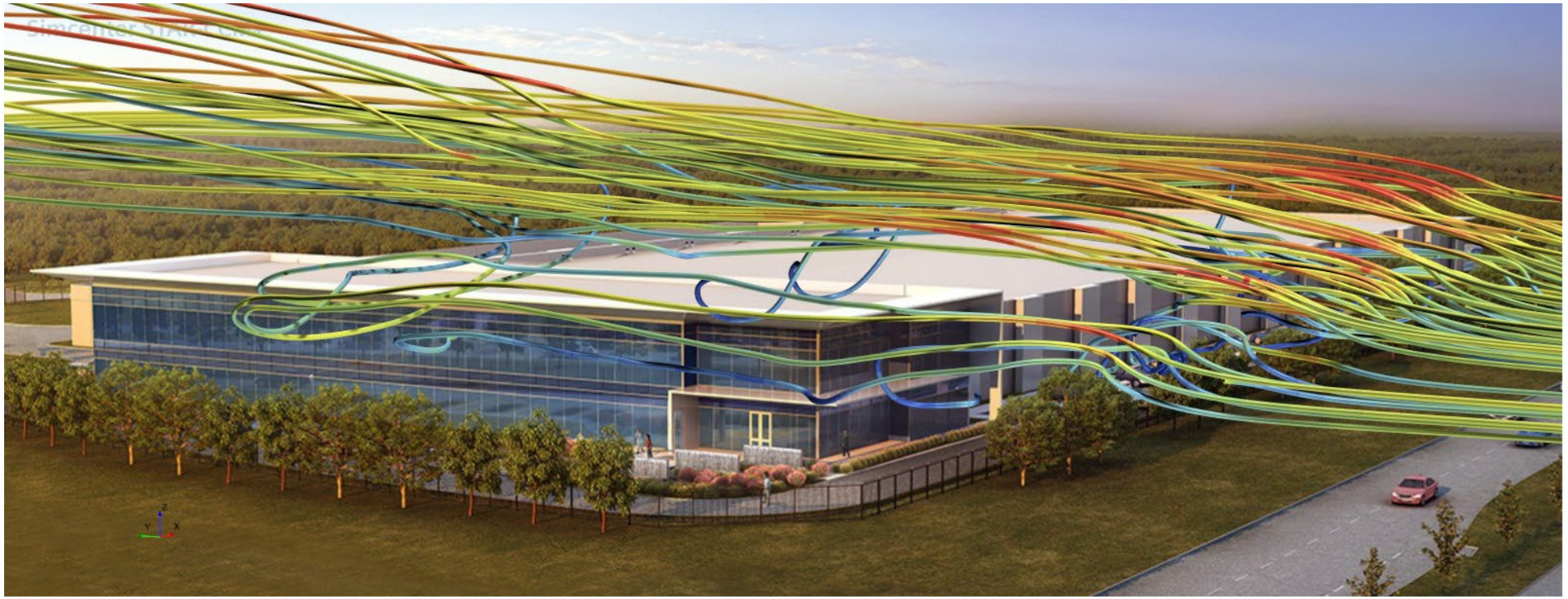


Computational Fluid Dynamics Analysis

CFD Engineering Services

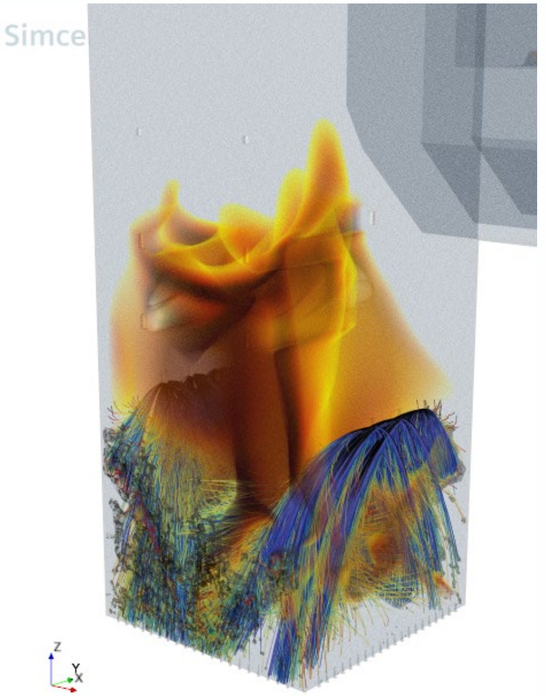
info@predictiveengineering.com
www.predictiveengineering.com
(503) 206-5571

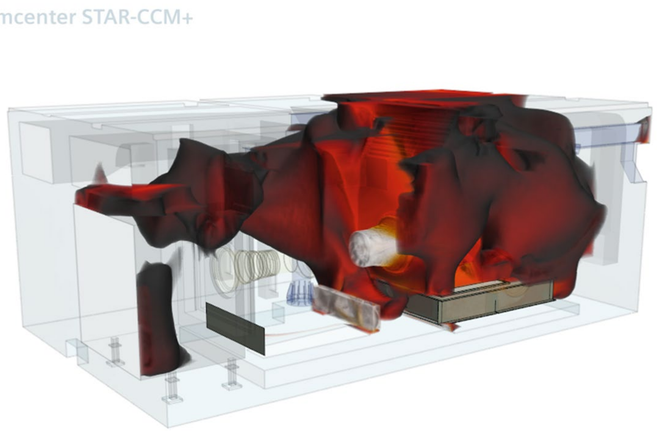
At Predictive Engineering, we developed our expertise in computational fluid dynamics (CFD) consulting with years of CFD project work in medical, aerospace, marine, HVAC, civil and automotive. Our work has been extensively benchmarked by experiments and in-service testing.

Our portfolio of case studies provides hard evidence of our many successful CFD consulting projects: space-based communications equipment, bio-fuel combustion, aerodynamic drag, hydroelectric spillways, dispersed particle flow, data centers, power electronics and HVAC air handling systems, to name just a few. Besides these graphical writeups we also have a searchable text listing of our projects.

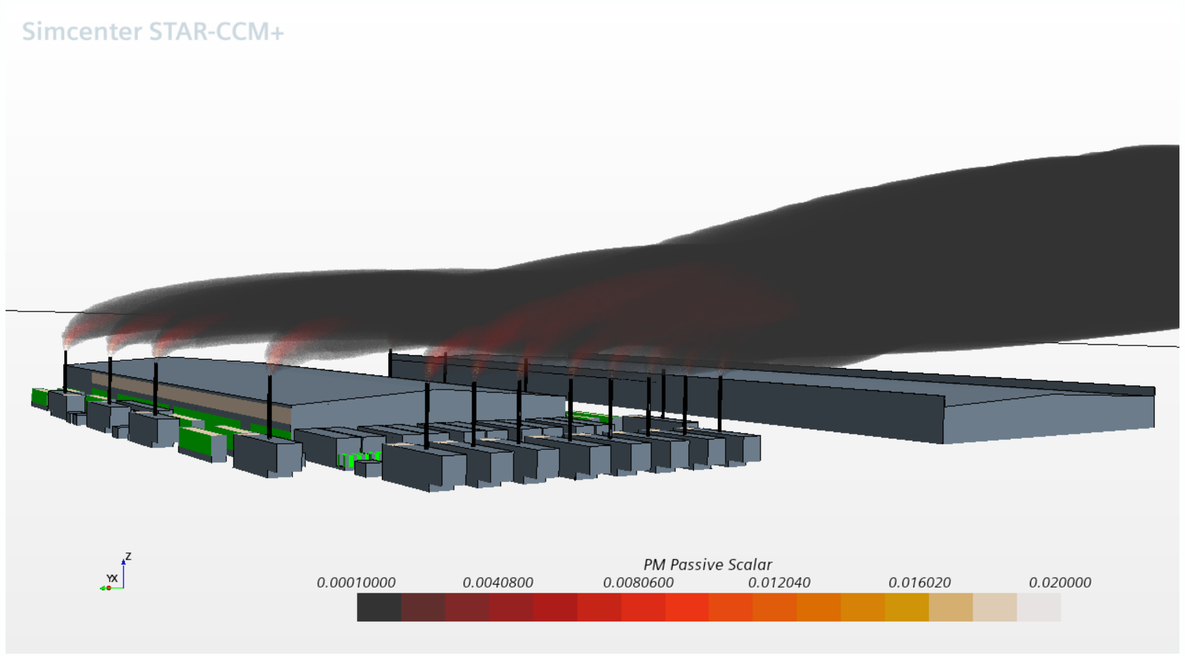
In CFD and really, all types of engineering simulation, experience is the key in knowing whether one has a cartoon or a true digital twin that can be taken from design into production. We feel that this is our competitive edge and our proven value to our clients.

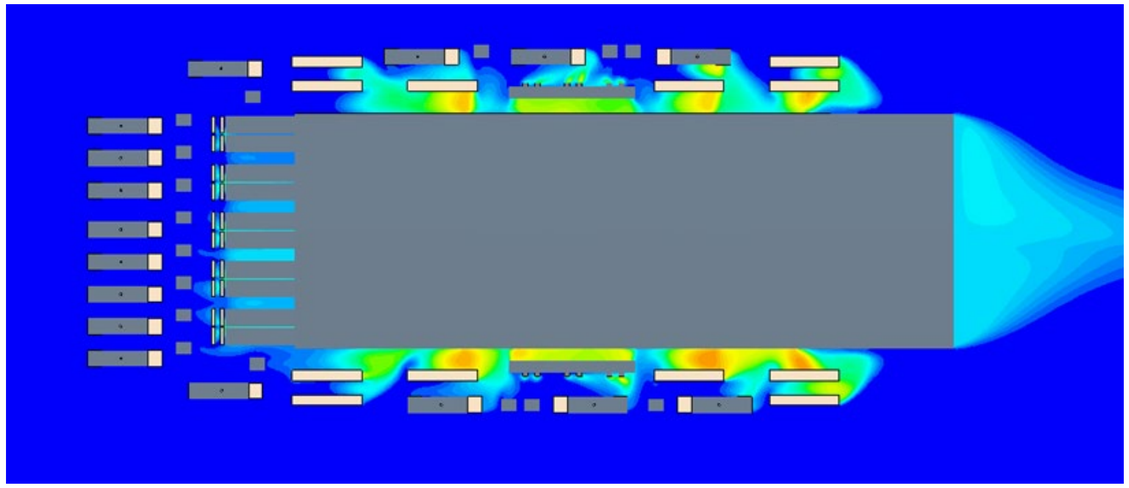
If you have a need for experienced CFD consultants, Predictive Engineering has continually expanded its client base since 1995 with more than 800+ projects, so please feel free to contact us and learn more about our CFD consulting services.





Data Center Exterior Exhaust Flow Analysis

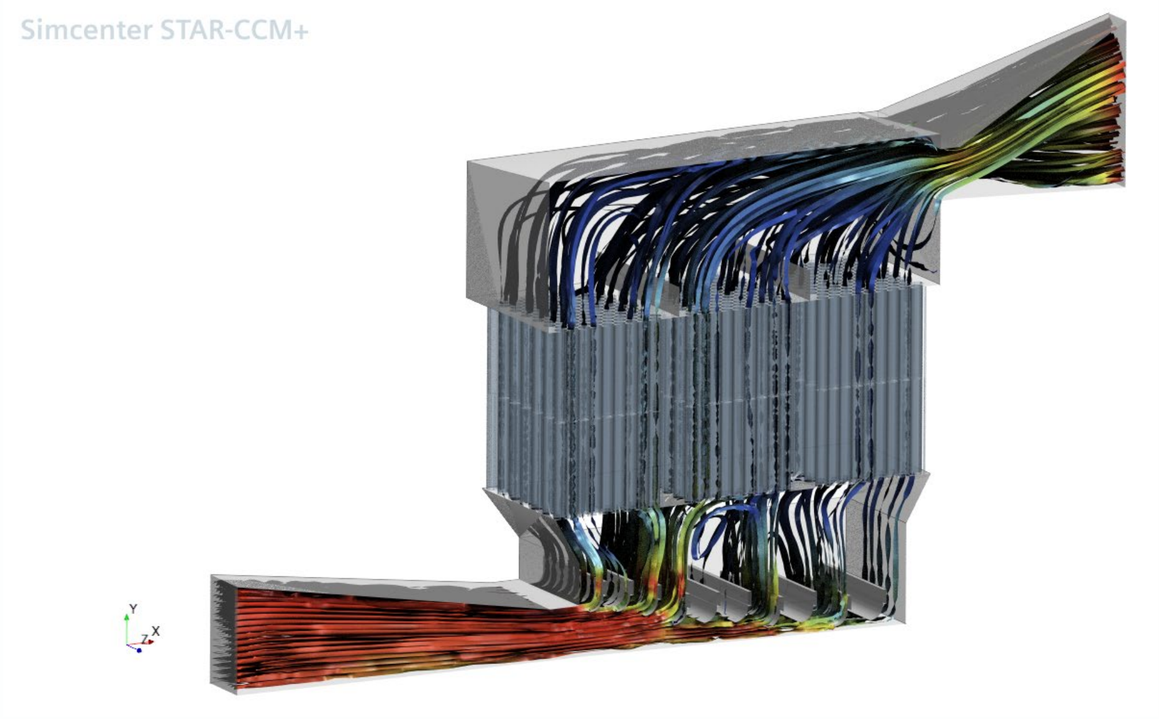


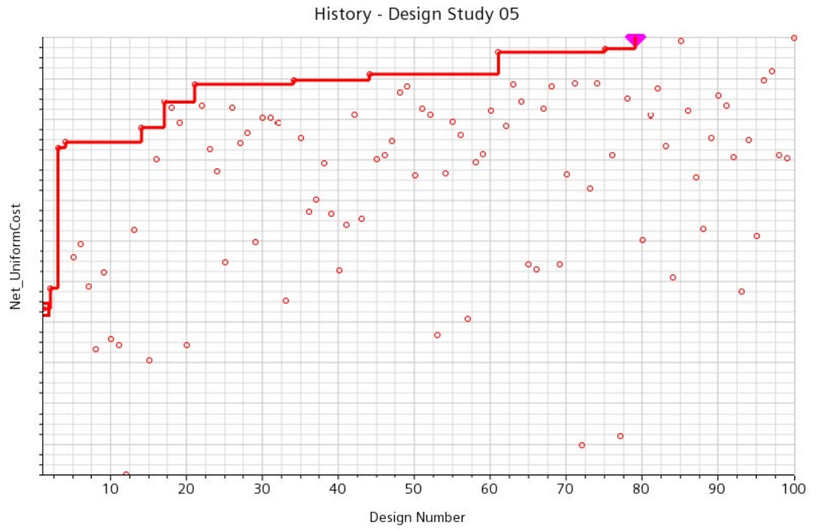


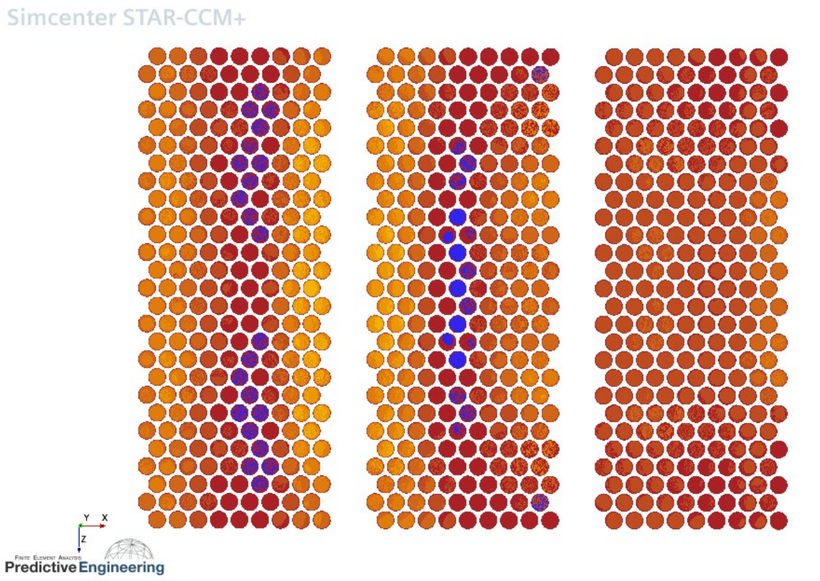
Predictive Engineering has done numerous exterior flow analyses for data centers which are powered by onsite diesel generators. The analyses are performed at different wind conditions to ensure that exhaust and heat from the diesel generators do not contaminate the supply air of cooling equipment into the data center of adjacent buildings.

Optimization of Flow Distribution in Industrial Precipitator

A recent study evaluated optimizing flow through a large industrial precipitator. The structure consisted of three tube banks, where the incoming flow must turn up and out of the structure. For peak operational performance, uniform flow is desired through the three tube banks. Vanes at the bottom of the precipitator turn incoming flow up and through the tube banks. The optimization study evaluating positioning of the vanes and number of required vanes to achieve best performance.

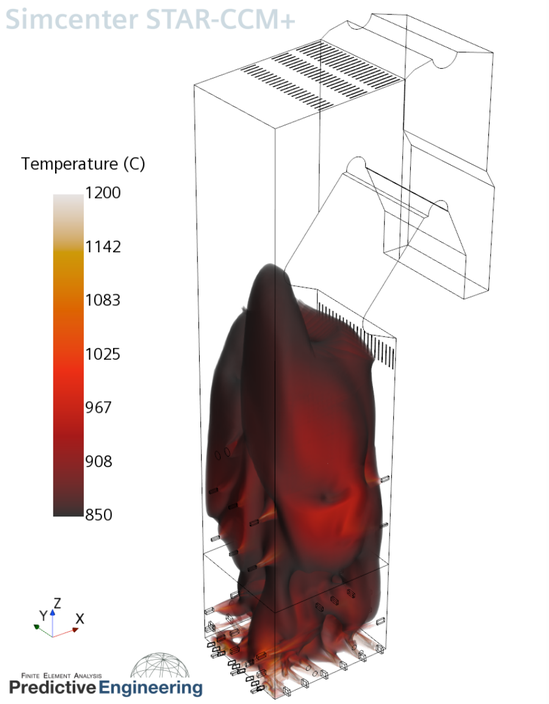


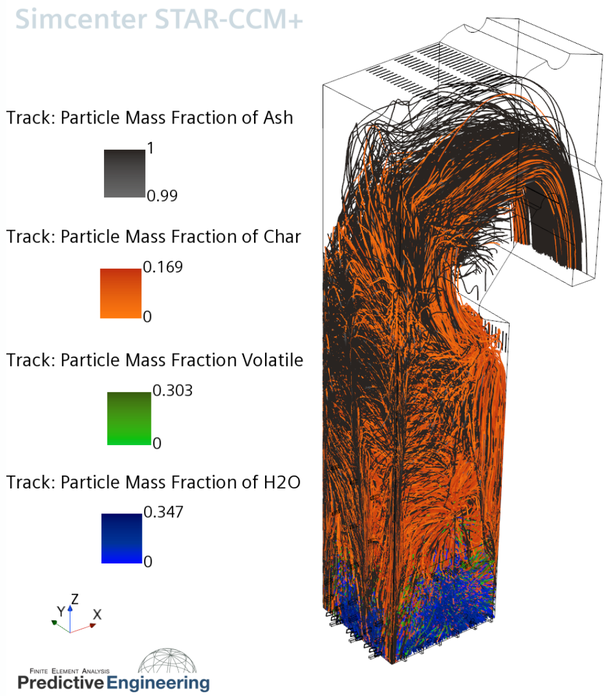


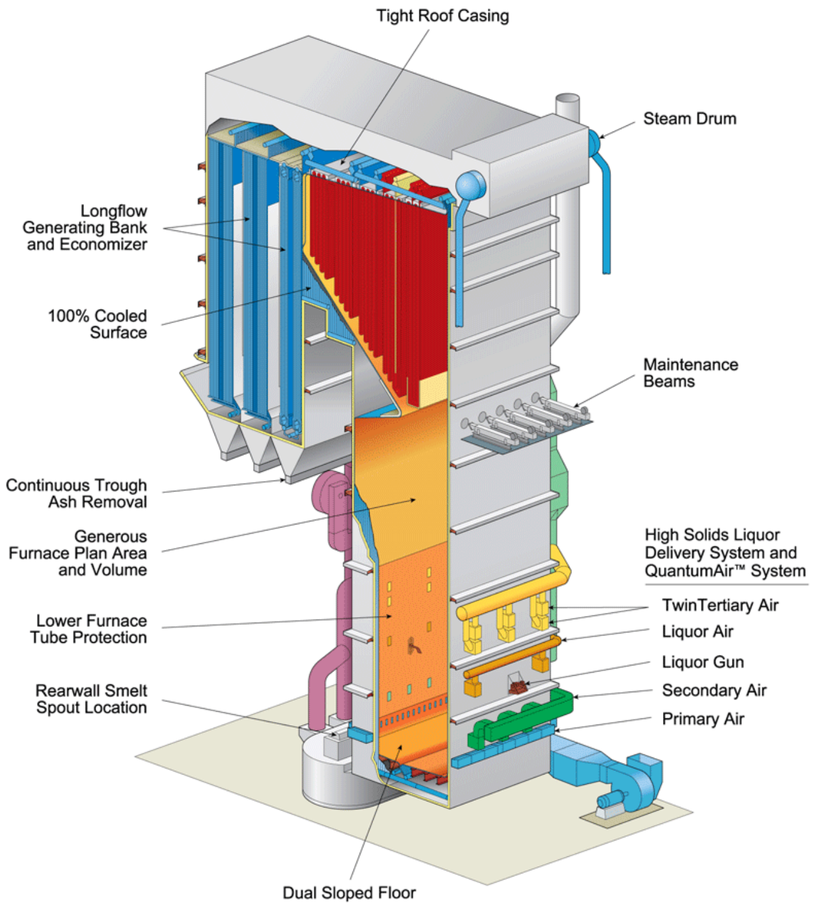


CFD Simulation of Boiler Combustion

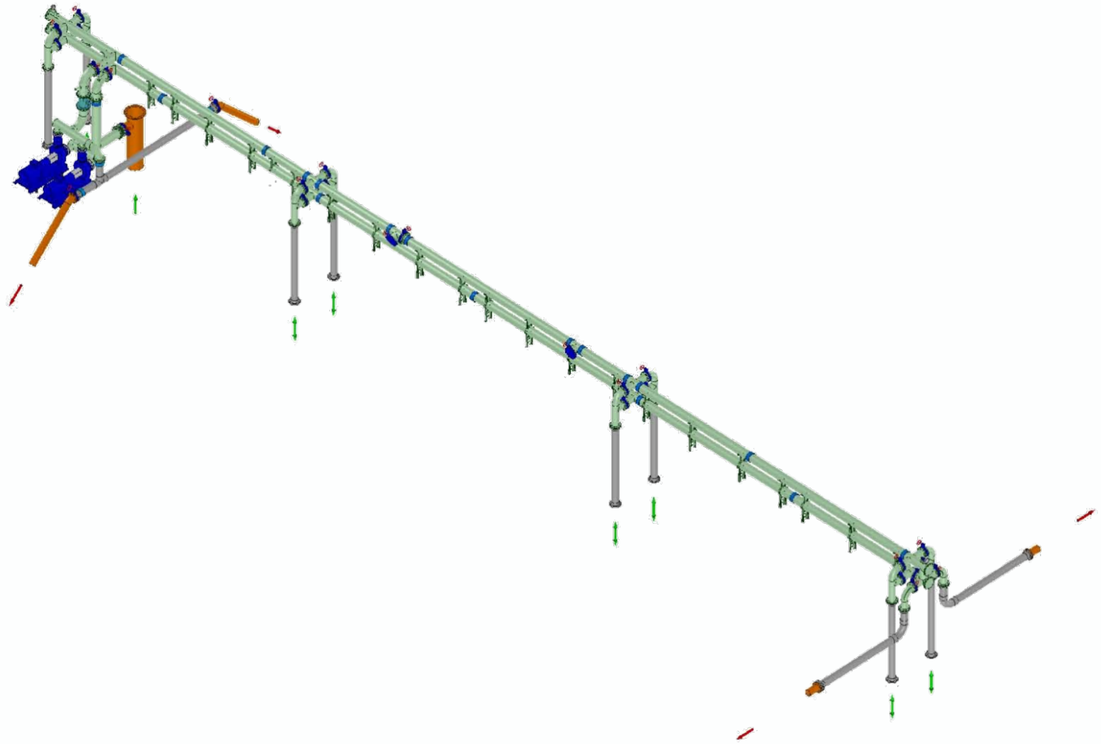
We have supported a local company who retrofits recovery and power boilers in the paper industry with CFD modeling. The goal is to assess ways to improve furnace performance and reduce emissions. These models are full mutli-physics simulations and very challenging to model, as they include solid fuel combustion, radiation, and heat transfer. The fuel is modeled with the Lagrangian phase coal combustion model in STAR-CCM+, which captures drying, devolatilization, and char burn out.

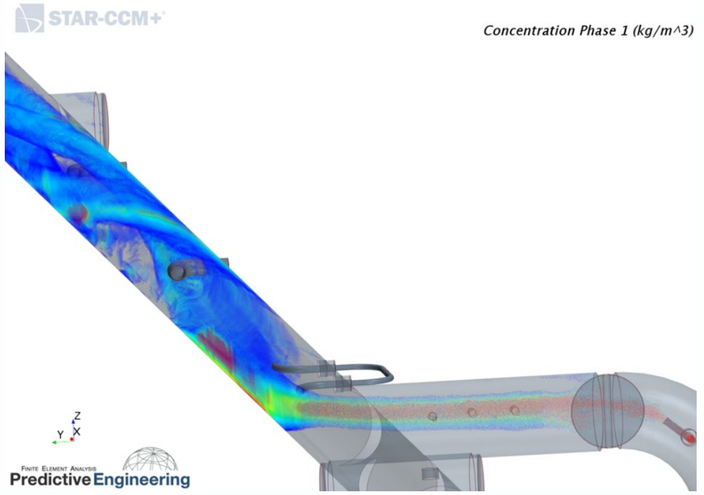






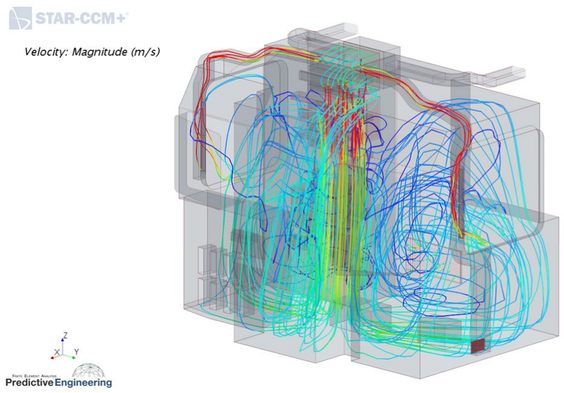
Sediment Flow within Marine Dredging System

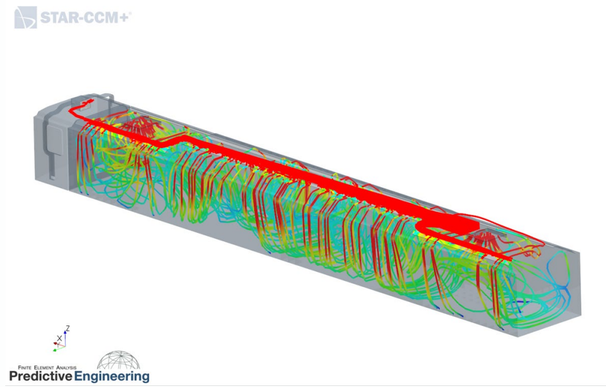


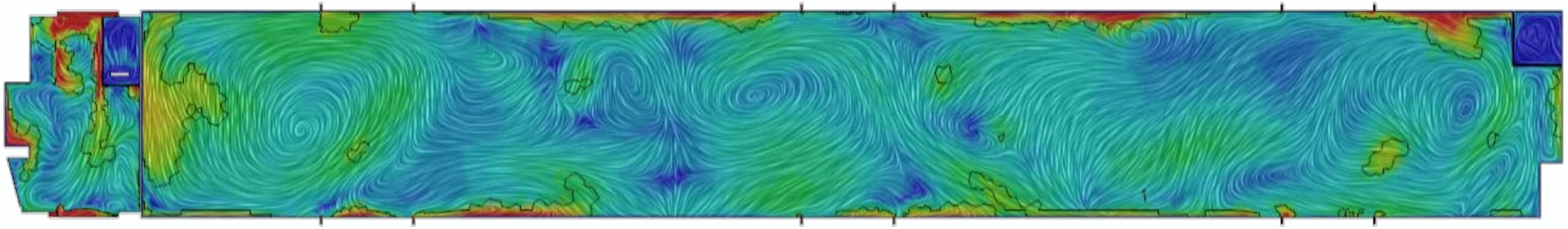


In this application we used Lagrangian multiphase models to simulate fine silt particles with sizes on the order of microns in a seawater dredging system. The Lagrangian models predict the path of the particles based on the drag and lift forces induced from the surrounding fluid. The fluid makes two 90° bends through a pipe coupling and a sharp bend through a tee, which causes a significant amount of swirl in the fluid. Our analysis showed much of the injected sediment was getting caught up in this swirl and pushed to the outside of the pipe walls. The results were able to clarify measurement observations the end-client had from test.

HVAC CFD Analysis of Light Rail Transit Car



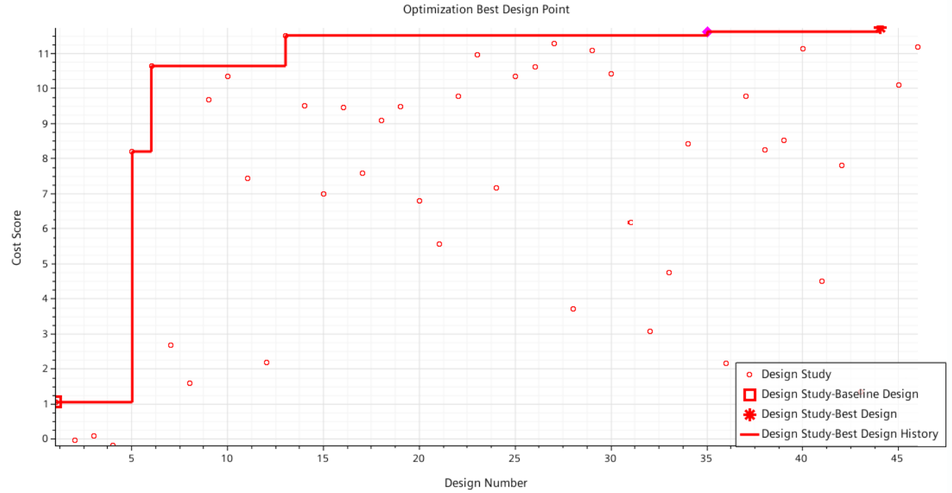


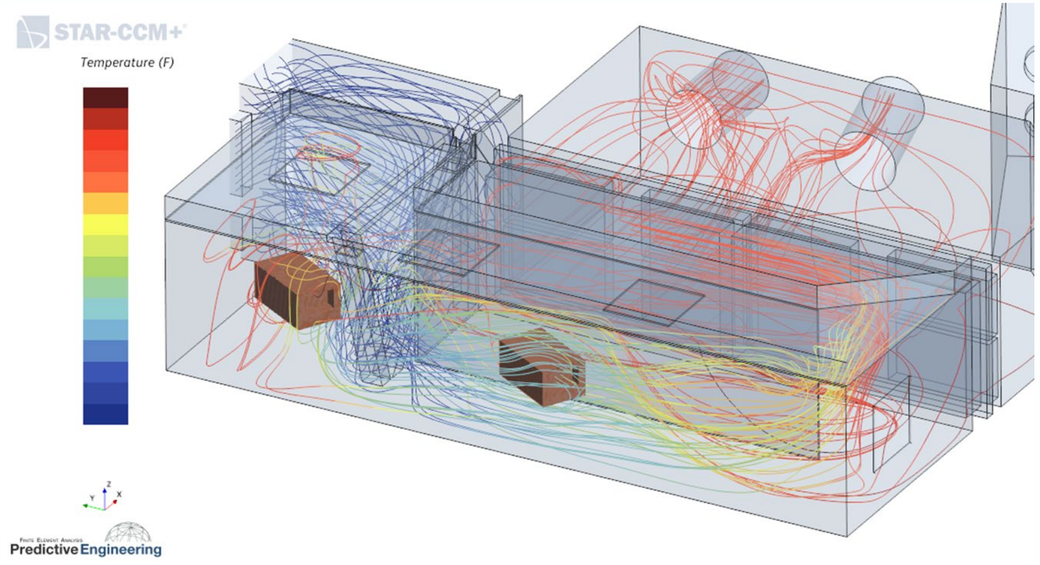


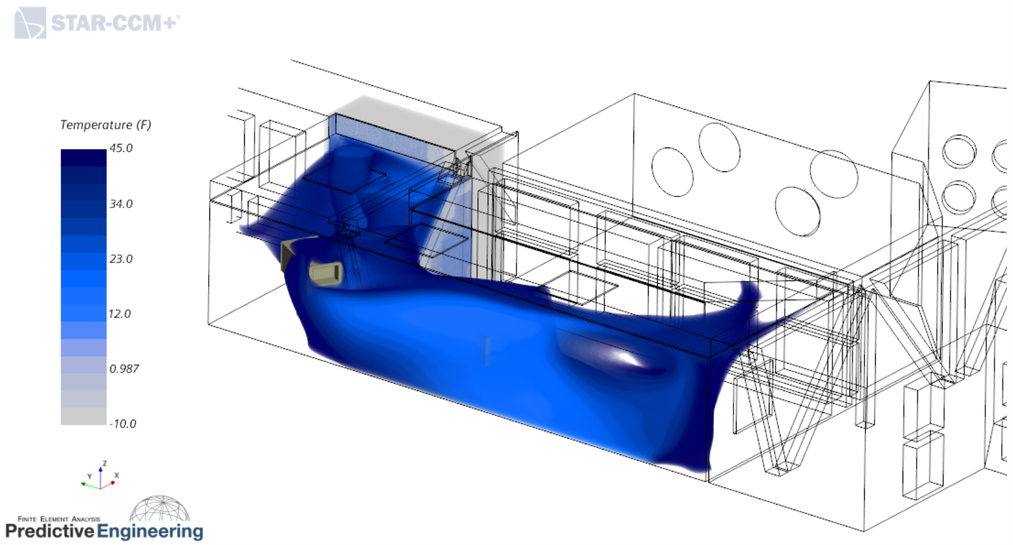
Evaluation of thermal and flow uniformity analysis for a subway car was performed to ensure the design met end customer specifications. The thermal loads included sensible and latent heat generation from full passenger load, electronic loads from three different cabinets within the vehicle, exterior radiation loads, and flow leakage out of the door gaps. Predictive's CFD consultants provided design input ensure that uniform flow could be achieved from the main overhead plenum, as well as even flow distribution in the front driver cab. Experimental measurements aligned well with our CFD results.

Optimization of Air Mixing in Office Tower Fan Room

Thermal optimization CFD analysis was performed for fan room designs in a new multistory office tower. The supply air from these rooms is made up of approximately 30% outside air mixed with return air. For very cold winter days this can result in a temperature difference of 48°C . Design optimization tools within STAR-CCM+ were used to automate the simulation to determine the appropriate placement for heaters and damper settings to provide the best air mixing between cold and return air.

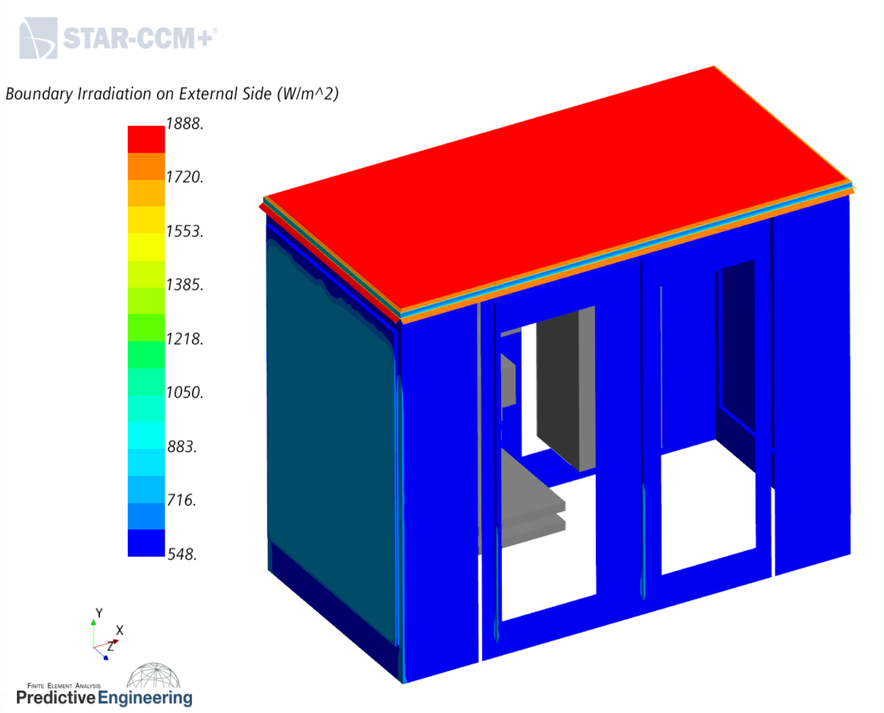


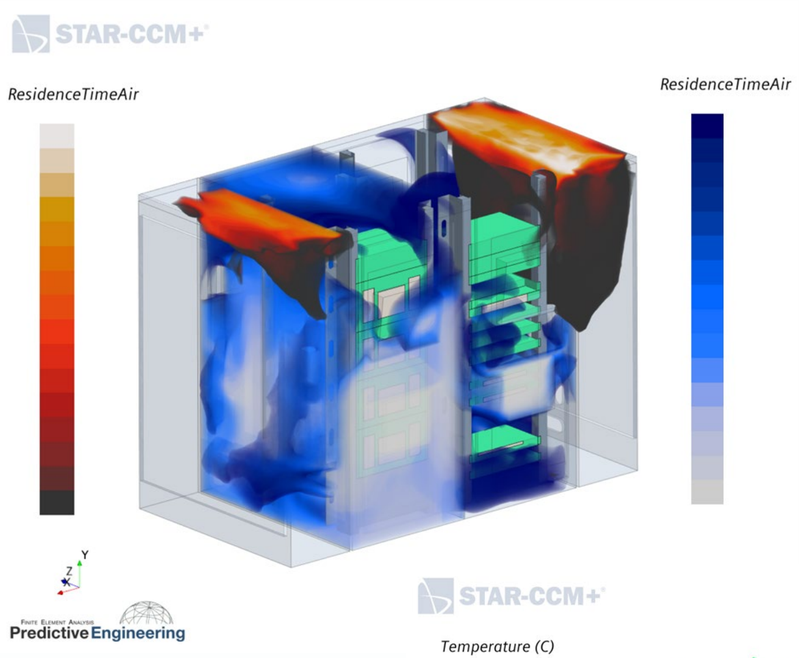


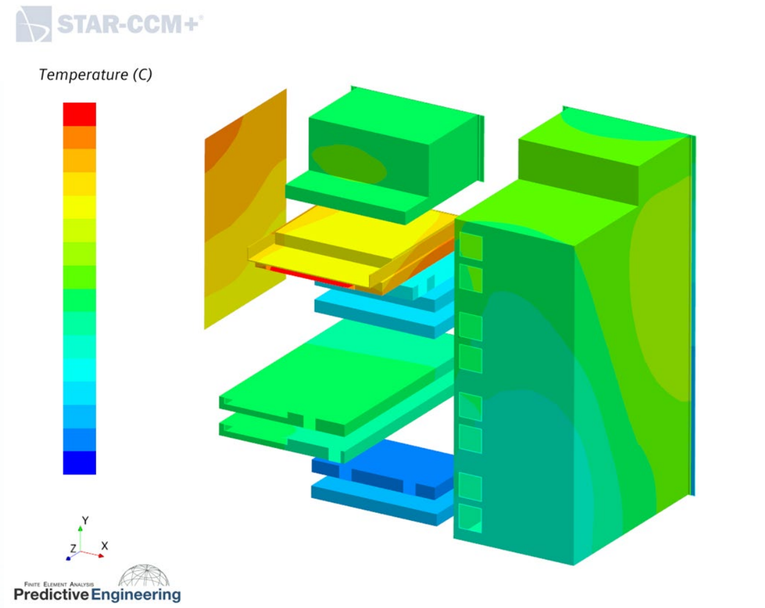


Electronics Cabinet with Incident Solar Loads

Electronic cabinets for exterior applications must be able to survive and operated in extreme environmental conditions. Predictive Engineering provided services to analyze air flow and temperature distribution within a UPS cabinet for telecommunications. HVAC units provided cooling, but the insulated walls are relatively thin compared to typical building applications, so external heat gain from environmental temperatures and solar radiation was a driver. We worked with the end customer to provide suggestions for improving air flow within the unit to reduce residence time and temperatures for the internal electronic devices.

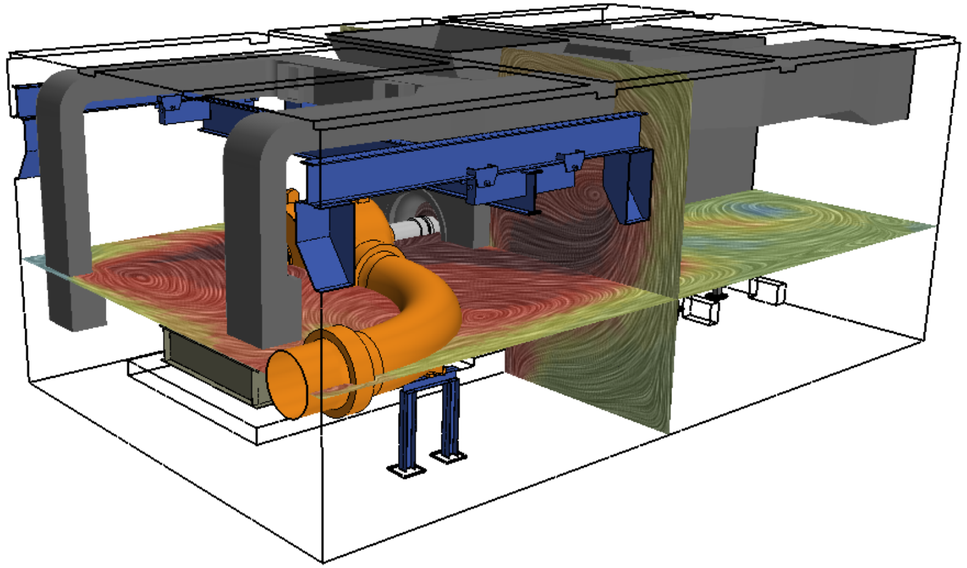


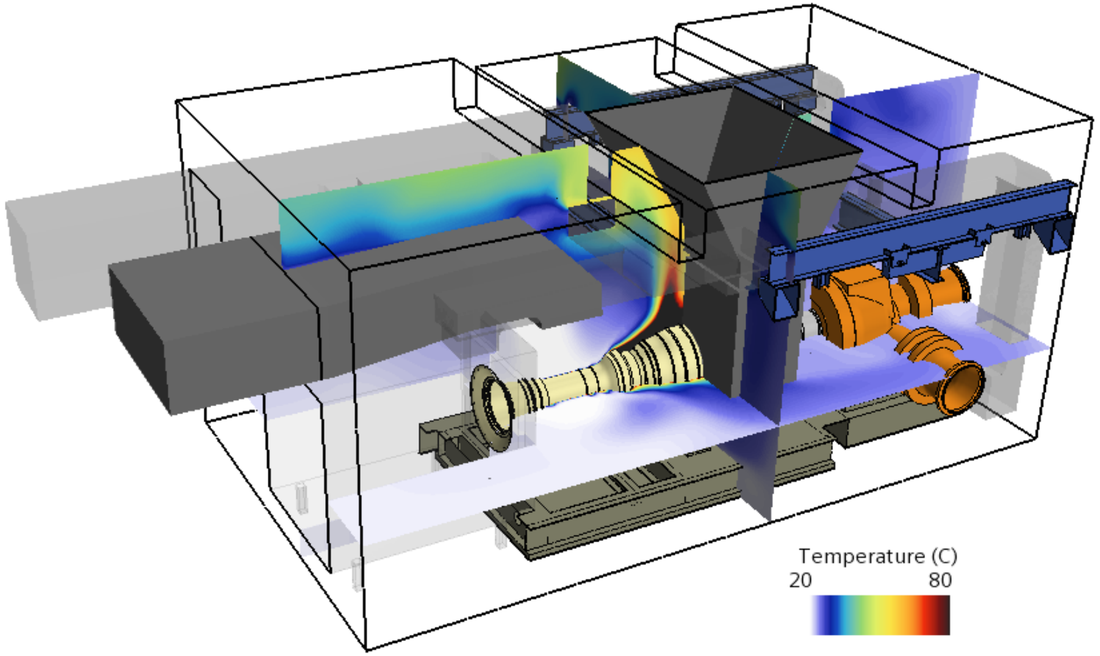




Turbine Enclosure Flow Analysis

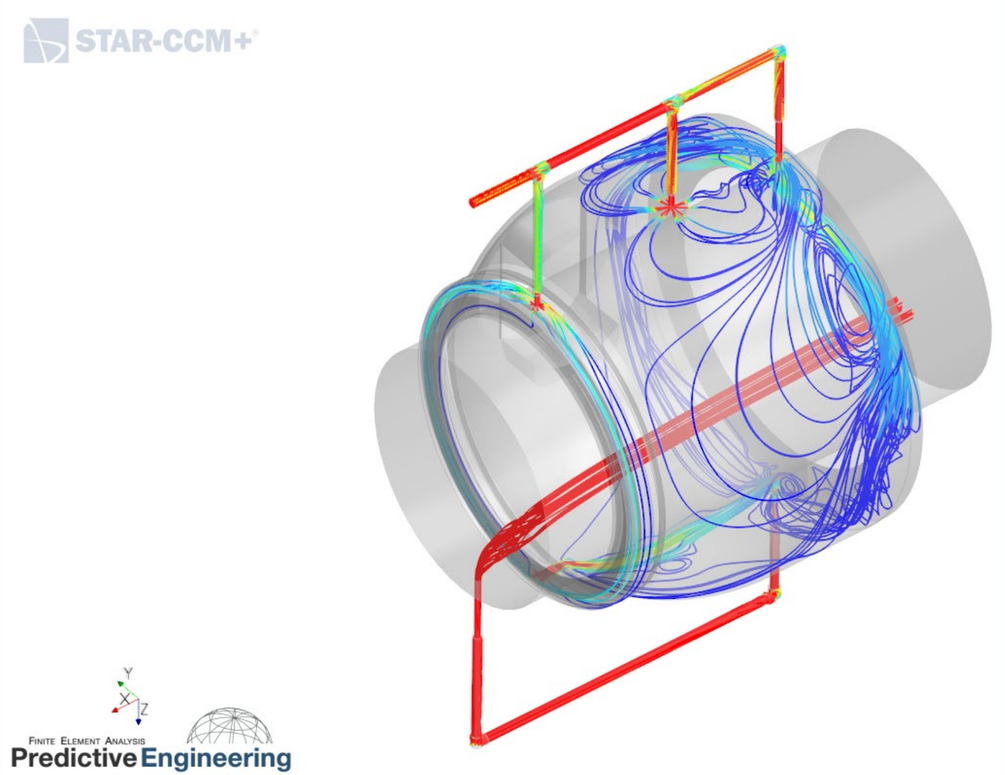
Gas leaks within enclosures for natural gas turbines can lead to catastrophic situations. Gas leak and fire detection systems are critical for reducing risk of fire or explosions. Predictive Engineering performed CFD analysis of two such turbine enclosure designs to identify regions where significant concentrations of combustible gases could accumulate. The air handling systems are design to pull air out throughout the enclosures, but the tight spaces within the room and waste heat from the turbine can induce local vortices that trap air flow.

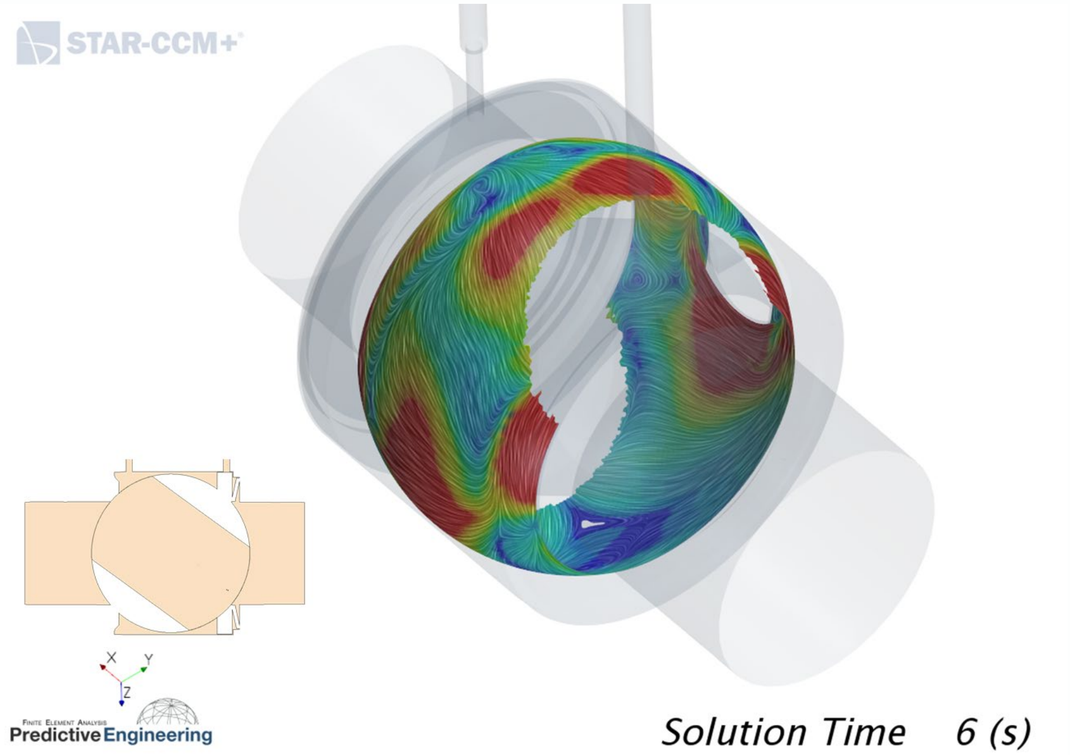




Purging Flow Industrial Ball Valve

Industrial ball valves require purging systems to ensure particulates do not interfere with sealing surfaces and overall valve operation. CFD analyses were performed on two such ball valves, up to 36 in. in diameter that operated at elevated temperatures with viscous hydraulic oil. Transient analysis was performed for one of the designs where purging is engaged during the transition process.





Propane Gas Vaporizer

Vaporizers are used for propane and other gas products to increase the flow rate out of the tank in cold storage conditions. Liquid propane is run through a heating jacket to vaporize as a usable gas at the outlet. CFD analyses were performed to evaluate a current design that was having issues at very low temperatures and provide design guidance for reducing pressure drops and increase overall flow rate. The analysis utilized wall boiling models to capture the heat transfer and phase transition of the fuel.