

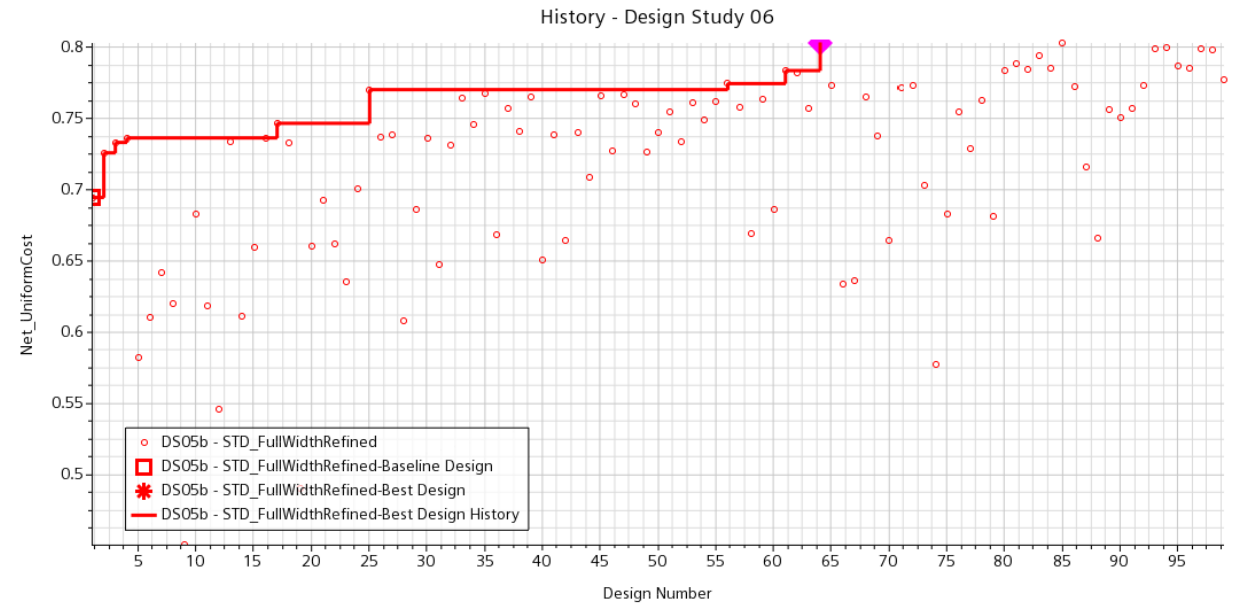
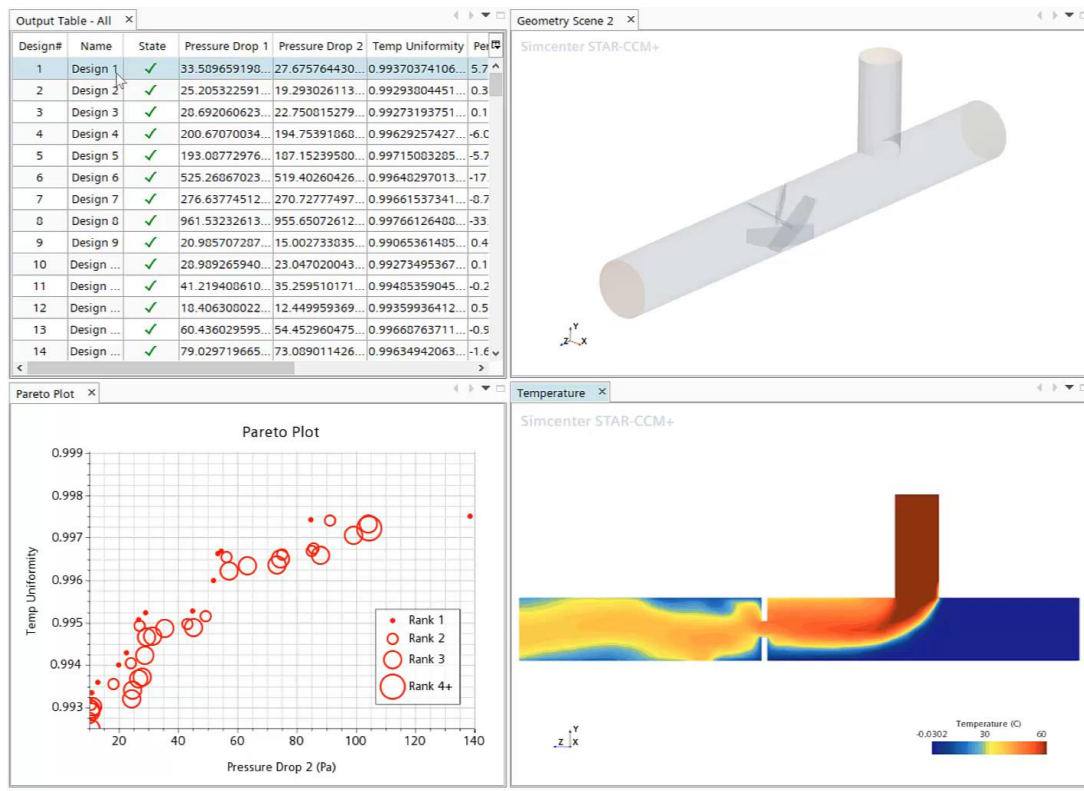
# CFD Design and Optimization Services

## Introduction

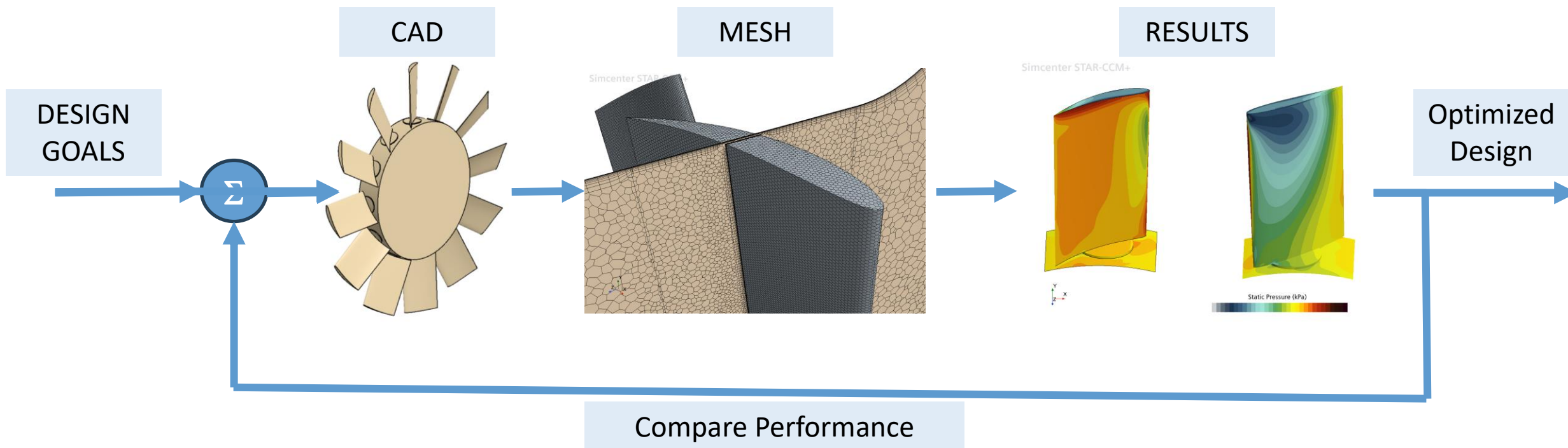
Computational fluid dynamics is a powerful numerical tool in airflow modelling for accurately predicting flow, pressure, and heat transfer. However, its true value lies in leveraging these predictions to guide design decisions for airflow optimization. This is one of the driving reasons we use Siemens Simcenter STAR-CCM+ for all our CFD analysis tasks.

In addition to its extensive physics modelling capabilities and advanced automation features, Simcenter STAR-CCM+ enables seamless integration with your design geometry. With this streamline procedure, we can help you realize your optimal design quicker, cheaper, and avoid potential failures in the future.

In this study, we will outline the general CFD design optimization process and showcase real-world examples where we have successfully applied these techniques to benefit our clients.



## General Optimization Scheme



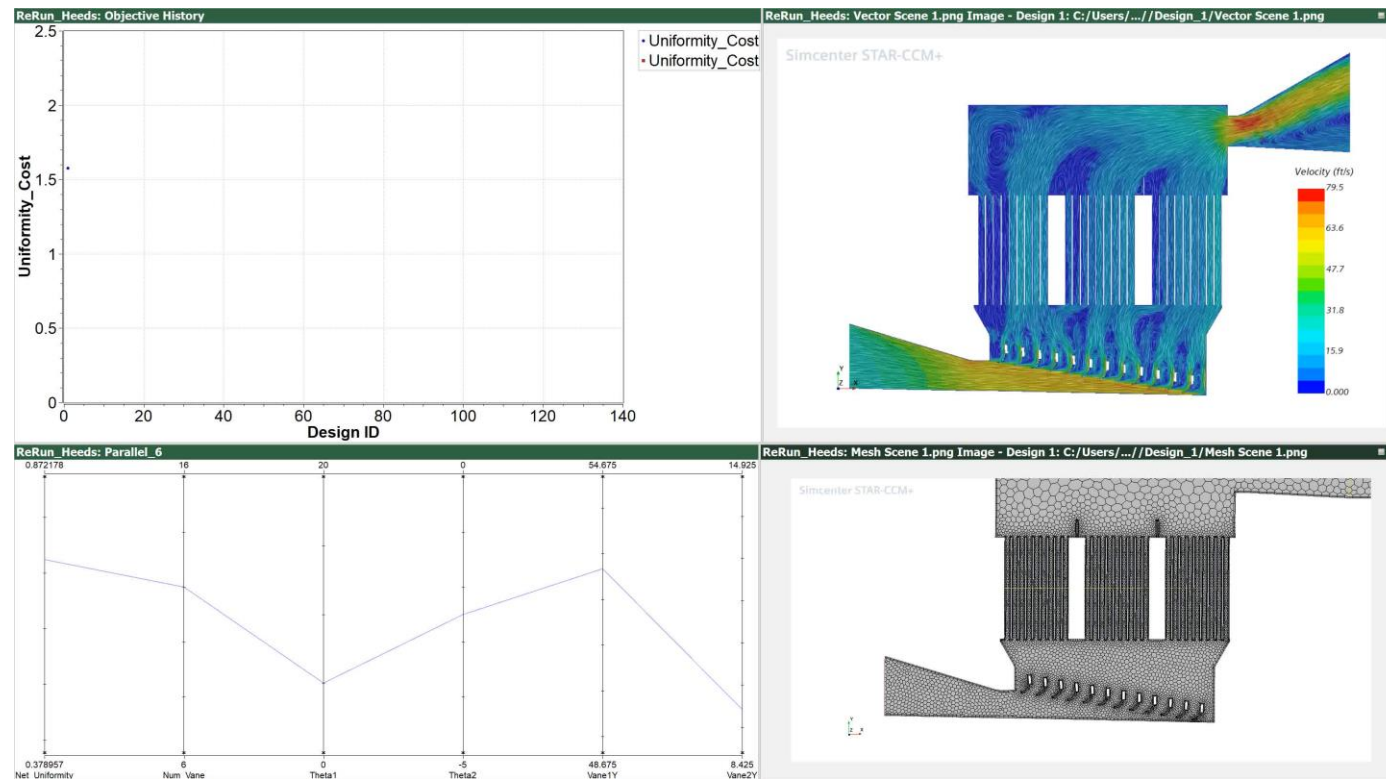
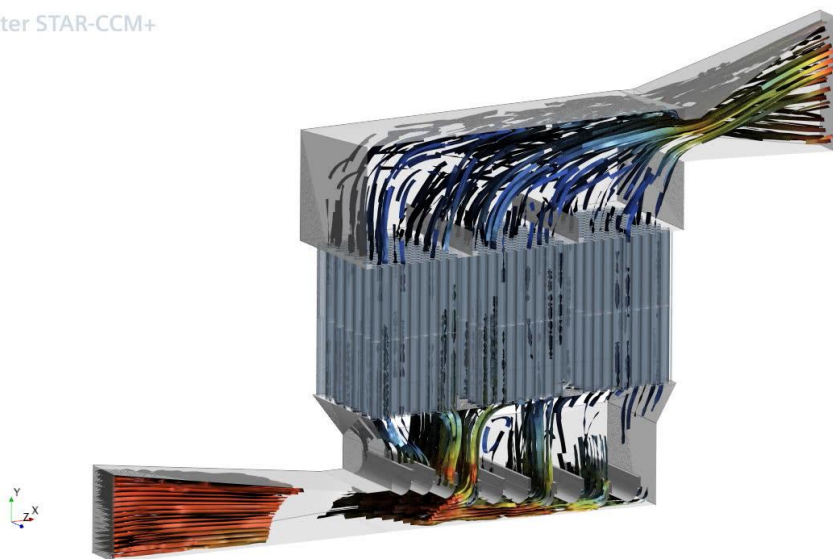
Every engineering projects begins with a set of design goals. From these goals the engineering team develops an initial design with CAD geometry to meet the requirements. For a CFD analysis, we will take this CAD geometry, define the flow domain, boundary conditions, and physics to create a finite volume mesh and analyze the performance.

This process provides engineering results, such as temperatures, pressure drops, flow velocities, or any other key metrics, which we use to compare to our original design goals. At this stage, the workflow resembles a control loop. By leveraging an automation process for adjusting parameterized CAD geometry, re-meshing, and re-analysis, we can streamline this analysis procedure.

With the built-in optimization tools in STAR-CCM+, we can efficiently iterate through this loop. The software evaluates hundreds of design iterations, significantly reducing engineering time and eliminating the need for manual trial-and-error, ultimately delivering a final, optimized design in a fraction of the time and cost.

## Baffle Placement Optimization for Industrial Precipitator

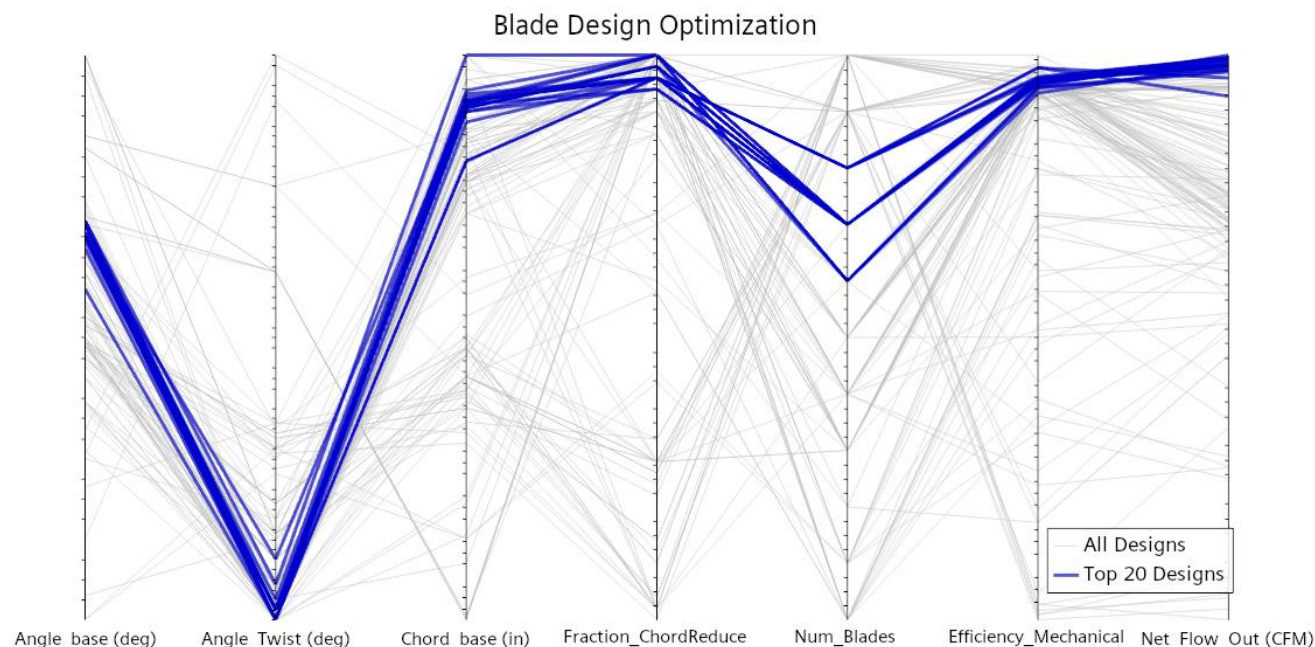
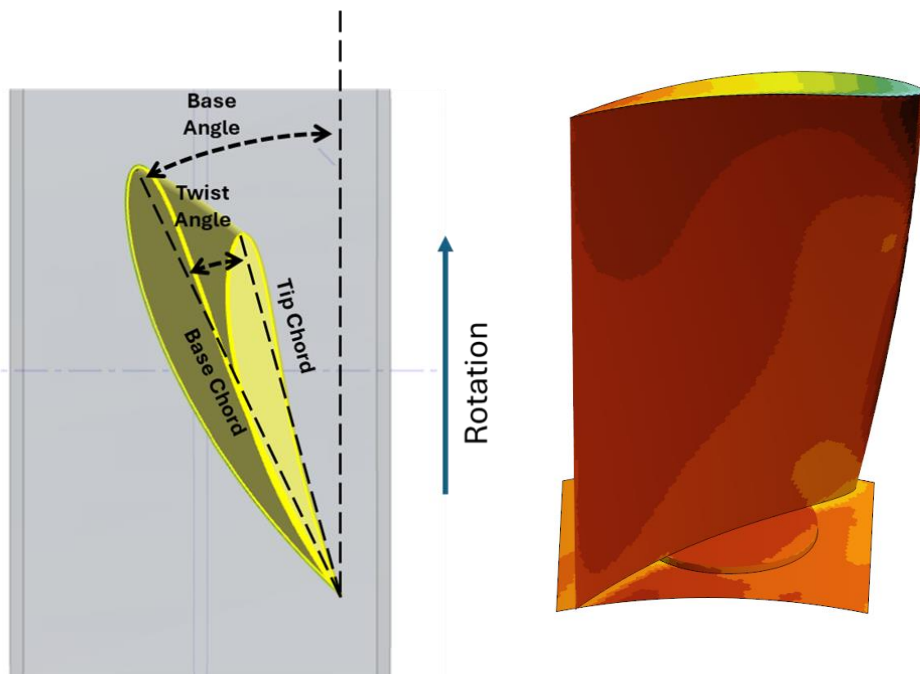
Simcenter STAR-CCM+



We have successfully applied design optimization techniques to determine baffle placements in industrial precipitators as a prime example of airflow optimization. These systems typically consist of 3 to 5 banks of electrostatically charged precipitator tubes, with turning vanes positioned beneath them to direct airflow. The objective is to configure the vanes to achieve uniform airflow through the tube banks while minimizing backflow flow in individual tubes.

To accomplish this, we parameterize the position and number of vanes below the tube banks. This allows us to automate the exploration of the design space, evaluating hundreds of potential configurations to identify the optimal solution. This approach is significantly more efficient and cost-effective than relying on a single engineer to manually navigate the design space.

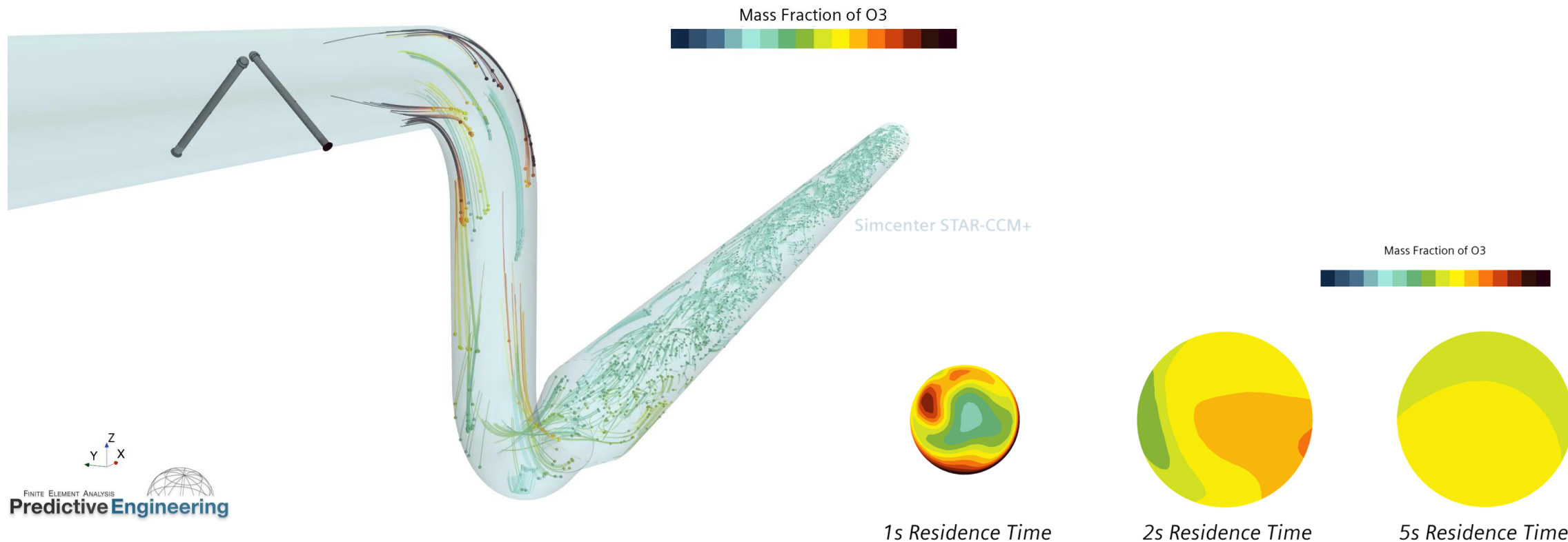
## Fan Blade Design Optimization



Proper fan blade design is often overlooked, yet improper designs can result in poor or unstable flow through the system. In this study, we focused on a high-pressure and high temperature airflow modeling application. From the baseline CAD geometry of the original fan hub, we broke down the blade design to NACA foil profiles and clearly identifiable parameters such as chord lengths and angles. With the discussed optimization procedure, we developed an automated workflow to efficiently assess the influence of these parameters on the design. By integrating this workflow with optimization algorithms, we explored the entire design space, gaining a deeper understanding of parameter impacts and identifying an optimal design for the application. As a result, we achieved blade designs that significantly improved both efficiency and increased flow rate 50% compared to the initial candidate.

## Probe Placement Optimization - Dispersion

Simcenter STAR-CCM+



This project focused on evaluating the injection of ozone gas into a production waste stream to remove harmful pollutants. A key challenge was designing injection probes capable of achieving uniform mixing at the required downstream distances.

To address this, we parameterized the hole locations and sizes on the injection probes, enabling us to identify a design that met the client's performance requirements. This repeatable and systematic approach saved the client significant time and costs while delivering an effective solution

## Predictive Engineering FEA and CFD Consulting Services, Portland, OR USA


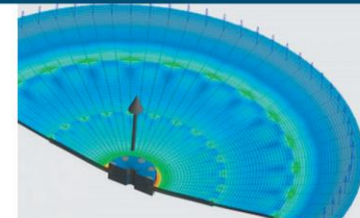
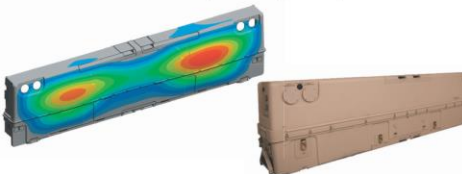
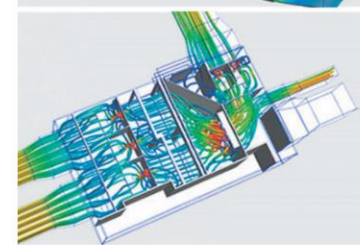
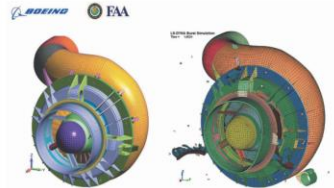
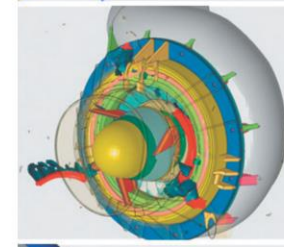
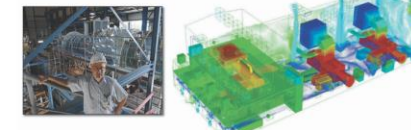
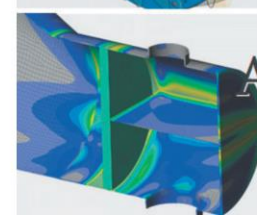


FINITE ELEMENT ANALYSIS  
**Predictive Engineering**

Finite element analysis consulting services, software, training and technical support

- Composites, Pressure Vessels, Vibration
- NASTRAN: Linear Dynamics
- LS-DYNA: Drop-test, Impact, Burst Analysis
- STAR-CCM+: Thermal/Flow Analysis



PROJECT EXAMPLES	OUR SERVICES
<p><b>FAA 16g sled test validation</b></p> <p>Femap FEA Model    FEA + LS-DYNA Model    Validation is Gold</p> 	 <p><b>FEA</b></p>
<p><b>Stress and Vibration Analysis of Large Composite Container</b></p> 	 <p><b>CFD</b></p>
<p><b>LS-DYNA turbine burst simulation</b></p> <p>BOEING    FAA</p> 	 <p><b>LS-DYNA</b></p>
<p><b>CFD study on co-generation power plant building</b></p> 	 <p><b>ASME BPVC</b></p>

We welcome your inquiry about how we may digitally prototype your design from mechanical to thermal fluids simulation.