GE Power & Water Summer Fellowship

Secondary Flows in a Gas Turbine Engine

David Weston
September 16, 2013

BYU
MECHANICAL ENGINEERING

Introduction

Secondary flows are flows that are not directly involved in the Brayton cycle of a turbine engine, but are very important in maintaining the integrity of the turbine. The secondary flows prevent hot air from the combustor from entering the rotor cavity and interacting with the rotor disc. In this way, the rotor disc does not have to be constructed from the same heat resistant material as the turbine blades themselves.

Thermally Resilient Material
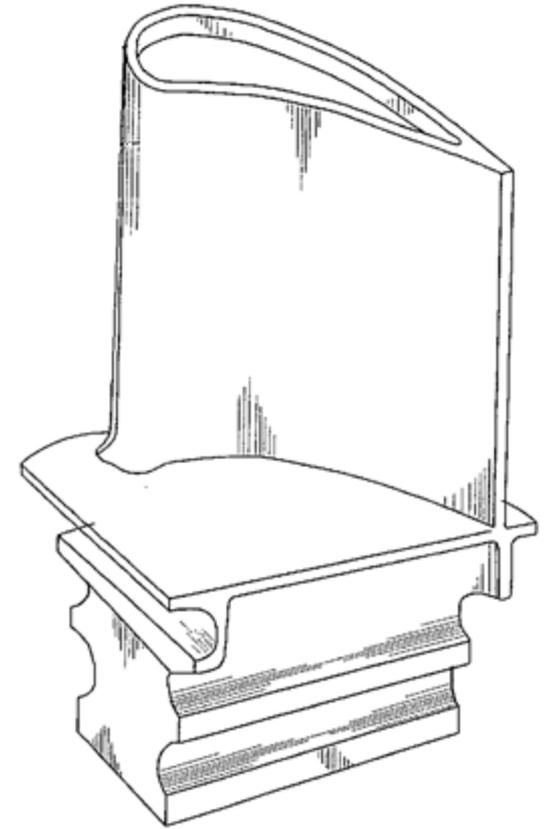
Thermally Sensitive Material

Introduction

Some air is removed from the primary flow path before it reaches the combustor. This air has experienced some viscous heating from the compressor, but is cool compared to the air leaving the combustor. It is injected into the primary flow path through the gaps between turbine rotor/stator stages and keeps hot air from entering the hub cavity through those gaps.

imagination at work
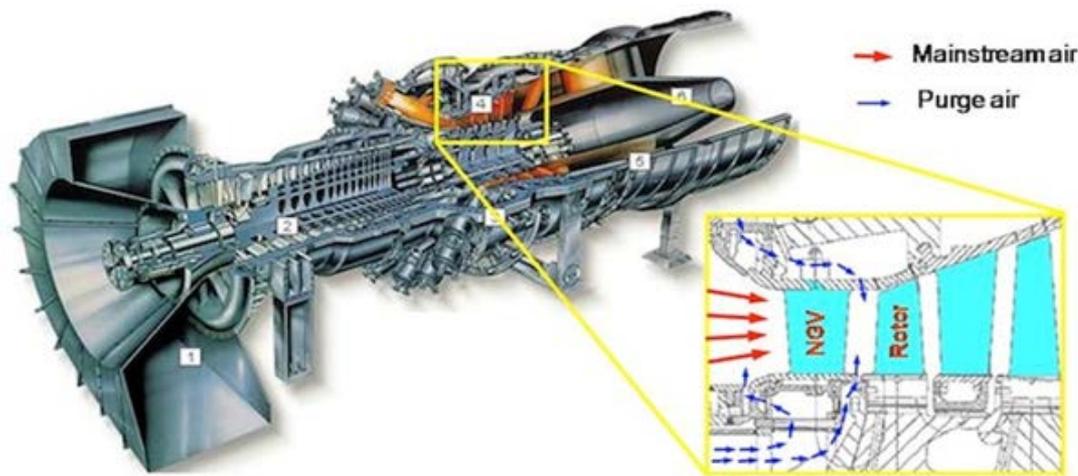
Introduction

These secondary flows are necessary to maintain the health of the turbine. However, their net effect reduces the efficiency of the engine, since the air is drawn off after the compressor has done work on it and is re-injected where the turbine cannot extract work from it. The goal of my work on the secondary flows was to **increase the overall efficiency of the engine by reducing the amount of air removed to maintain the turbine's integrity.**
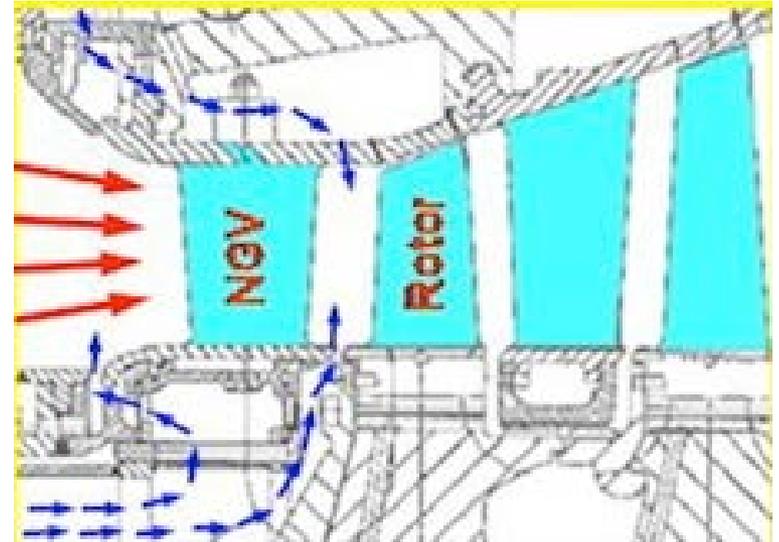
Introduction

My work for the summer consisted of 3 separate areas of work for improving the configuration for the wheel-spaces of GE's gas turbines.

1. Validate the CFD model

2. Propose various possible configurations

3. Analyze the performance of each possible variation

imagination at work
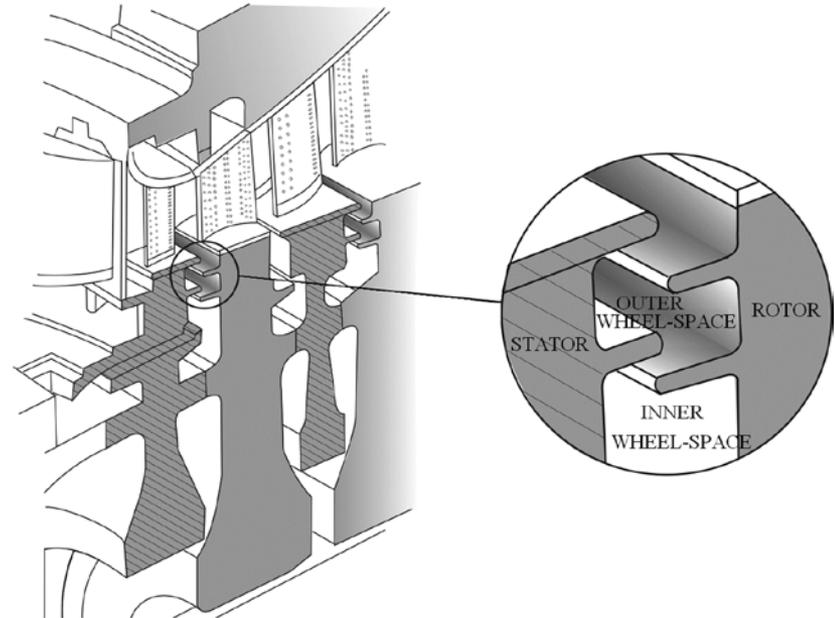
Validate the Model

Before proposing any new wheel space configurations, the CFD model has to be validated. GE has a scaled down version of their turbine that uses heated air from a compressor to simulate the air from the combustor. This test rig has a dense array of instrumentation that obtains a detailed suite of information for every configuration tested. This detailed information makes the rig geometry ideal for validating CFD, since the experimental data is dense enough to compare with a high fidelity CFD simulation. The first phase of my work was to validate the CFD simulations I performed against the data obtained experimentally from the test rig.

Validate the Model

In order to establish a robust validated case, I ran three separate cases to compare to experimental data.

Simulation	Description
High Purge	The rig is run with much more purge flow than it would see in normal operation
Low Purge	The rig is run with much less purge flow than it would see in normal operation
Nominal Purge	The rig is run with nearly the same purge flow that it would see in normal operation

Note: To differentiate between purge air and hot gas path air, the purge air is seeded with CO_2 before injection into the wheel space. This provides a quick and convenient way of measuring the amount of ingestion seen at any point in the wheel space, because ingested air will drop the CO_2 concentration at that particular location.

imagination at work

Validate the Model
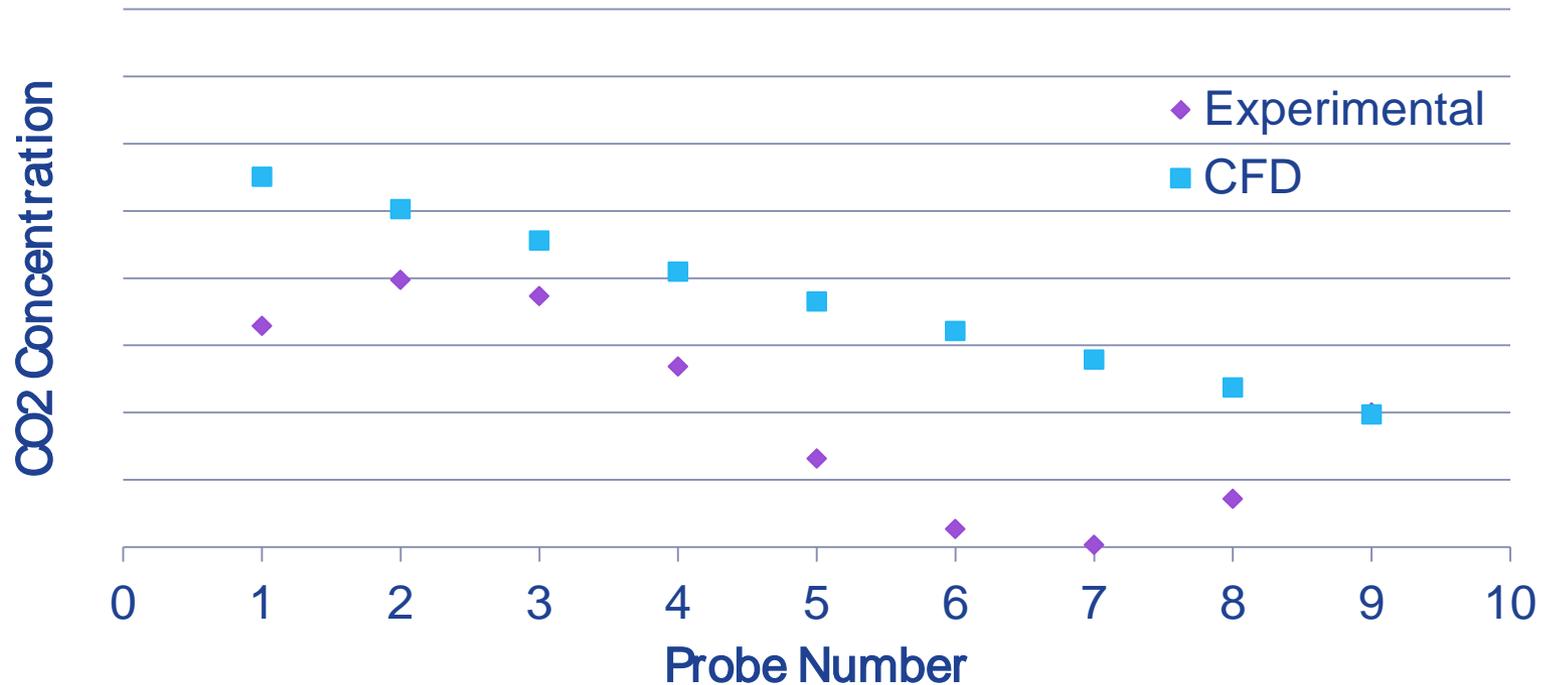
This is a sample set of results from the a CFD simulation plotted against a sample set of data representing the experimental results from the rig. The y-axis represents CO_2 concentration. These results show a significant difference between the CFD and experimental data.

Validate the Model

With a difference as significant as the one shown on the previous slide, some work needs to be done to bring the CFD in line with experimental results before making any decisions based on CFD data. This generally means that a lot of trial and error will be required to find whatever is causing the difference between the two sets of results. Some possible solutions are:

1. Refine the grid in key areas

2. Expand the simulation to cover a larger axial or circumferential extent

3. Apply a different turbulence model to the simulations

Any one of these factors could cause the deviation seen in the previous slide, as well as several other individual factors or a combination of many factors.

imagination at work

Generate New Concepts

The effectiveness of a seal configuration depends on how well it can isolate the air in the inner wheel-space (which is very thermally sensitive) from the outer wheel-space (which is thermally resilient). Specific seal configurations are generally proprietary, and each company has their own guidelines for creating new seals.

imagination at work
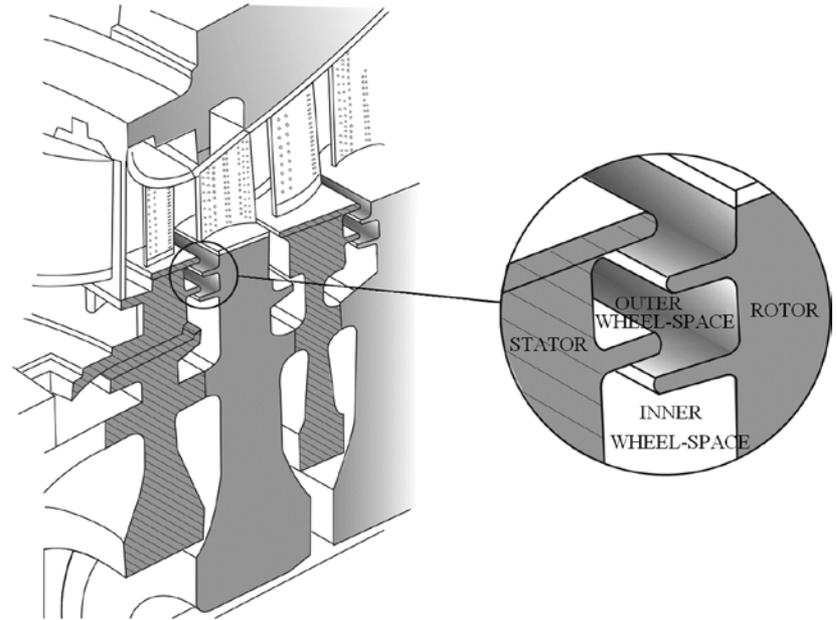
Generate New Concepts

Once a configuration has been proposed and approved, the idea needs to be translated into a model. NX 7.5 was used to generate the geometry for the various proposed configurations.

Stationary Domain   Domain Interface   Rotating Domain

imagination at work
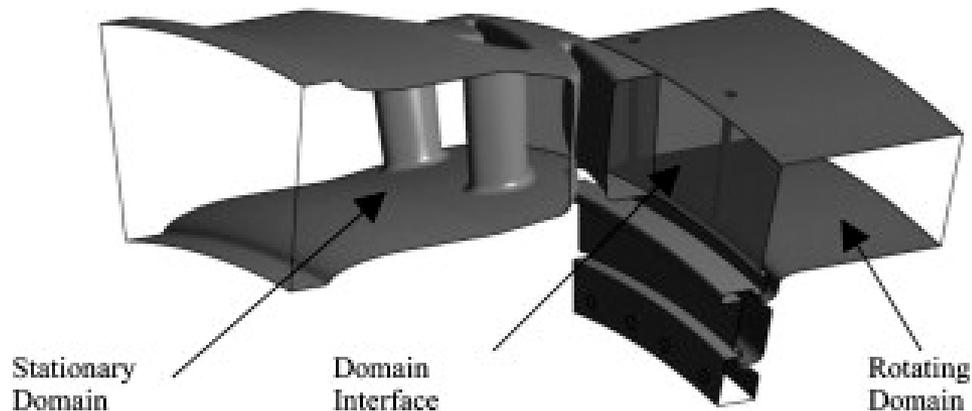
Generate New Concepts

After generating the CAD model, the model is meshed in ANSYS Workbench. This process generally requires some iterative improvement to create a mesh that is an acceptable quality for the flow solver. As configurations become more complex, this may even involve some changes to the geometry in order to reach the requisite quality.

imagination at work
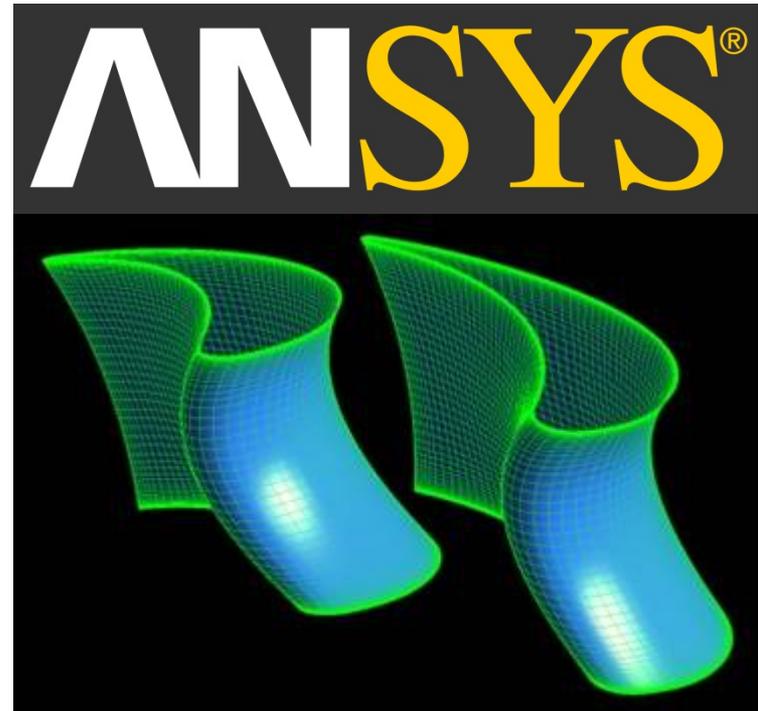
Analyze the Performance

The finished mesh of the new configuration is input into the ANSYS CFX Solver. The most current predictions for flow temperatures and mass flows are used as boundary conditions for each run, and each simulation is run a fixed number of iterations to reach convergence.

Sample CFX output from a turbomachinery simulation.

imagination at work
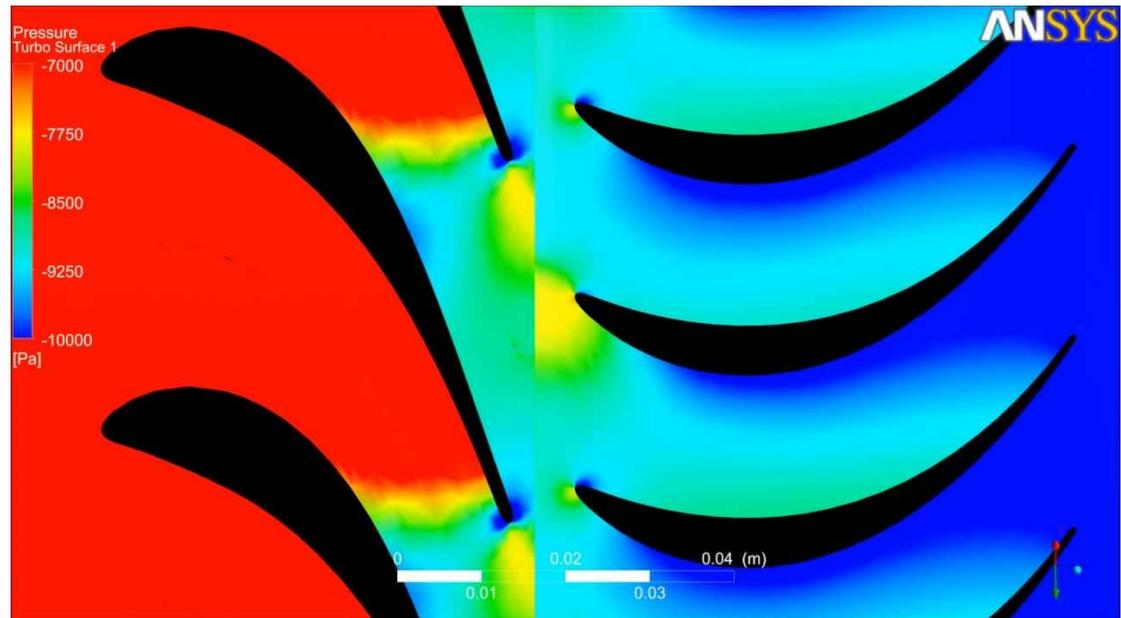
Analyze the Performance
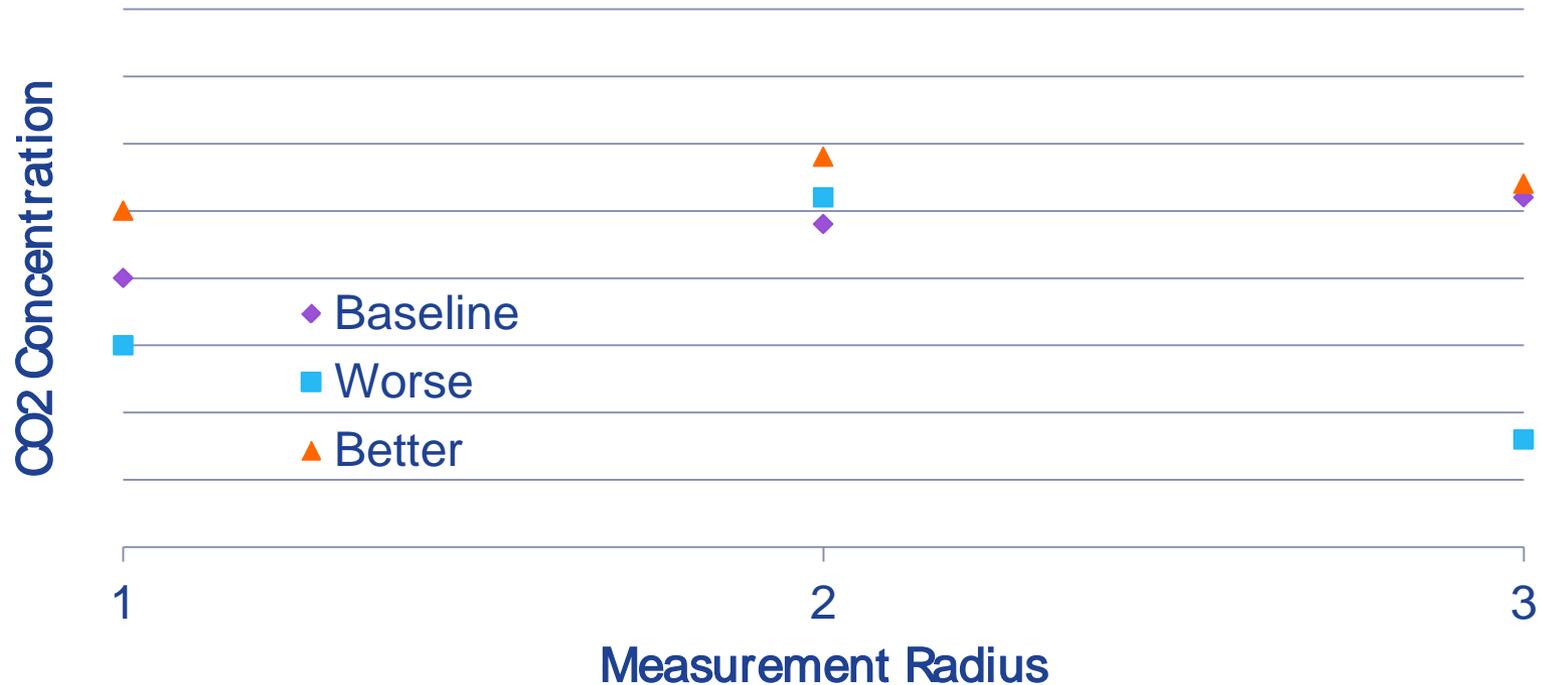
The process for analyzing the various configurations is nearly the same as the process for validating the CFD results. Data is extracted from the simulation at various crucial points and compared with a "Baseline" case to determine whether a particular concept should be investigated further or discarded.

Analyze the Performance

This process is repeated until a sufficient number of data points have been produced, and the whole set is compared to determine which configuration will provide the best sealing characteristics. If any further improvements are desired, variations on the best of the original concepts may be proposed and studied to enhance the seal performance. The final seal configuration may be incorporated into the next generation of gas turbines.

imagination at work
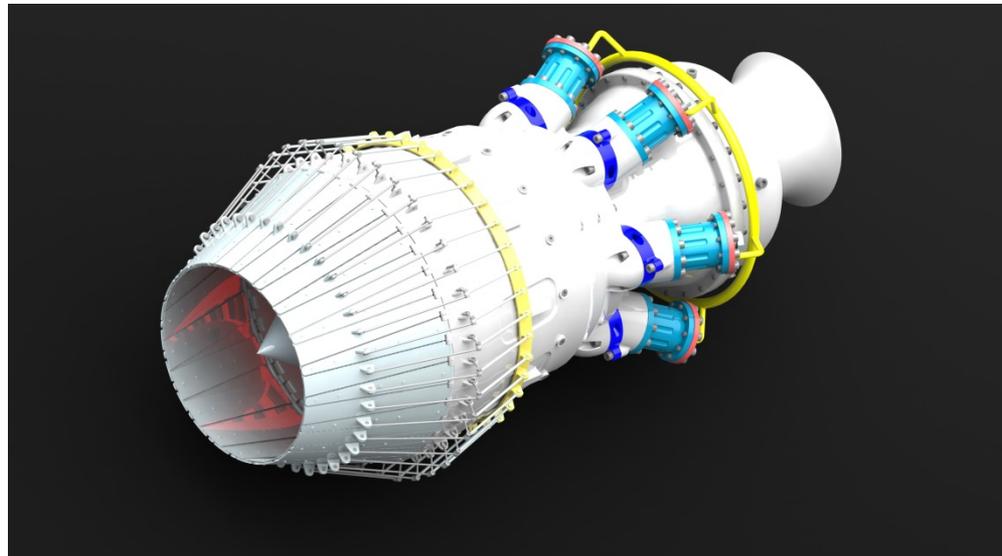
Acknowledgements

I would like to thank GE Power & Water for hosting my fellowship. I would also like to thank David Johns for inviting me to join his team for the summer. My thanks go to him, as well as to Richard Johnson, Andrew Giametta, Ben Ding, and Radu Danescu for their help in understanding the work I was doing and the concepts that I needed to employ to accomplish it.

imagination at work