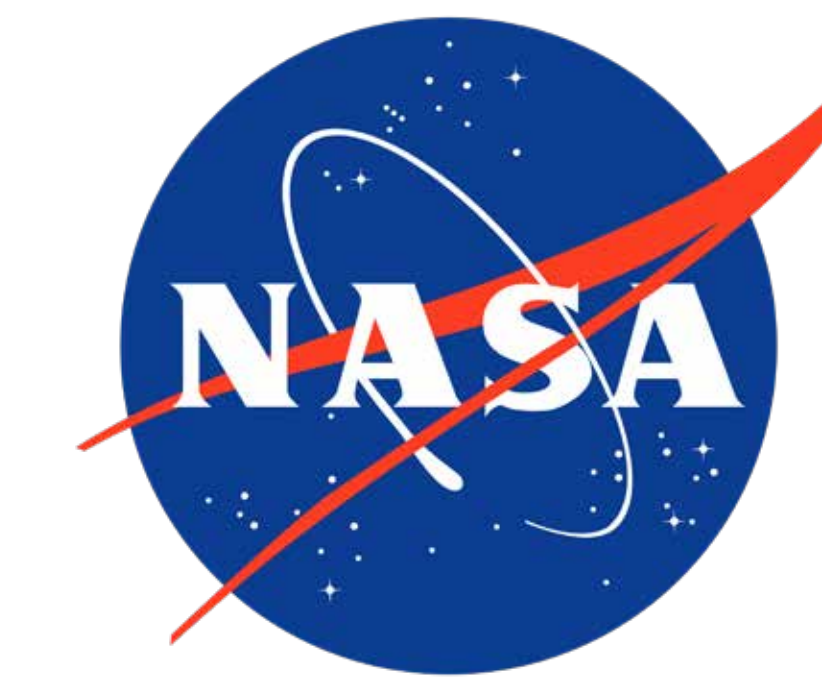


# GT2024-139262: Development of a Computational Fluid Dynamics Model for a High-Speed Centrifugal Compressor



Tammy Nguyen-Huynh<sup>1,2</sup> & Randall M. Mathison<sup>1</sup>

1. Gas Turbine Laboratory, The Ohio State University, Columbus, OH, 43235, USA  
2. Turbomachinery and Turboelectric Systems Branch, NASA Glenn Research Center, Cleveland, OH, 44135, USA

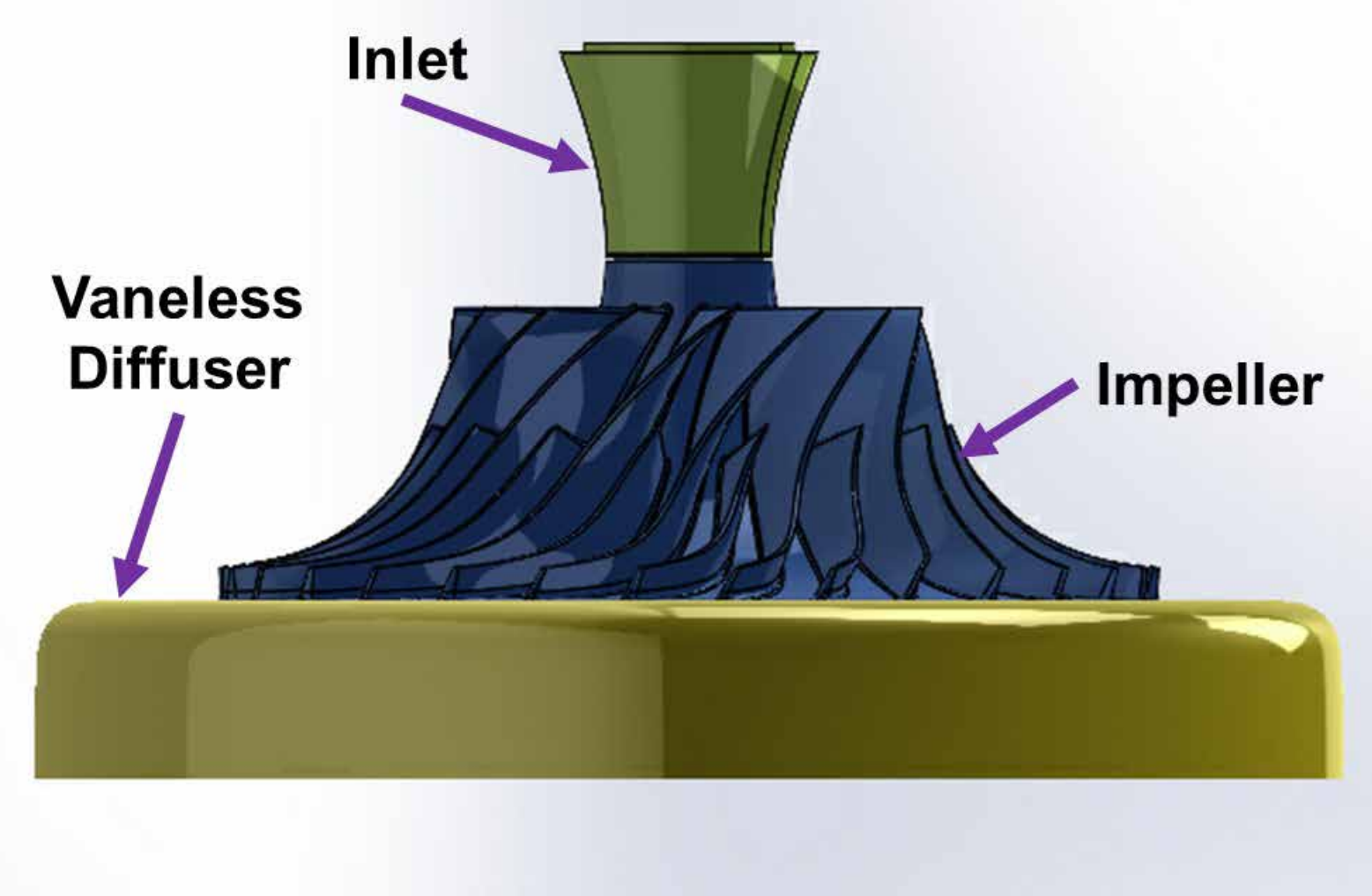
## MOTIVATION AND OBJECTIVES

The High Efficiency Centrifugal Compressor (HECC) was developed by United Technologies Research Center with aims of a high pressure ratio, high efficiency compressor. The motivations behind this effort and the work featured on this poster are to:

1. Investigate barriers that cause difficulties in predicting centrifugal compressor performance, and
2. Provide a high-quality open data set of the compressor's performance and geometry to the research community.

In this poster, a CFD model for HECC operating at its design point was developed and compared to an equivalent experimental case in order to gain better understanding of the limitations of CFD to determine an acceptable level of uncertainty.

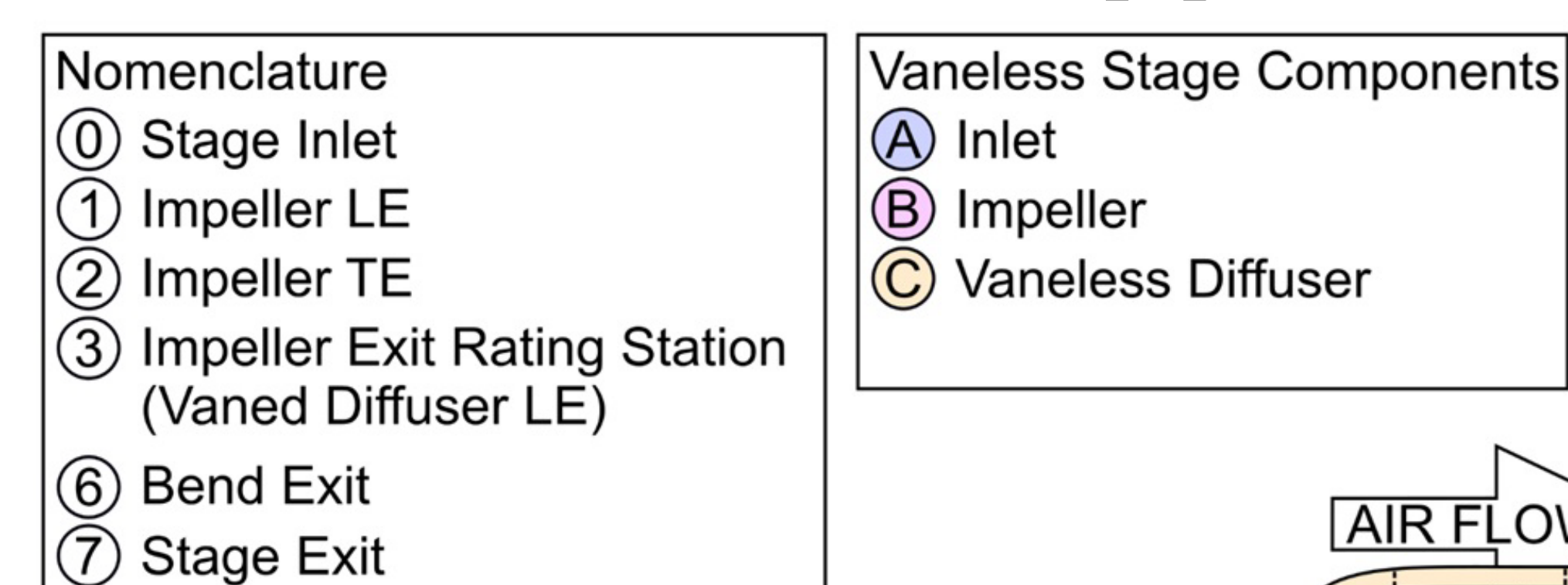
## VANELESS CONFIGURATION



### Design Specifications

Mass Flow Rate	11.0 lbm/s
Rotational Speed	21,789 RPM
Stage Total Pressure Ratio	4.55
Impeller Total Pressure Ratio	5.10

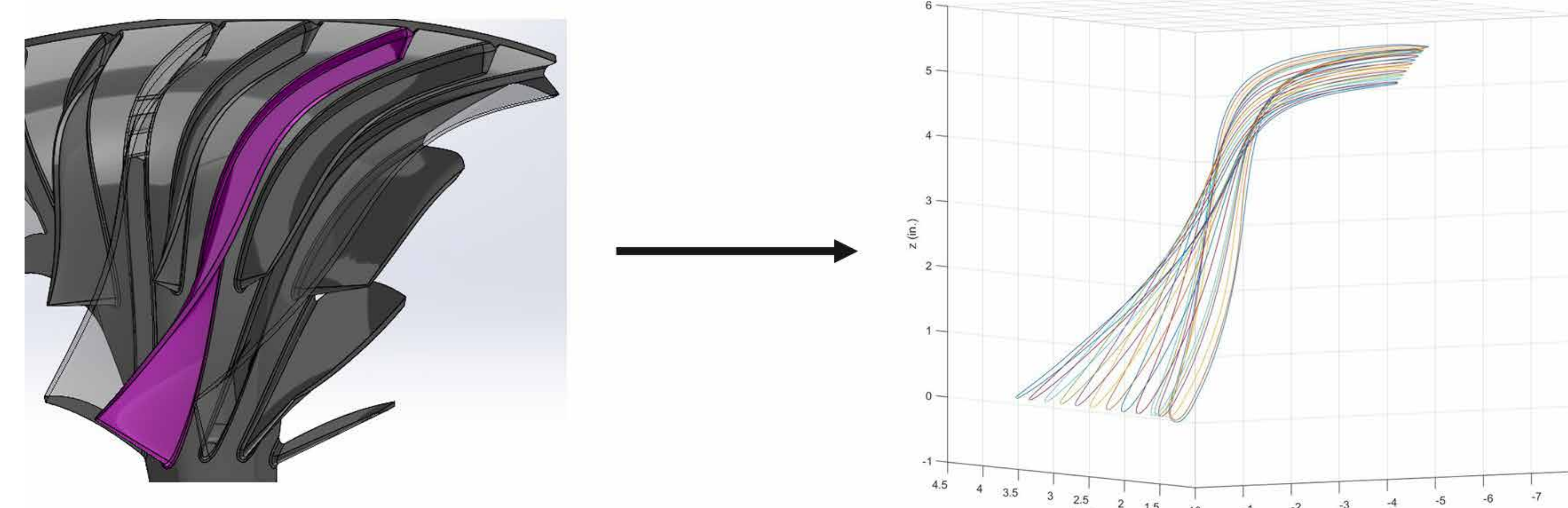
### Meridional View [1]



[1] Harrison et al., ASME Turbo Expo 2023, GT2023-103128

## METHODOLOGY

### Mesh Development



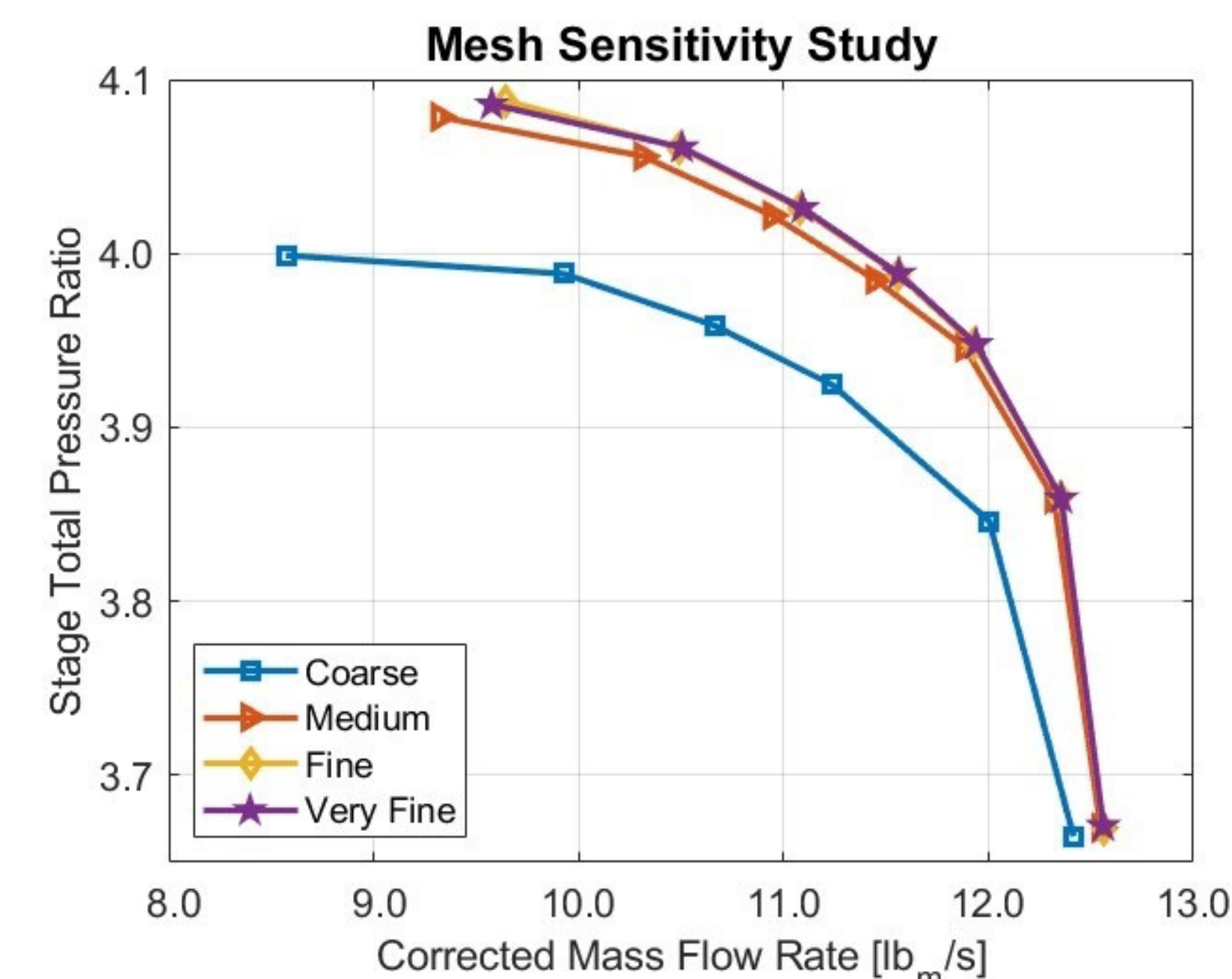
Blade profiles, including fillets on the hub, were extracted from the HECC impeller solid model to create the impeller mesh (pictured right). The remaining compressor components were meshed from endwall coordinates. The employment of a periodic boundary required the modeling of a single main-splitter blade pair.



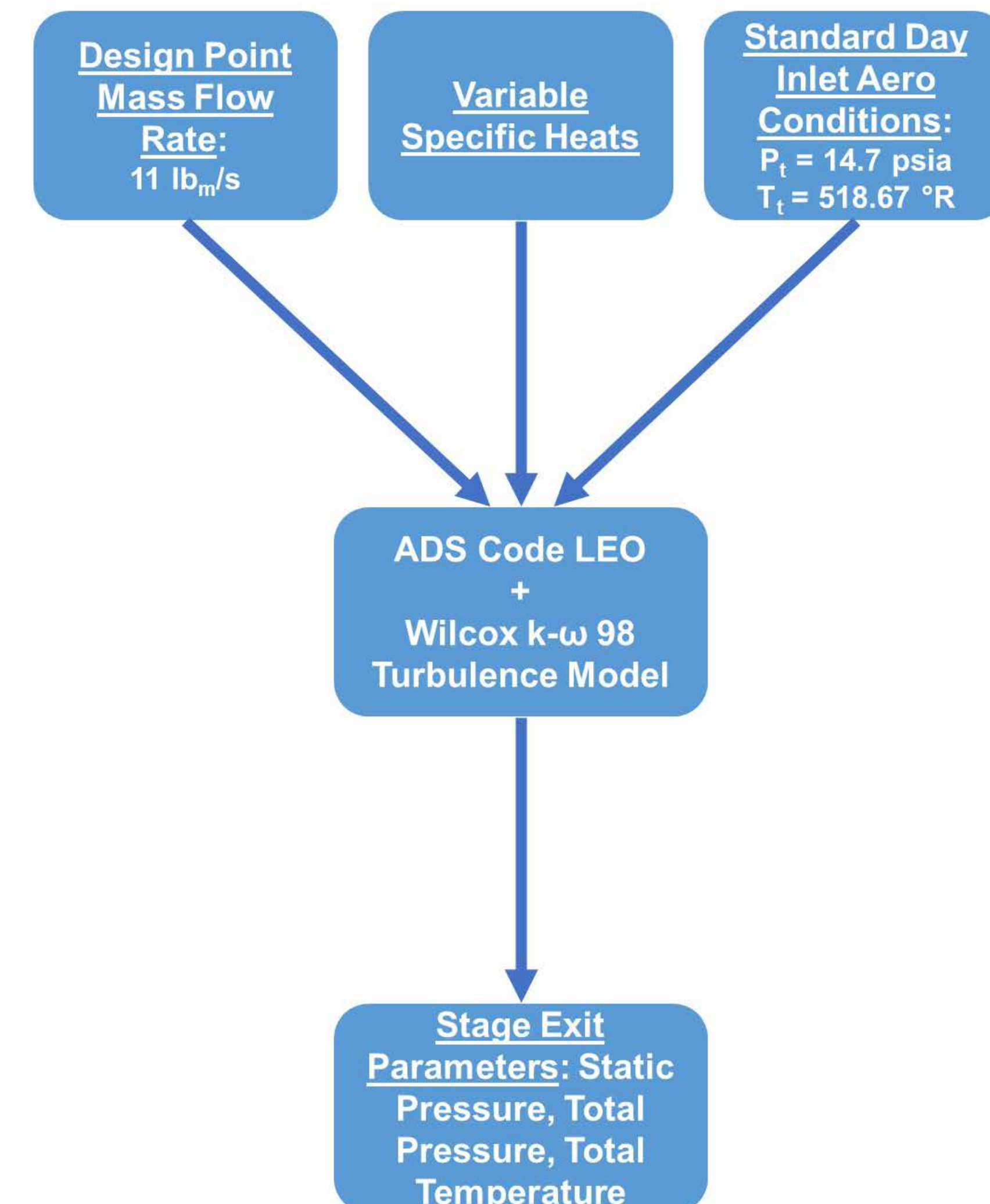
### Mesh Sensitivity Study

• Fine mesh utilized in analysis based on study results

Mesh Resolution	Grid Points
Coarse	766,599
Medium	1,313,887
Fine	2,027,603
Very Fine	2,794,519



### Model Setup



## CONCLUSIONS AND CURRENT WORK

- The model overpredicts performance, though predictions follow general behavior trends
- Impeller performance is more accurately simulated than stage performance
- The prediction of performance parameters such as total pressure ratio can only define how well performance is modeled. CFD models are multi-dimensional and high-fidelity models require accurate simulation of behavior in flow passages as well

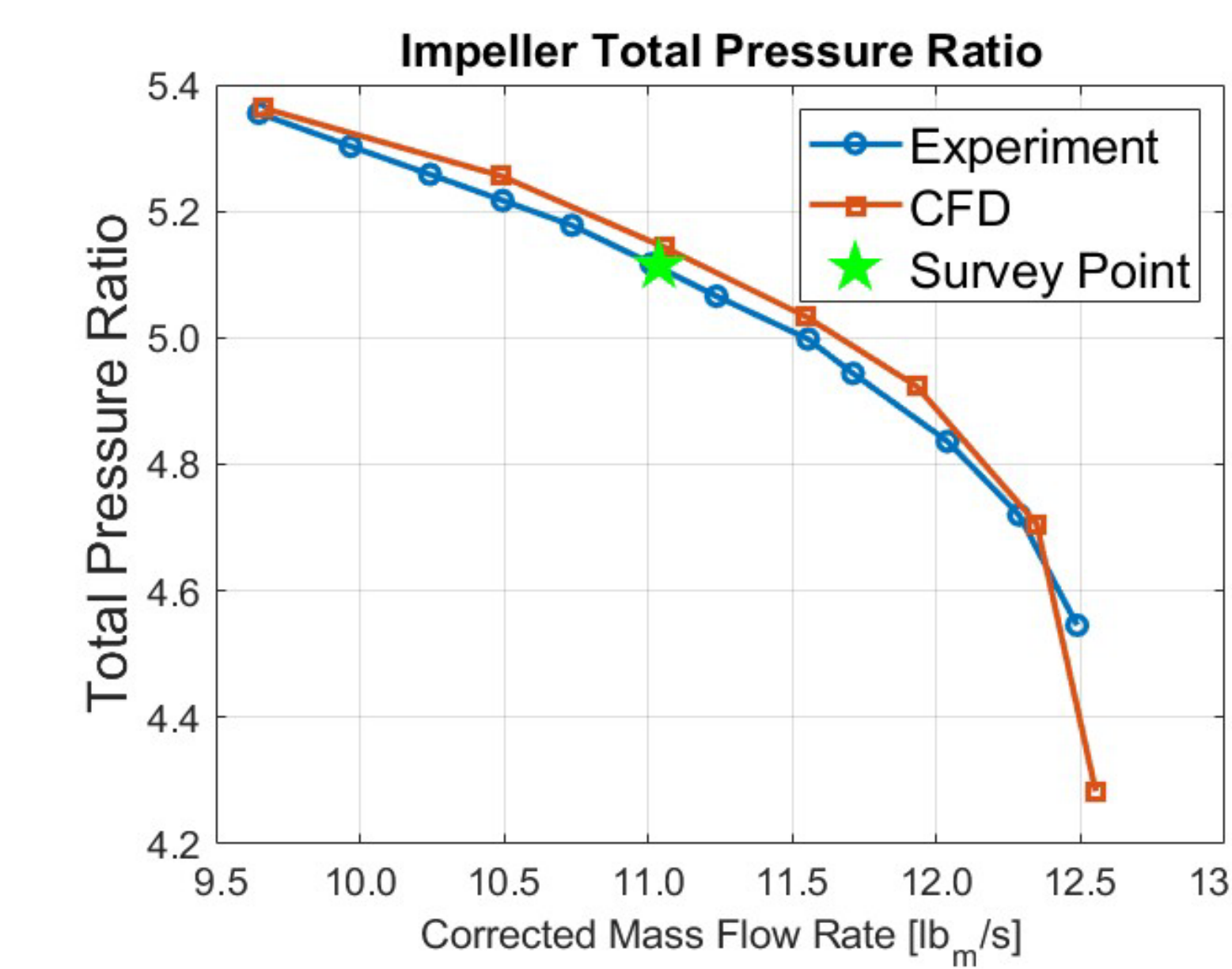
This model has been validated and used in analyzing HECC behavior in a geometric and performance review in collaboration with ADS CFD: GT2024-125128, GT2024-125360, GT2024-121641. The public archive contains geometric and experimental data of HECC and is accessible through the QR code.



## RESULTS

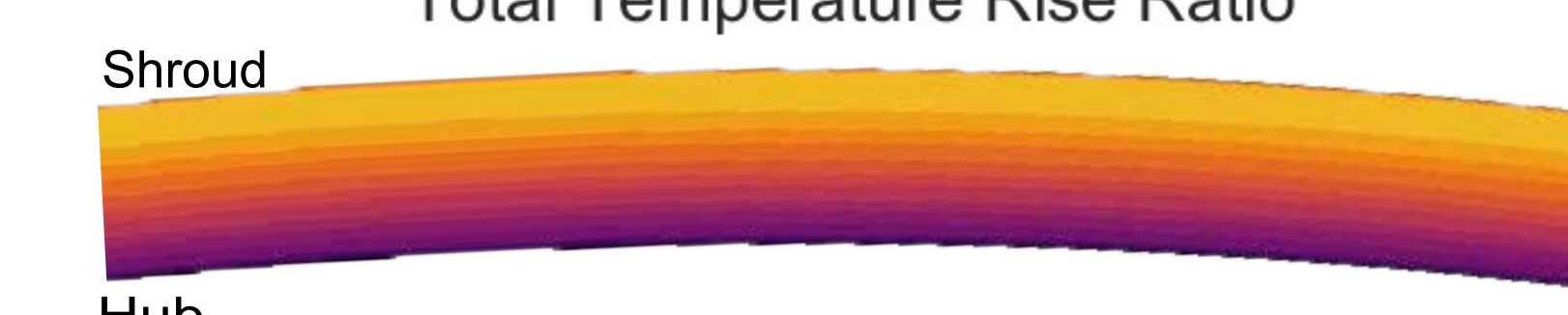
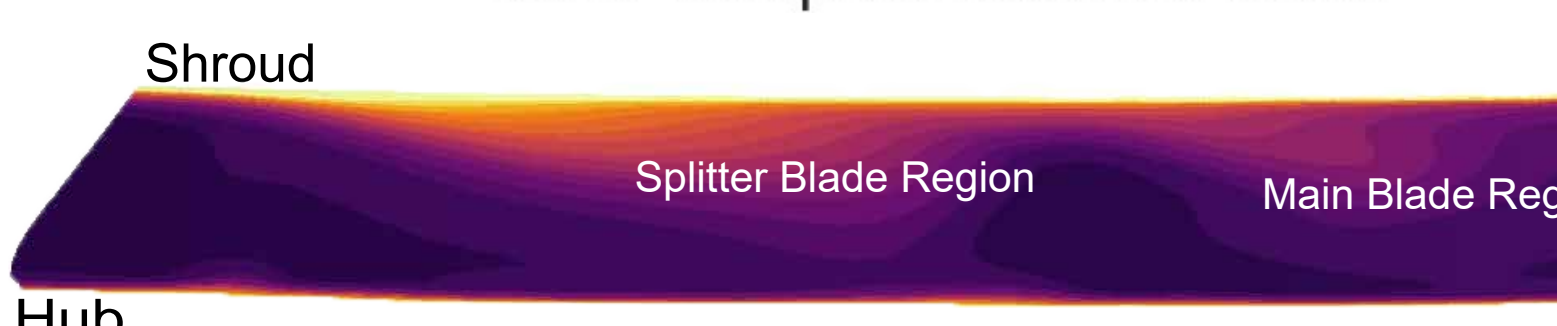
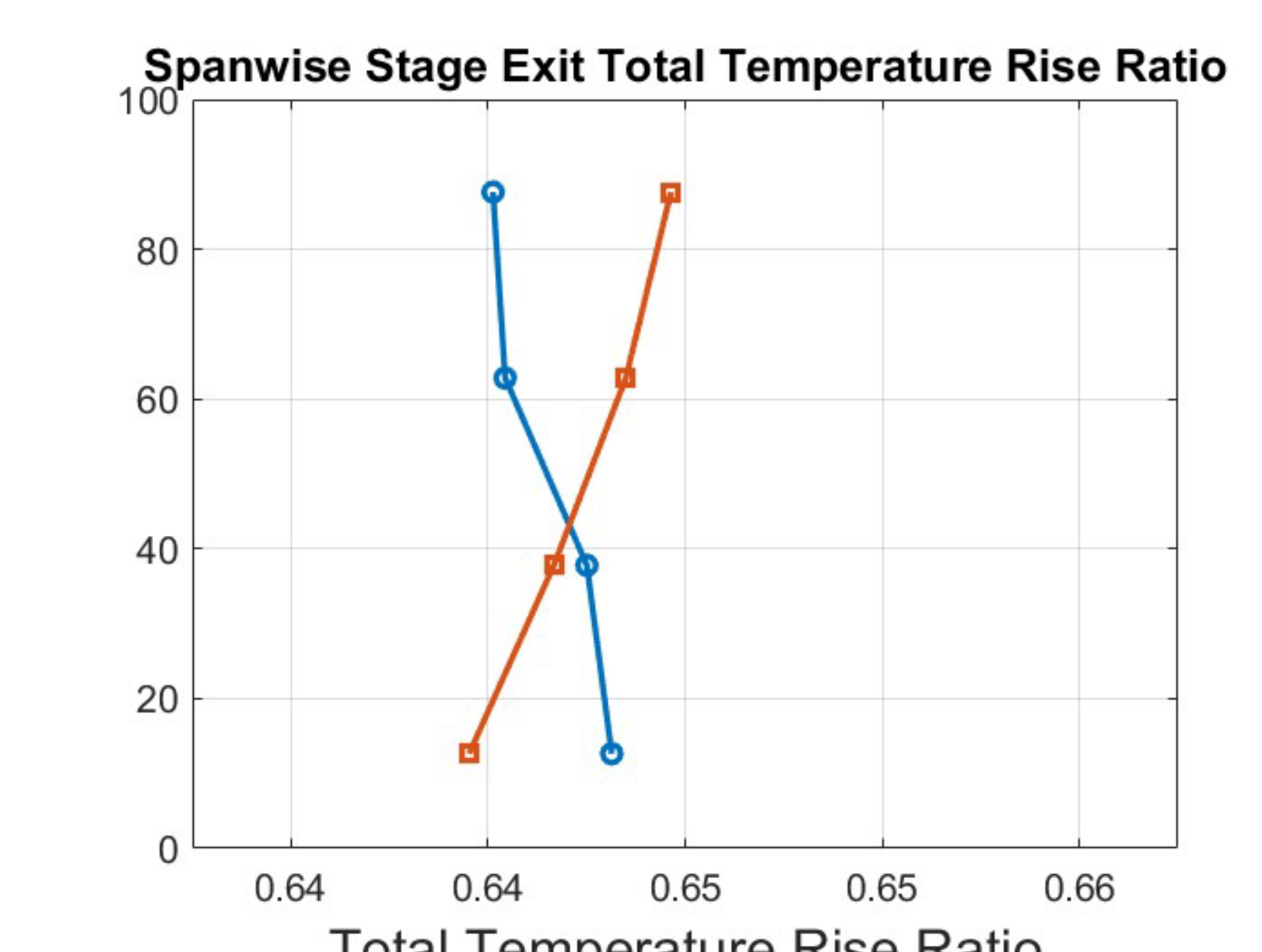
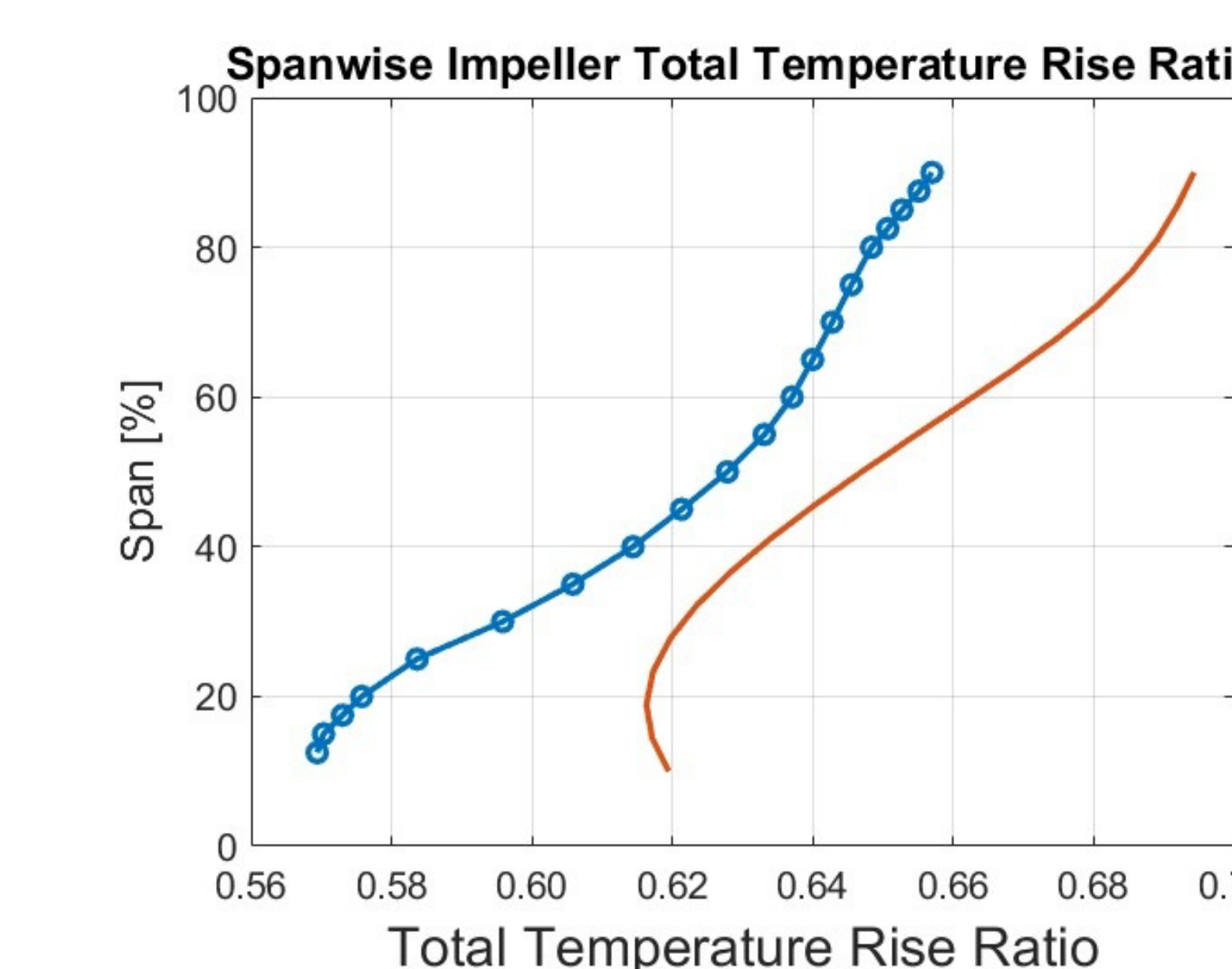
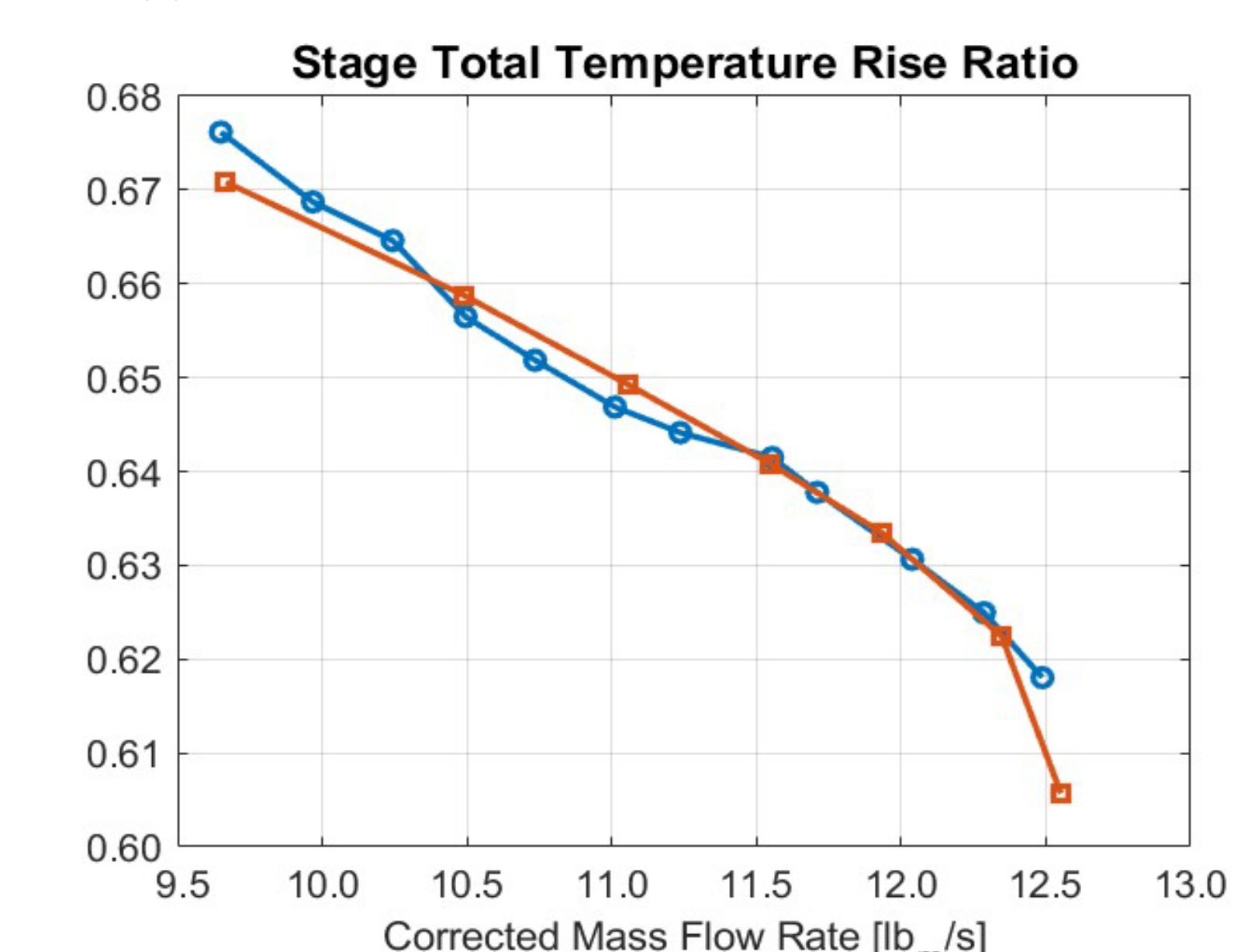
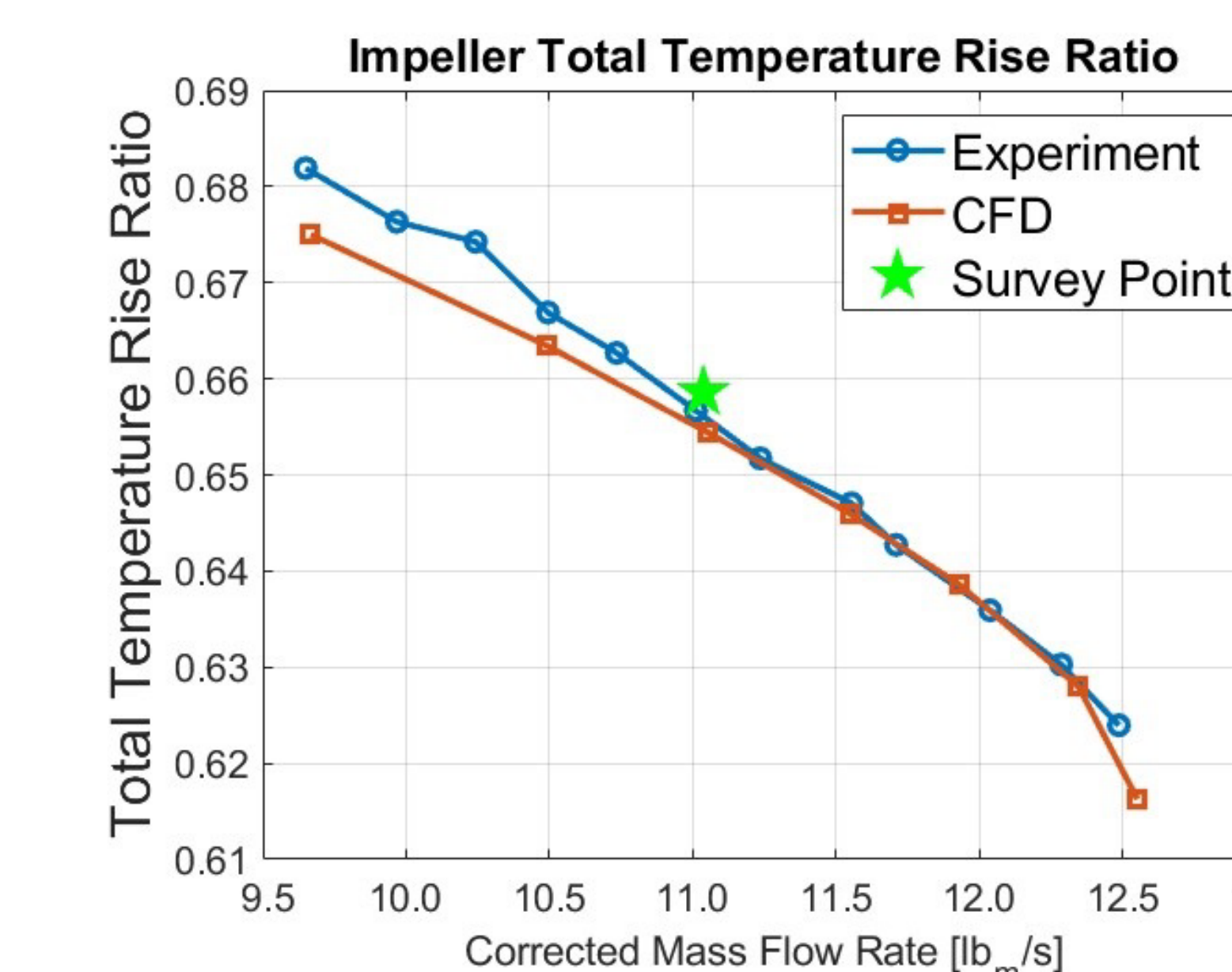
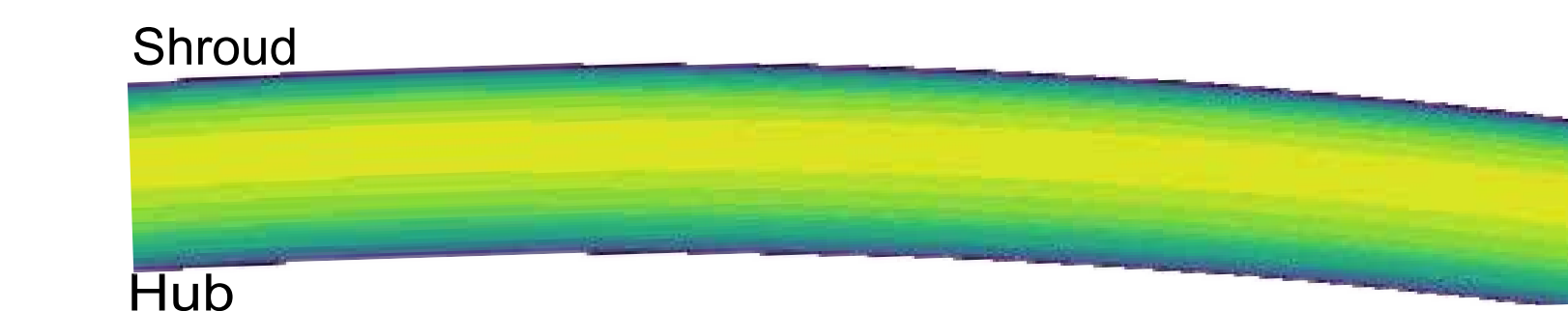
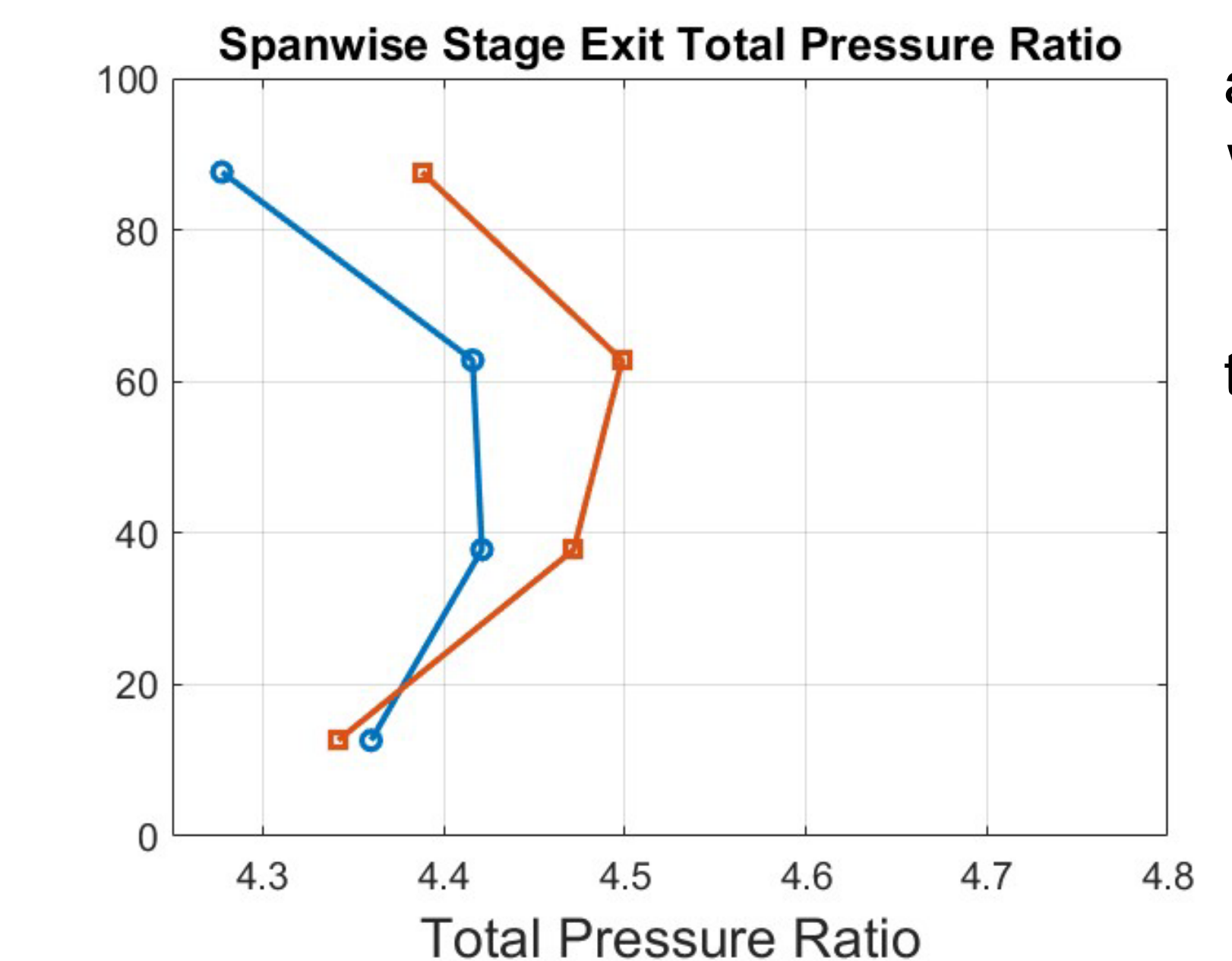
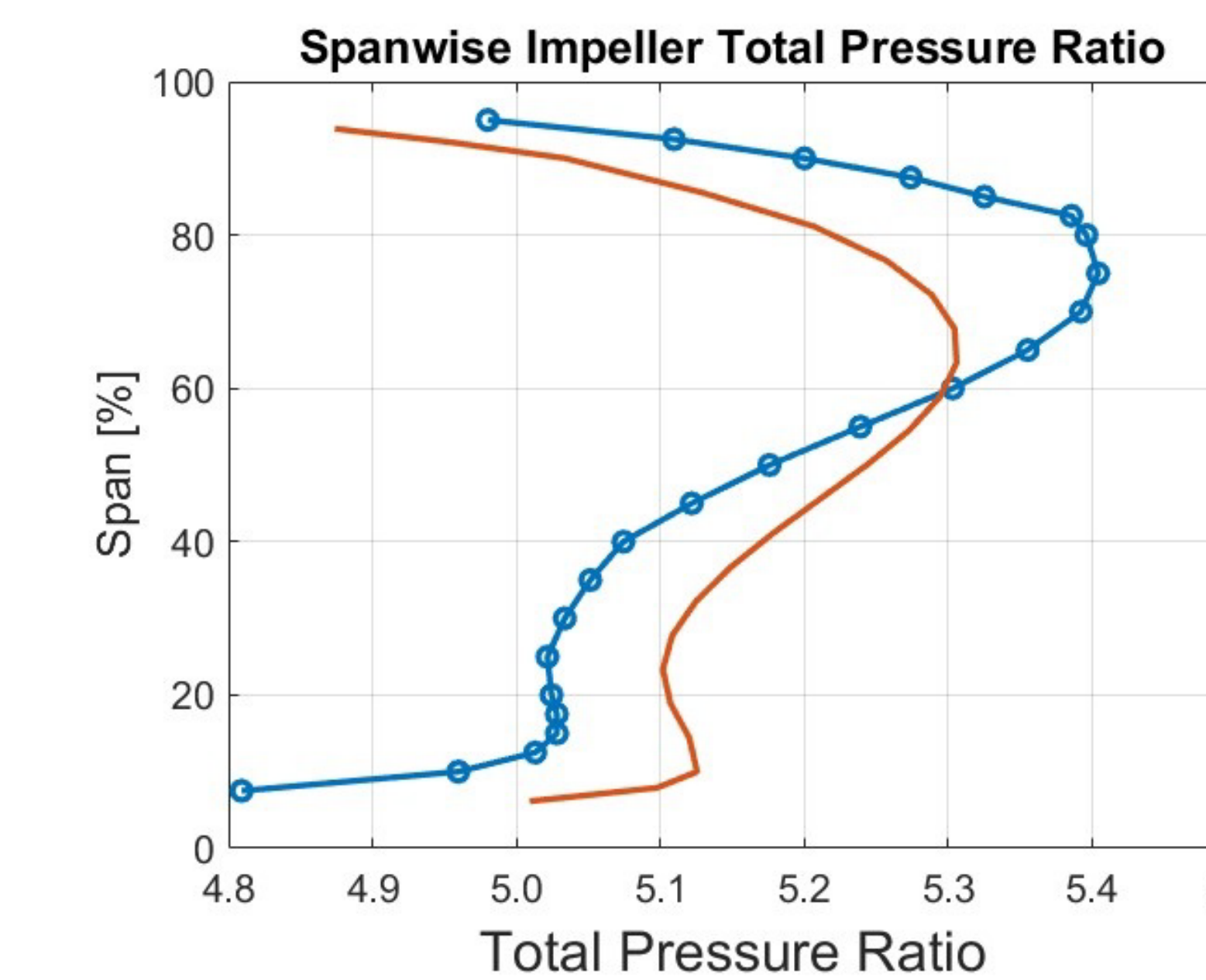
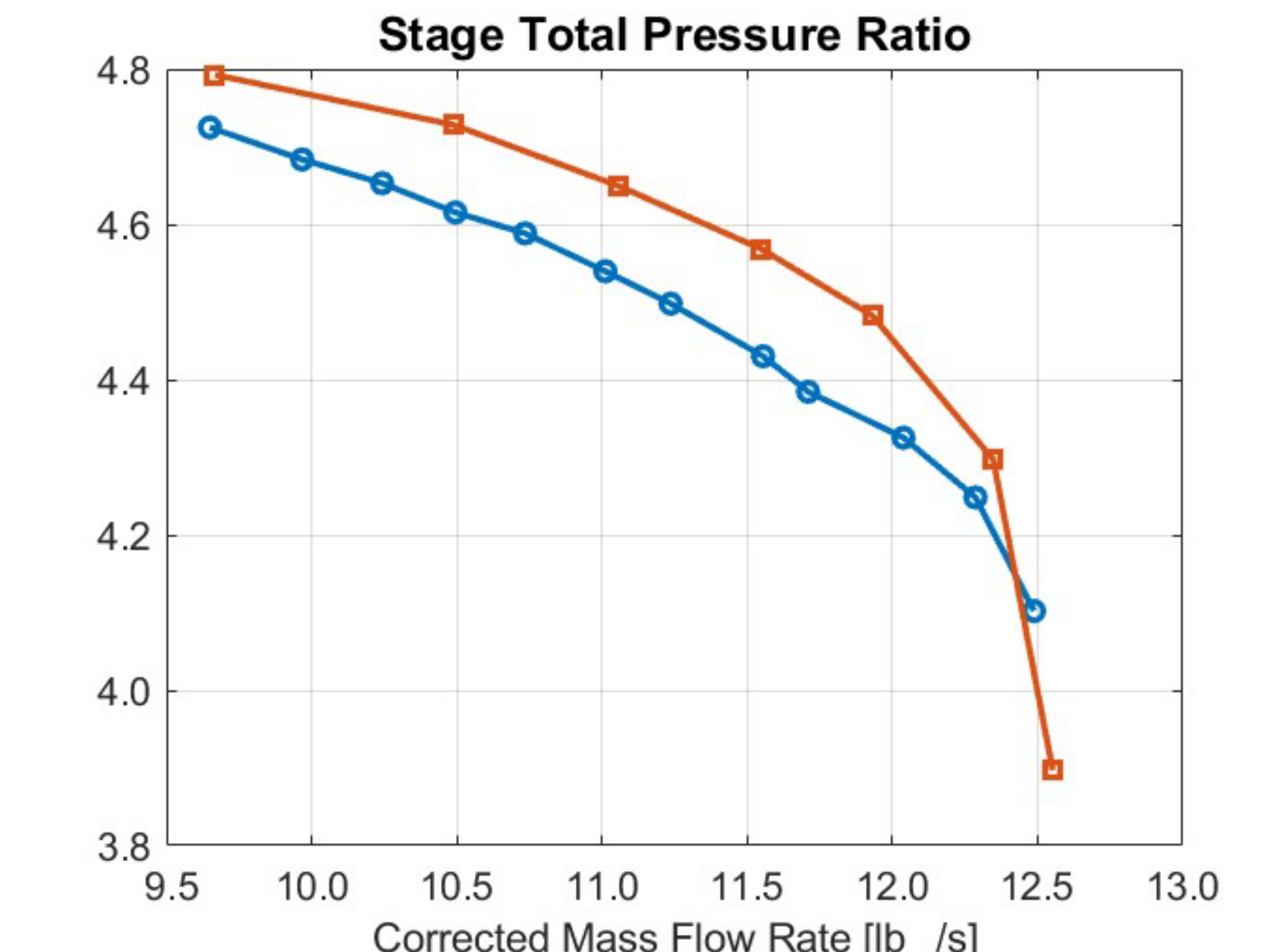
### Impeller Exit Performance

	CFD	Experiment	Percent Error
Total Pressure Ratio	5.143	5.117	-0.5081%
Total Temperature Rise Ratio	0.6544	0.6567	-0.3502%
Efficiency	0.9097	0.8946	-1.688%



### Stage Exit Performance

	CFD	Experiment	Percent Error
Total Pressure Ratio	4.650	4.540	-2.423%
Total Temperature Rise Ratio	0.6493	0.6469	-0.3710%
Efficiency	0.8469	0.8264	-2.481%



Probe values were circumferentially averaged for spanwise stage exit parameters as surveys were not taken at stage exit.

Misprediction of stage exit spanwise total temperature rise ratio is likely due to heat transfer modeling errors.