ):
  - Air intakes
  - External flow of exhaust gas
  - Cooling air flows in a building or a room or externally to the atmosphere

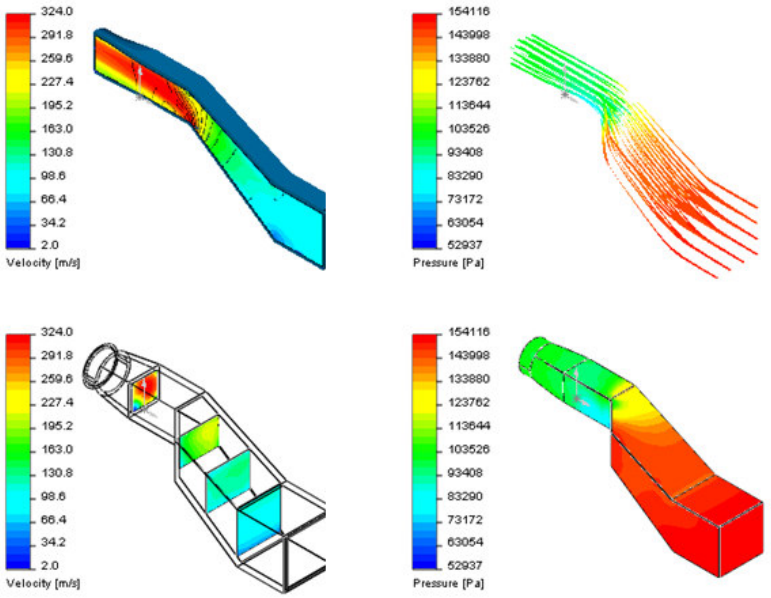


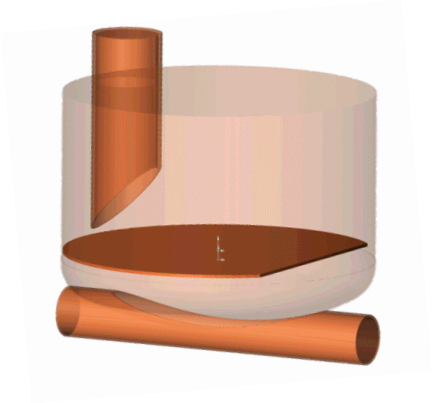


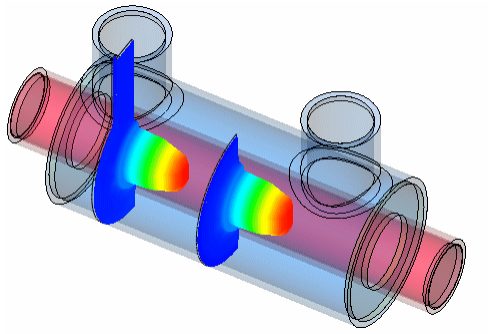
# Ducts, Piping, Bends, Manifolds, Fittings…

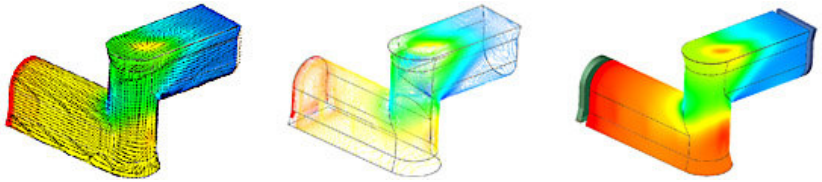


- **Primary goal** - Analyze and improve flow performance and thermal behavior:
  - Flow rate distribution
  - Pressure drop
  - Transient simulations
  - Temperature distribution – subsequent structural analysis

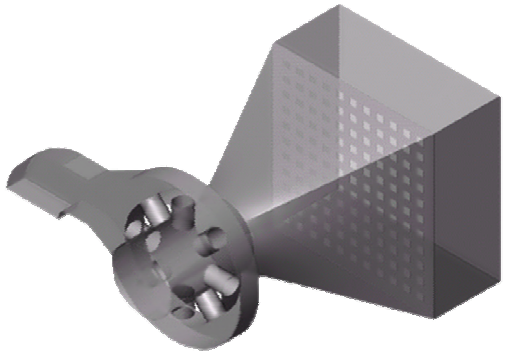


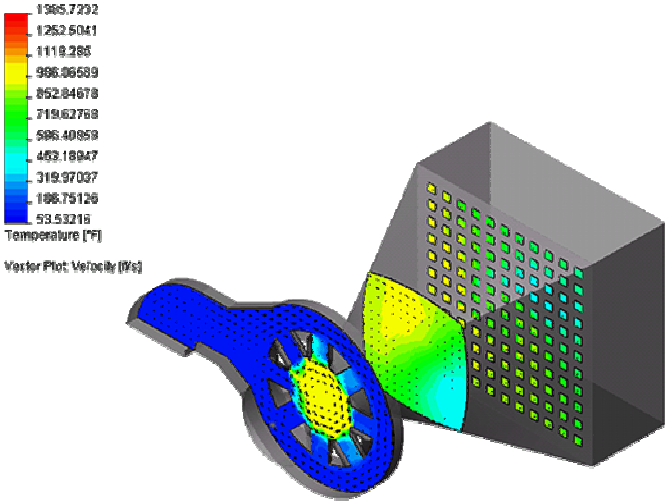



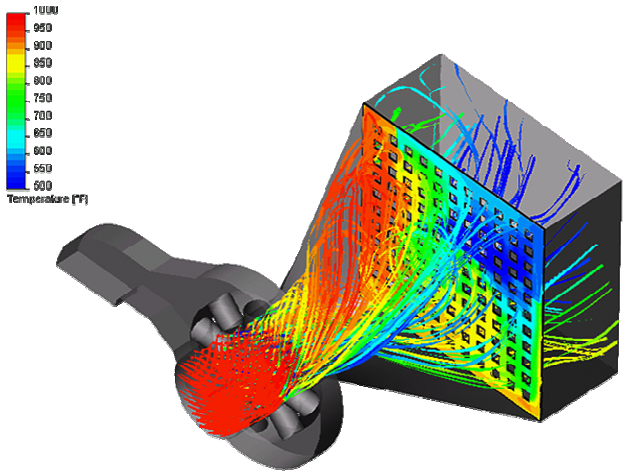




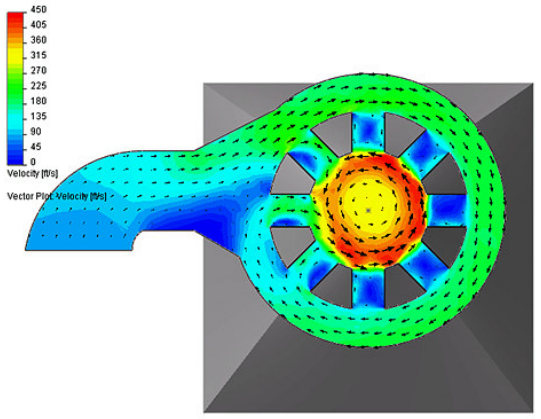
# Flow through an Industrial Heating Duct Fan-Diffuser-Perforate Plate arrangement





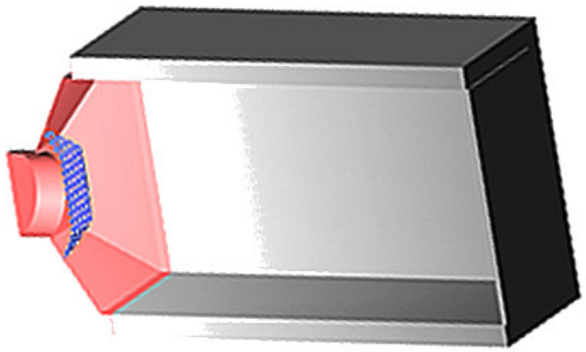



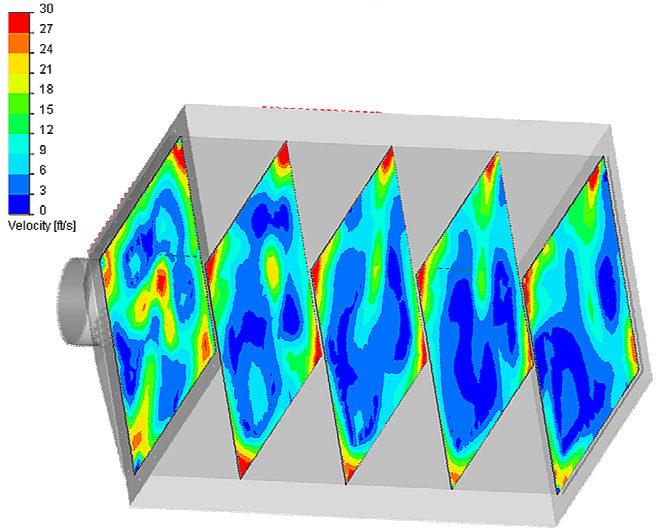

- In this industrial ducting associated with a heater the fan in feeding flow into a square diffuser section
- Pressure drops in such situations can be very high but with CFD you can determine what sort of plate within the ducting will smooth the air flows with the least pressure drop possible

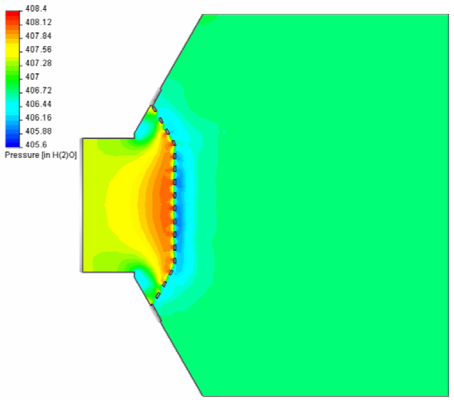



Mentor Graphics

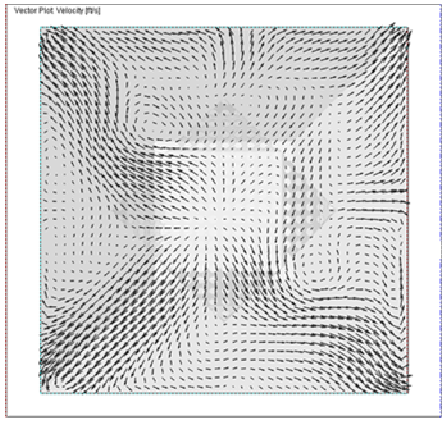
# Flow through a Perforated Diffuser





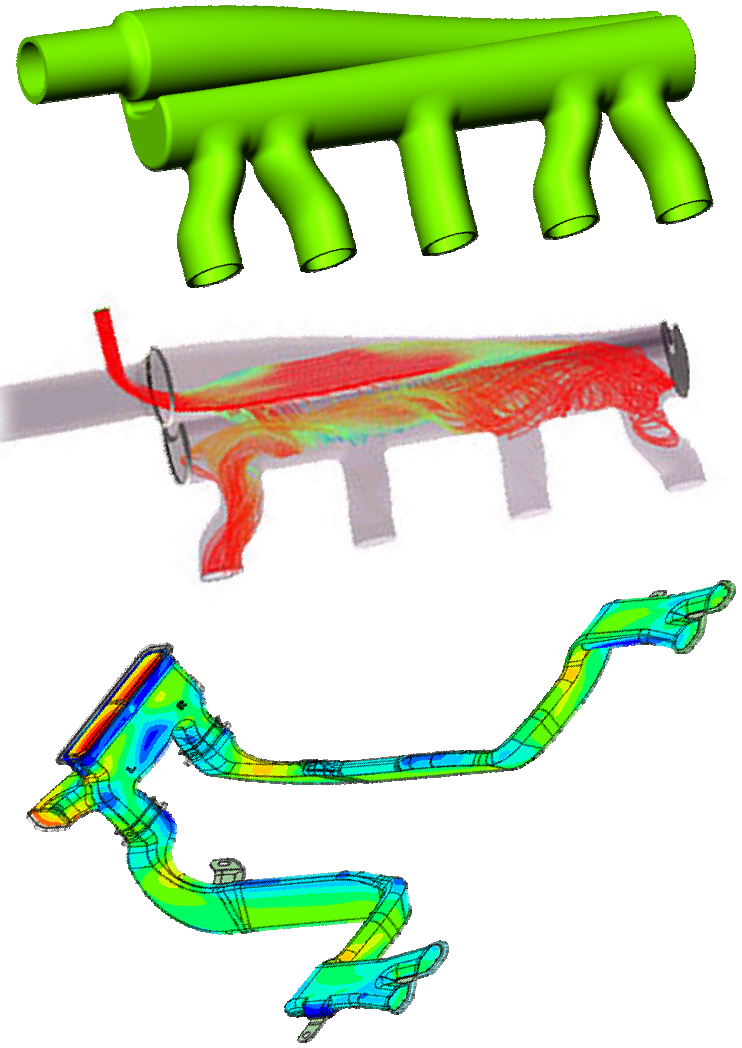




- Diffusers are frequently used to smooth flows through ductwork in expansion situations
- FloEFD is ideal for picking up all the details of a diffuser plate geometry plus the complex flow fields resulting from this interaction



# Flow in Complex Ductwork & Piping

- Frequently within the HVAC industry you have to deal with complex ductwork and piping that has to fit into a predefined space
- CAD-embedded CFD software is ideal for picking up such complex geometries and simulating various parametric arrangements to determine the lowest pressure drop or best flow split
- Here several examples of flow pathlines and pressure drops are illustrated from real-world FloEFD examples

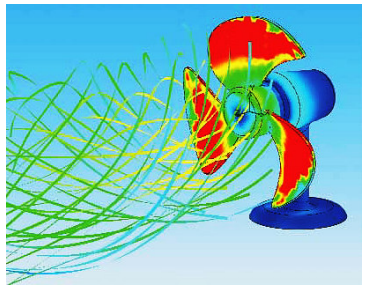


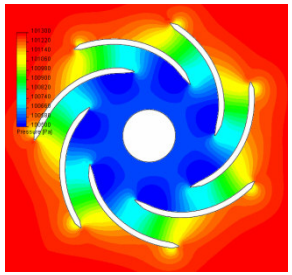
Mentor Graphics

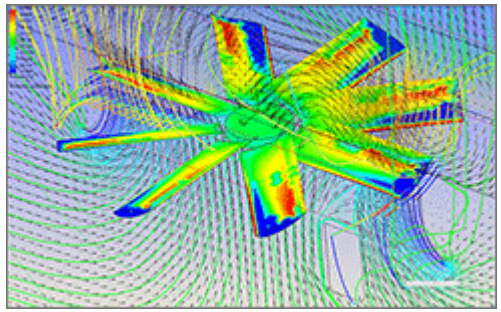
# Fans, Compressors & Blowers

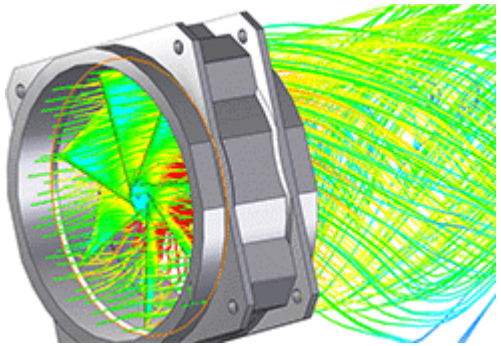


- FloEFD can easily model axial, centrifugal, desk, roof and crossflow fans
- Fan design aimed at creating a uniform airflow in its exhaust:
  - Air conditioning fan for a large vehicle
  - Multiple 'what if' studies performed
  - Optimization of fan blade and volute geometry
  - Calculate fan performance



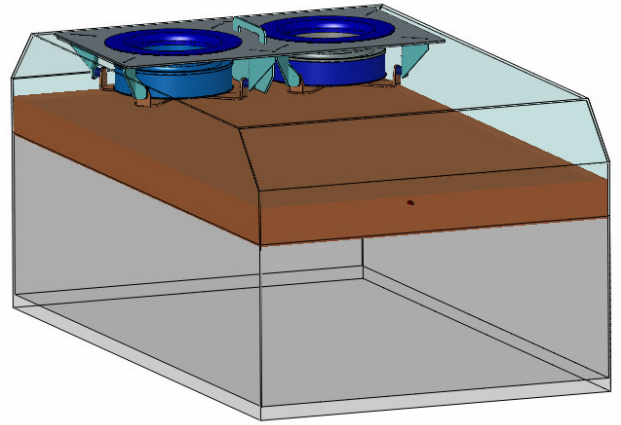


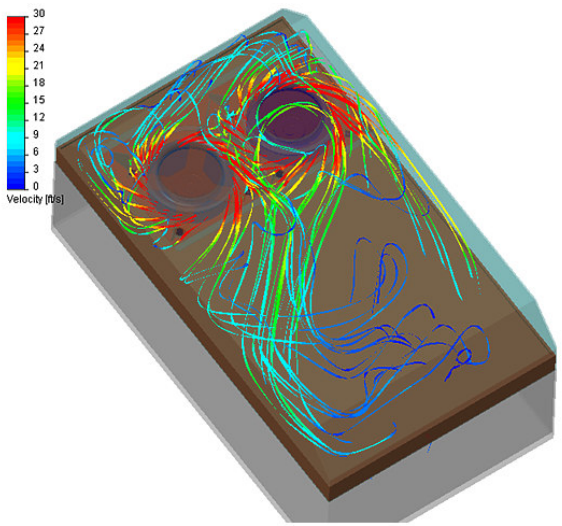


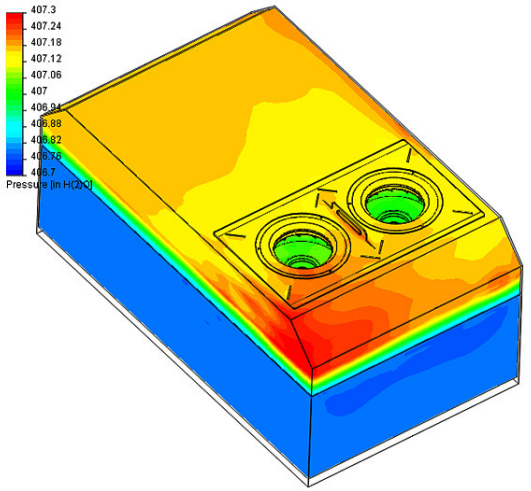


# Airflow Through a Plenum/Filter Assembly with Associated Fans

- This plenum example illustrates how FloEFD has been used to model high pressure loss zones across filtration media
- CFD is ideal for predicting:
  - Pressure drops across the system
  - Flow patterns in the plenum, and
  - Where particulates will end up



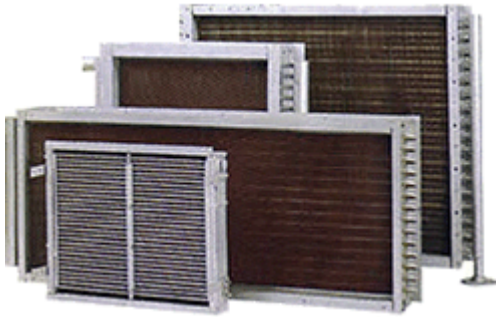


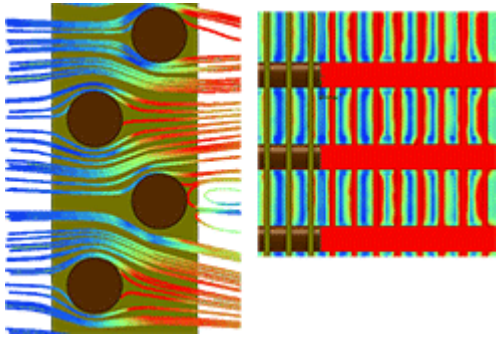




Mentor Graphics

# CFD Aids Oil Cooler Heat Exchanger Design

- Oil Coolers are geometrically very complex heaters used in residential and industrial settings
- The complex heat exchange surfaces and repeated passageways in such heat exchangers can be modeled very well with FloEFD's periodic boundary conditions and thin-wall technologies for small optimized computational grids
- FloEFD being integrated within the engineer's native CAD system permits unparalleled ease of use and computational speed in this example
- The design engineer can determine alternative design configurations for air-flow and heat transfer set-ups using 'what-if' scenarios.
- This helps the engineer to visualize and better understand the reasons for a particular temperature distribution, and why the results may be higher or lower than the allowed technical specifications.

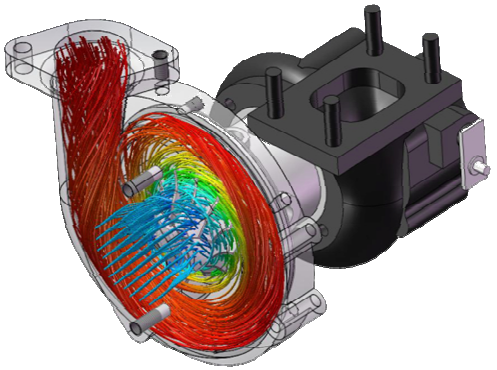


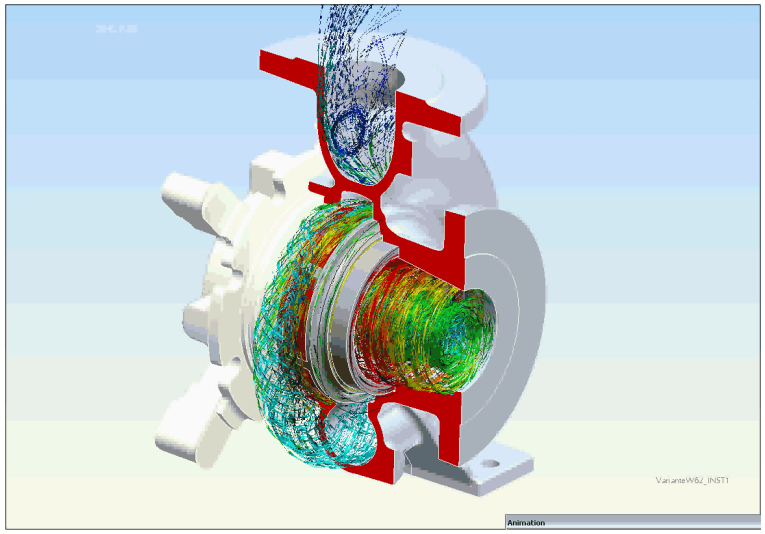


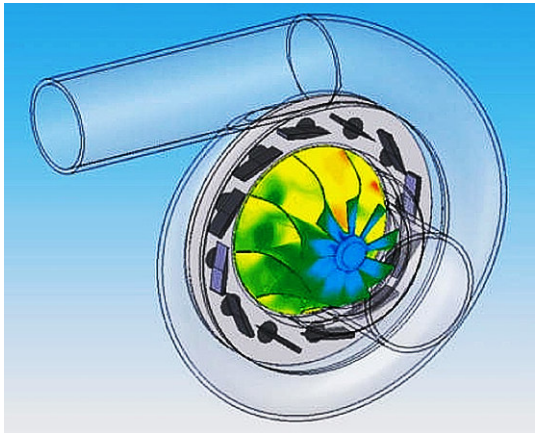
Mentor Graphics

# Pumps

- **Primary goal** - analyze and improve the efficiency of equipment (as delivered and in situ):
  - Intakes & exhaust flow patterns
  - Flows in pipes and ducts
  - Produce pump characteristic curves
  - Assess for the possibility of cavitation







# CFD Simulates Flow Through a Centrifugal Pump

- Centrifugal pumps for liquids can be difficult to model with traditional CFD tools
- FloEFD has a unique set of meshing and geometry modeling technologies that make this automotive pump easy to simulate
- Good agreement with experiment was achieved in this case and the immersed boundary mesh in FloEFD allowed for a small optimized mesh to be applied to this difficult engineering geometry.

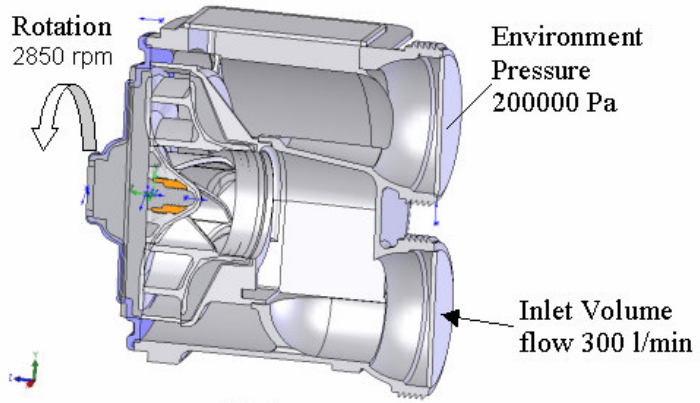
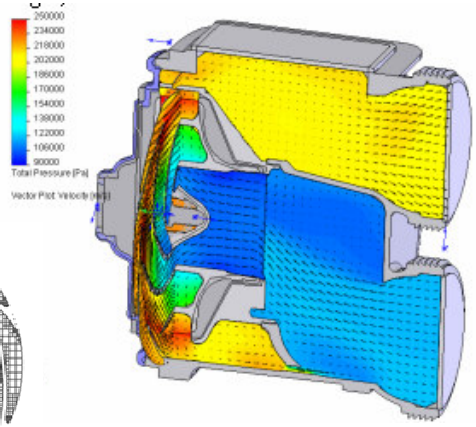
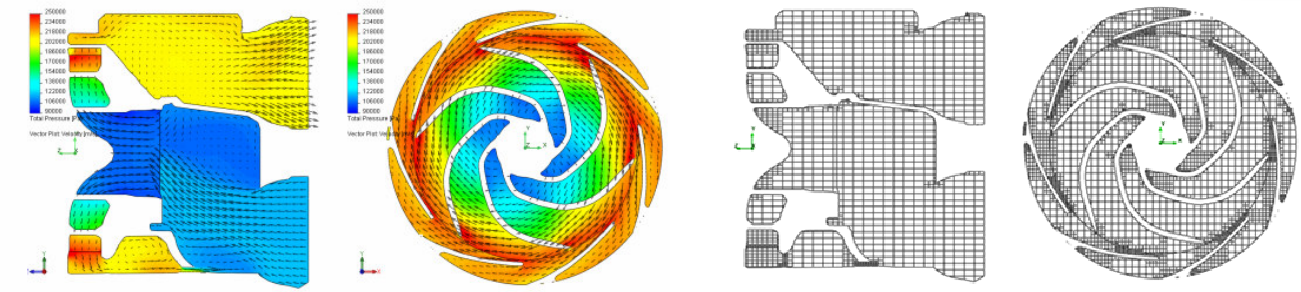


Fig.1

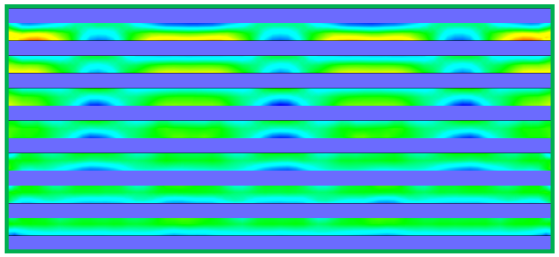




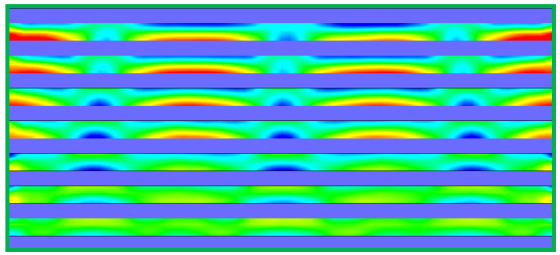
# The use of CFD to Optimize a Cold Storage Facility

- In the HVAC and Food Industries, cold storage facilities need to be designed to allow for uniform chilling whatever the number of trays being employed,
- In this example the design engineer examined several different tray configurations in FloEFD for the fan array being chosen before deciding on an optimum configuration.

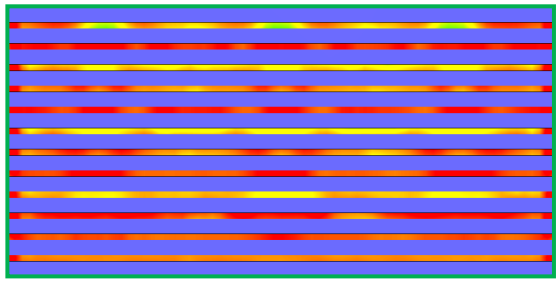
8 Trays



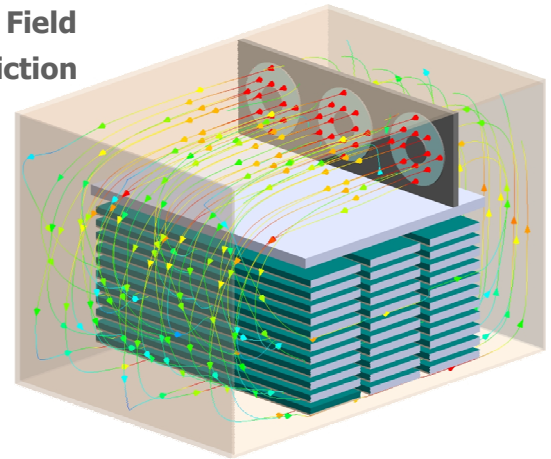
8 Trays with Ramp

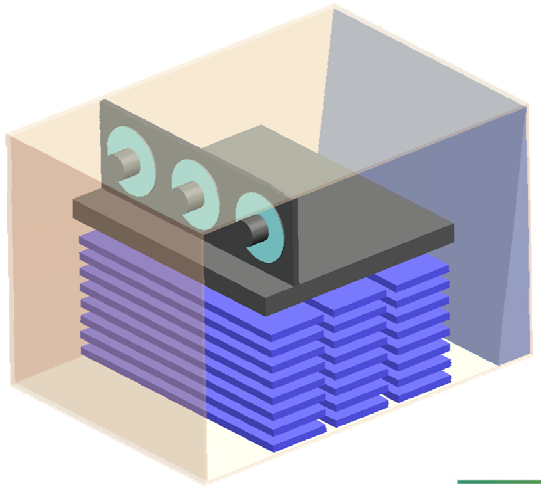


11 Trays

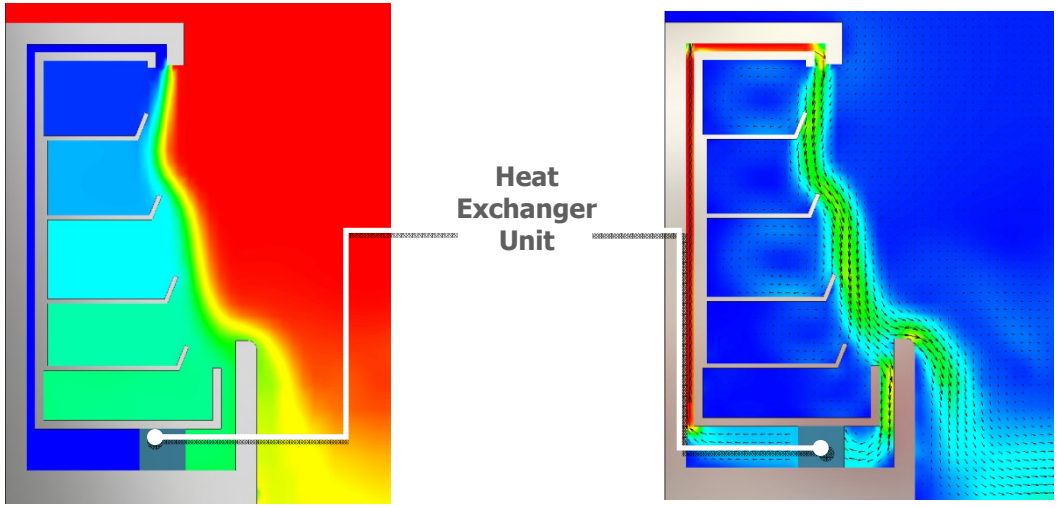


Flow Field Prediction





# Design of Refrigerated Self-Service Showcases Using CFD

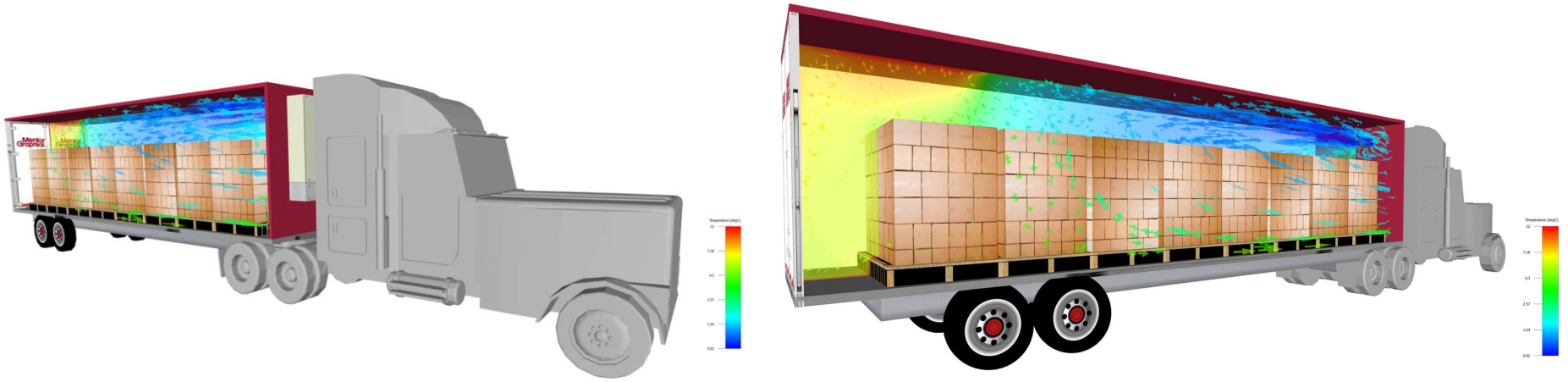




- In the Food & Beverage Industry display cabinets in supermarkets need to ensure a uniform showcase temperature when produce is on display.
- Fans that are integral to the Showcase design need to maintain a uniform thermal gradient both in front of and on the shelves of cabinets
- Design engineers use FloEFD to quickly create geometries of the refrigerated cabinets and then optimize the design for certain display characteristics
- Good agreement with experiment was found

Mentor Graphics

# Refrigerated Truck-Trailer Design Using CFD Modeling Techniques

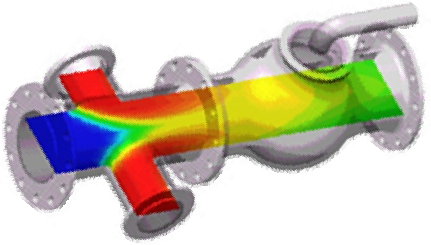


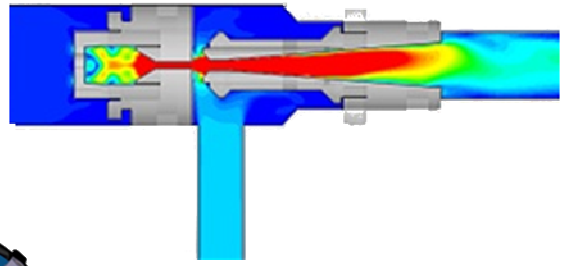
- In the trucking industry ensuring good ventilation and refrigeration in trailer arrangements is critical for the storage and transportation of goods
- FloVENT has been used in this example to model chilling flows in a refrigerated truck-trailer arrangement with various pallet arrangements in the trailer compartment

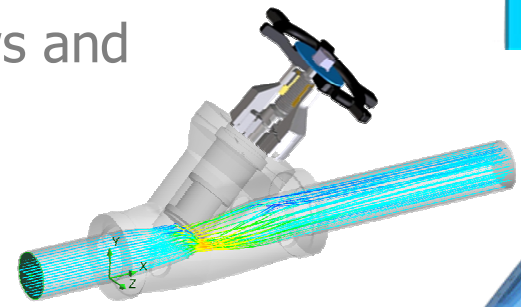
Mentor Graphics

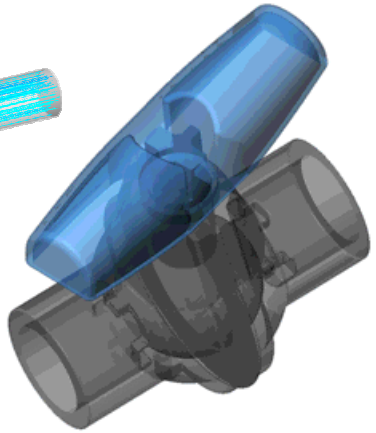
# Valves

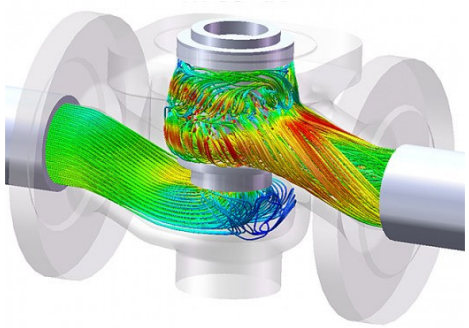
- **Primary goal** - analyze and improve the efficiency of equipment (as delivered and as installed):
  - Valve opening characteristic curves for pressure drop versus degree of opening
  - Visualize complex flow patterns
  - Identify high flow impact areas for wear, & forces on blades and components
  - Assess for internal shock flows and cavitation effects

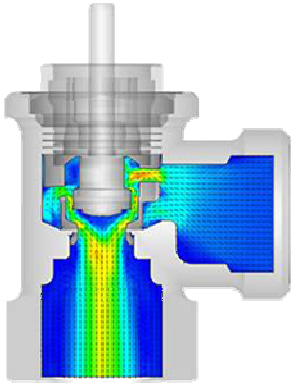






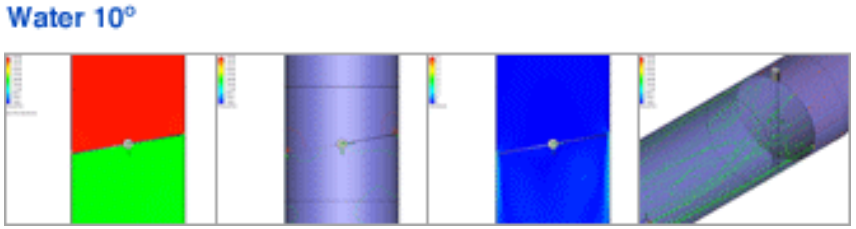


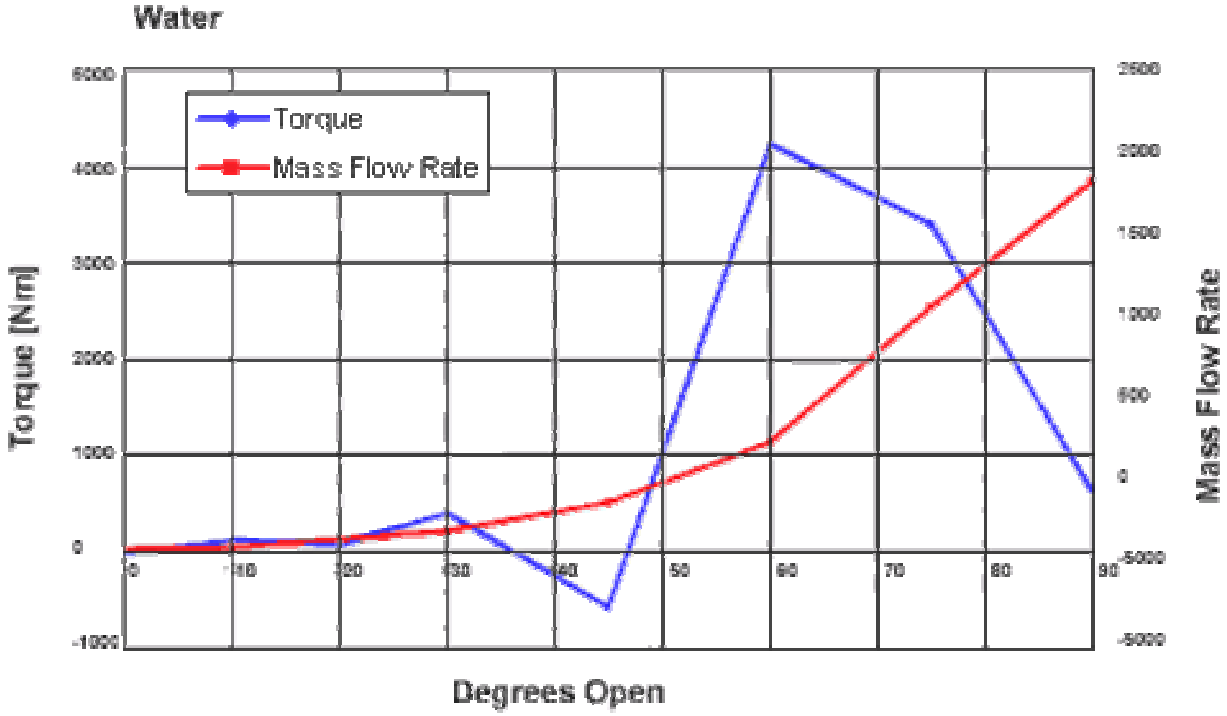




# Water Flow Butterfly Valve Validation Study with FloEFD

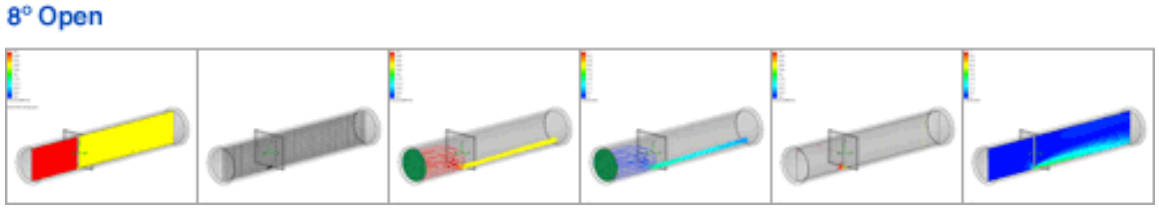
- CFD will yield valuable information on transient valve position characteristics as the valve in opened or closed
- This will help to understand performance once installed on site

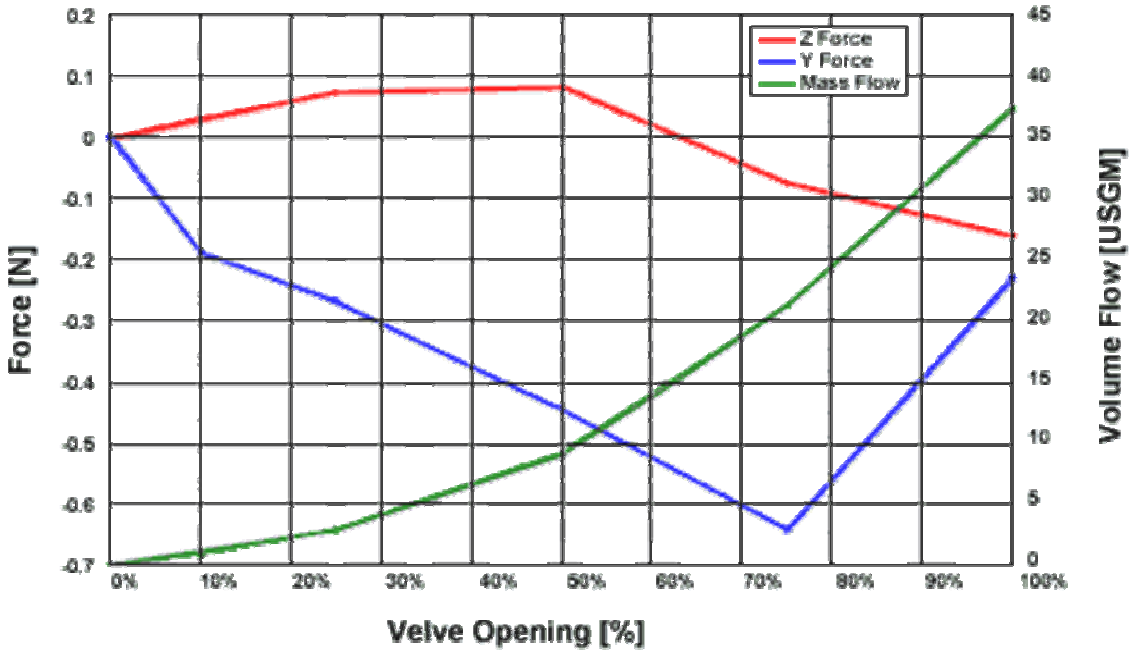


# Gate Valve
# Validation Study with FloEFD

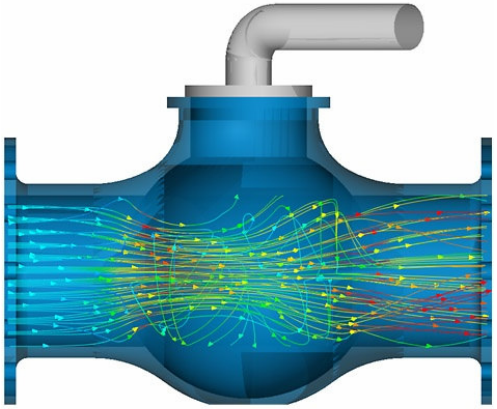
- CFD will help to characterize a valve system as installed in a pipeline
- Very important for non-traditional installations

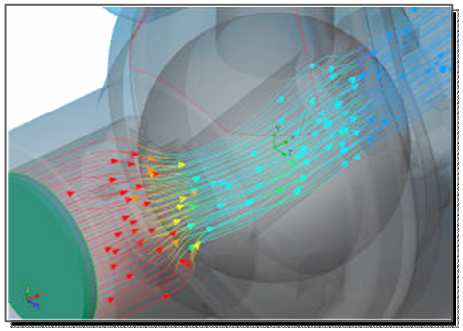


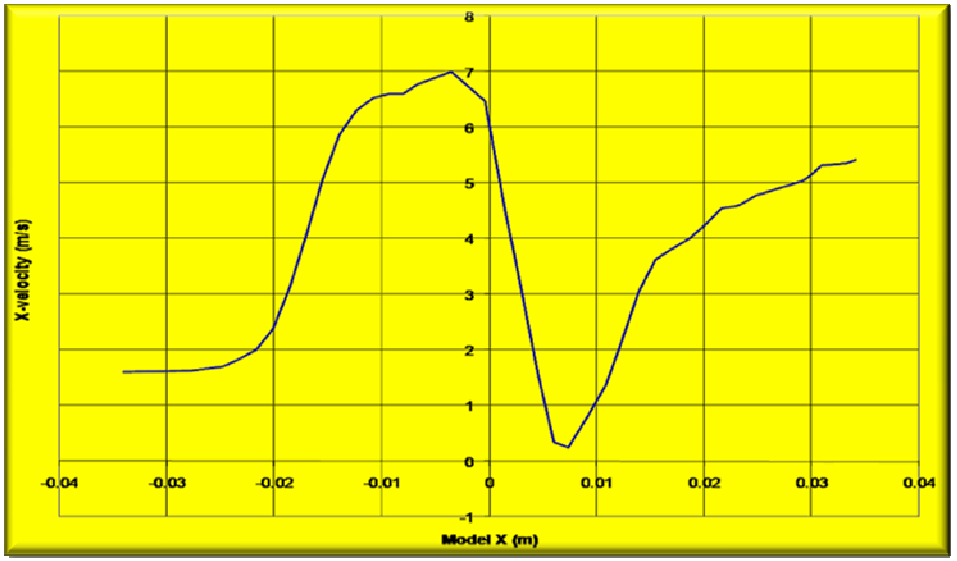



# The Characterization of a Classical Ball Valve Arrangement in a Pipeline

- In this example a new ball valve design was simulated in FloEFD over a wide range of valve openings
- CFD allows you to visualize water flowing through the valve body thus permitting a deep understanding and verification of various channel shapes for different pressure drop scenarios
- Precise X-Y co-ordinate graphical data for pressure drop along the axis of the valve in possible as is valve characteristic operating coefficients,
- This type of R&D is now possible for valve manufacturers to do without having to build and test prototypes.

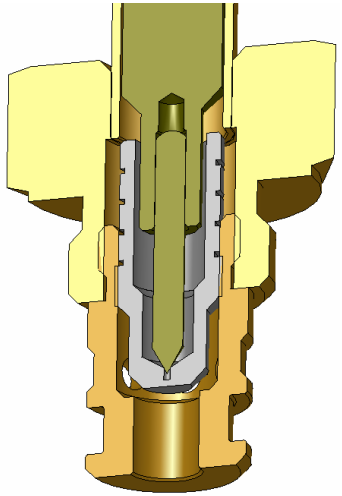


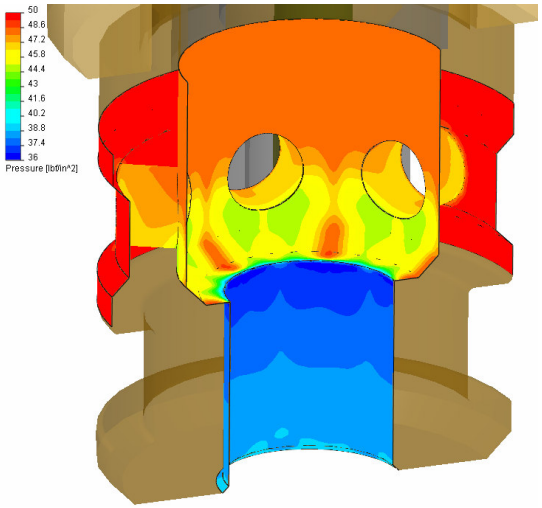


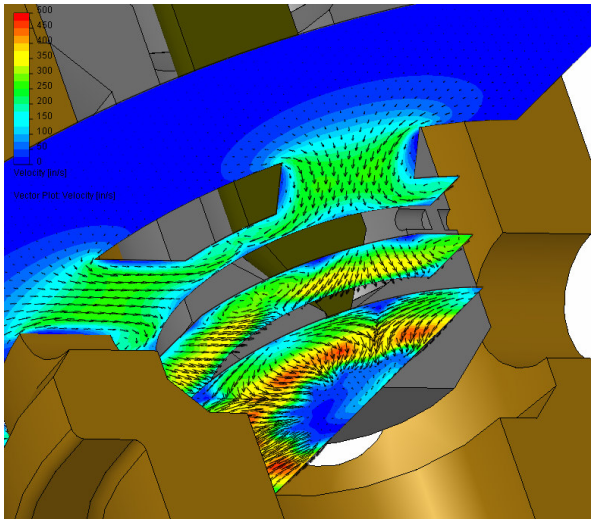



Mentor Graphics

# CFD Yields Detailed Flow Information for Flows Through a Poppet Valve






- In this example FloEFD was used to simulate pressure drops in a poppet valve assembly
- The complex cage with multiple holes was picked up accurately by the immersed boundary mesh and complex recirculating flows in the valve body were easily resolved