

UTSR Gas Turbine Industrial Fellowship Report

Daniel Reimann

Capstone Turbine Corporation

Introduction

Capstone Turbine Corporation is the leading producer of gas turbine engines used for power production in the 30 to 200 kW range. These microturbines provide a low emissions solution to a variety of energy needs. They are used as a source of power generation within cities, at landfills converting landfill gases into electricity, on offshore oil platforms which face harsh weather conditions, and have even been used in vehicular applications.

My experience this summer working at Capstone has been very valuable. I have been able to work and interact with other engineers, gain exposure to other disciplines, learn more about the manufacturing processes of gas turbine engines, as well as better understand the workings and organization of an engineering company. I have also gained new knowledge and learned new skills using tools of the engineering trade.

During my eleven week fellowship there were several projects that I worked on. The first project was the development of a heat transfer model within ANSYS. ANSYS is a software package typically used for stress analysis. I was also involved with an experimental project acquiring compressor test data. These compressor data are used to validate and update performance predictions. A third project I worked on was the modeling of a turbine nozzle, currently being used in a Capstone engine as a supporting project to a combustor redesign.

Heat Transfer Model

Heat transfer models are important because they predict temperatures which may be present in any part of the engine. This temperature information is used to aid in the design of components, the selection of component materials, and is used for predicting

stresses within the engine. Capstone already had existing heat transfer models, but wanted a heat transfer model in ANSYS with an integrated flow network. A heat transfer model in ANSYS is desirable so that the thermal and stress analysis can be coupled in the same model. The integrated flow network represents the secondary air flows through the core of the engine. These secondary air flows act as cooling for various components within the engine.

My contribution to this project was the development of the heat transfer model in ANSYS with the secondary fluid flow network. Since I had never used ANSYS before coming to Capstone, I spent some time learning how to use the program. In order to automate the process I wrote a script which imports the engine geometry, creates a flow network within the engine, applies the desired boundary conditions and flow rates, and uses the defined conditions to solve for the temperatures in the engine model under steady state conditions. Using this script, boundary conditions can be easily changed and the model can be run very quickly again. A color plot showing an example temperature distribution is shown in Figure 1 below. Due to the proprietary nature of the information included in all of the following figures, no numeric values will be shown.

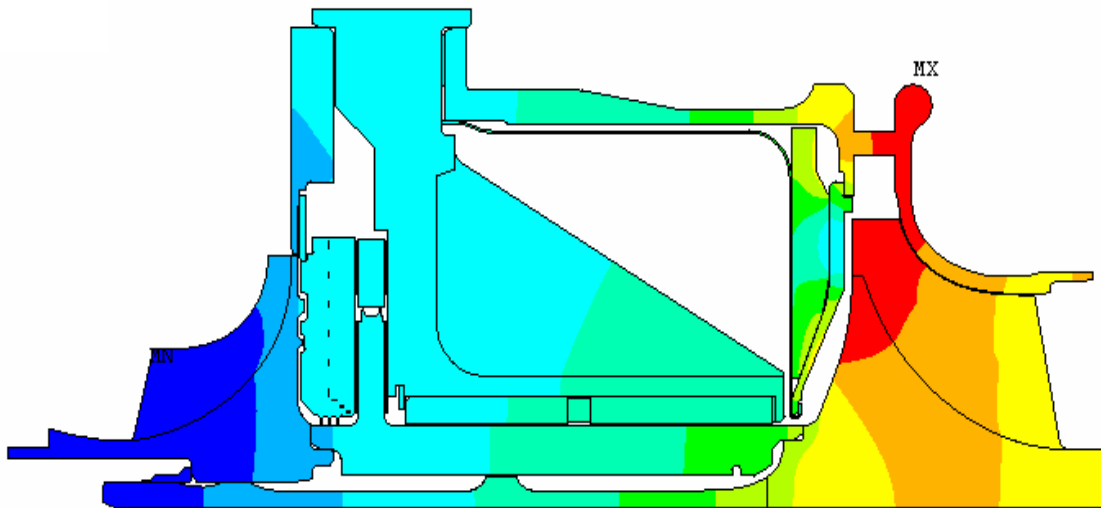


Figure 1: Contour plot of temperature distribution in the engine core

Further development of the model will continue after the end of my fellowship experience. This will include refining and validating the model, creating a live flow network where the flow rates will be solved for, as well as the capability of solving the temperature distributions under transient conditions.

Compressor Rig Testing

A compressor testing rig is owned by Capstone and is used to take measurements of the compressor stage of the engine. These measurements can be used to obtain performance characteristics of the compressor stage. By controlling both the compressor rotational speed as well as the mass flow rate, a non-dimensional compressor map can be created and used to predict the performance of the compressor under any operating condition.

I collected compressor data using the compressor test rig, reduced the data, and assisted in formalizing the data acquisition procedures. The main objective for collecting these data is to obtain a more precise measurement of compressor efficiency. This is valuable information used to improve Capstone's performance prediction models. Figure 2 below is a photo of the compressor rig and Figure 3 is an example compressor map.

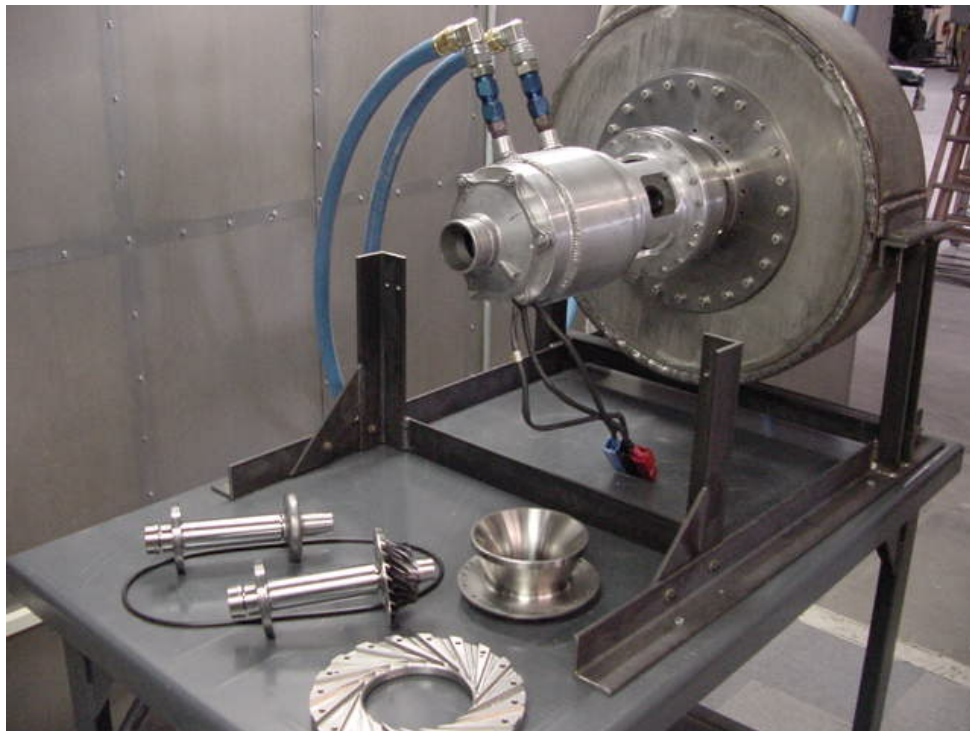


Figure 2: Un-instrumented compressor rig shell and compressor components

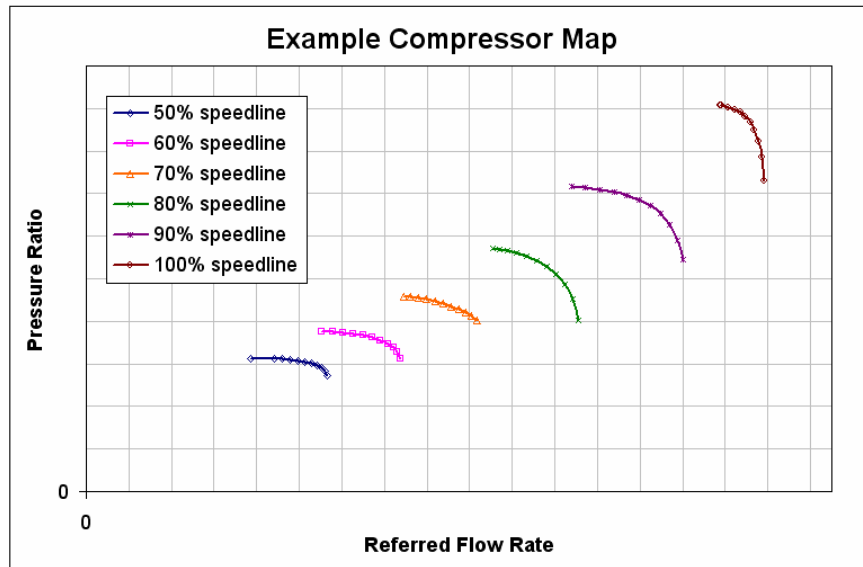


Figure 3: Typical compressor map displaying various non-dimensional speed-lines

Turbine Nozzle Modeling

As part of a continual effort to reduce the emission levels of the Capstone engine, the combustion group is currently working on design modifications of the combustor. As part of these modifications, the designers are able to control the momentum and temperature profile leaving the combustor and entering the turbine nozzle. The combustion group wanted to know how much swirl could be added to the nozzle inlet without significantly decreasing the nozzle efficiency. In order to understand the effect on nozzle efficiency and performance, a CFD study was performed to model the turbine nozzle under varying inlet conditions.

With the assistance of the turbomachinery tool G/Turbo, I created a three-dimensional model of the nozzle inlet geometry and the nozzle vane in Gambit. After the geometry was meshed and imported into Fluent, I varied the amount of swirl entering the nozzle geometry. As the swirl increases, the incidence angle changes which affects the nozzle performance characteristics. One of these performance characteristics is the vane efficiency defined by equation 1 below.

$$Eq. (1) \quad \eta_{Vane} = \frac{\left[1 - \left(\frac{P_{static-exit}^{area-avg}}{P_{t-abs-exit}^{mass-avg}} \right)^{\left(\frac{\gamma-1}{\gamma} \right)} \right]}{\left[1 - \left(\frac{P_{static-exit}^{area-avg}}{P_{t-abs-inlet}^{mass-avg}} \right)^{\left(\frac{\gamma-1}{\gamma} \right)} \right]}$$

Also as part of the study, the flow stagnation location on the vane as well as the vane loading were determined. To be able to visually represent this I have included color plots of a single plane within the computational domain at 50% of the vane span. The following figures are contour plots of Mach number and static pressure for both the nominal inlet conditions, as well as for the maximum amount of swirl studied.

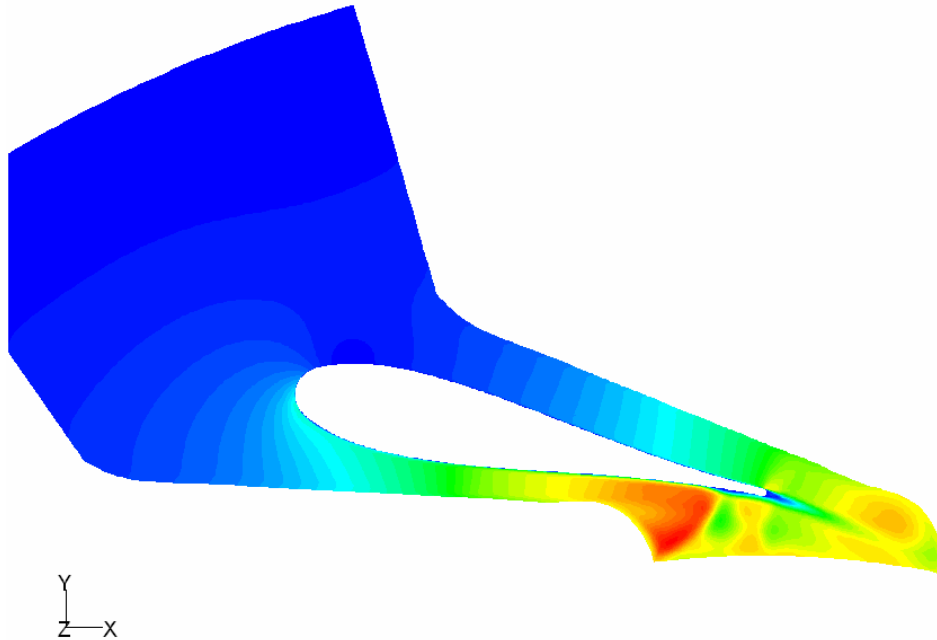


Figure 4: Contour plot Mach number through the vane passage at nominal inlet conditions

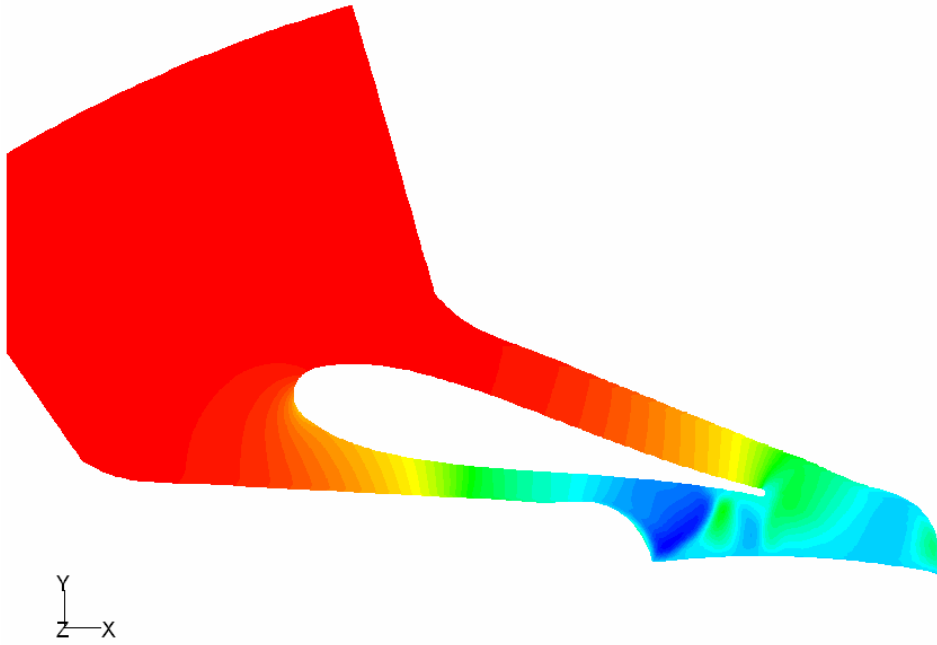


Figure 5: Contour plot static pressure through the vane passage at nominal inlet conditions

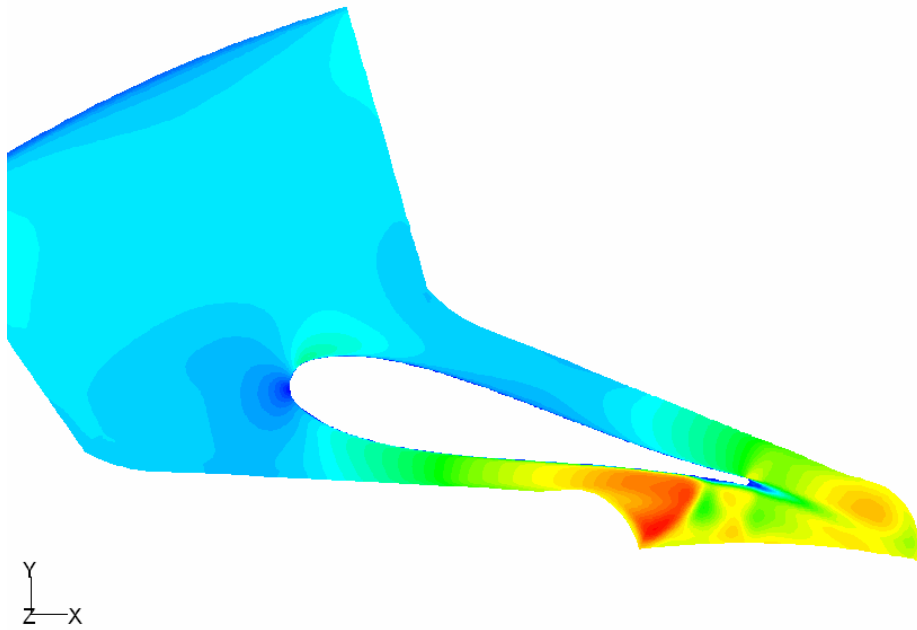


Figure 6: Contour plot of Mach number at maximum swirl inlet conditions

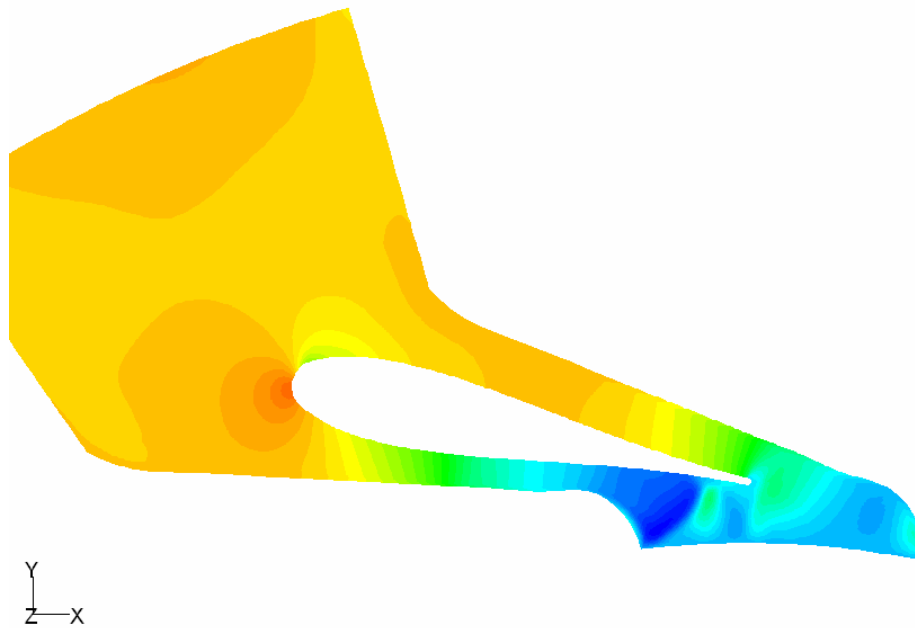


Figure 7: Contour plot of static pressure at maximum swirl inlet conditions

Acknowledgements

I would like to express appreciation to Miro Sir and Tony Ryu for their assistance and direction with the compressor rig testing and the heat transfer model. I would especially like to thank my mentor Dan DeMore for his continual help and for spending so much of his valuable time mentoring and teaching me throughout my time at Capstone. I would also like to thank University Turbine Systems Research for this great opportunity to learn and grow within the field of engineering.