2023 Utah Space Grant Fellowship Symposium
May 8, 2023, Ogden, UT, USA

ONE-DIMENSIONAL RADIAL TURBOMACHINERY MODELING

Chase Oliphant[1]

[1]Brigham Young University, Provo, UT, USA

ABSTRACT

Centrifugal flow impellers are commonly used in a wide variety of industrial machines. Predicting the performance of these impellers over the entire operating range is key to the system development during the early design stages. The two element in series (TEIS) and two-zone model can be used to accurately predict impeller performance based on flow physics and empirical correlations. Correlations were made with linear regression on a database of 50 pumps and 75 impellers. These correlations were later found to only apply to designs that are similar to those in the database. This paper proposes a new method to generate correlations for the TEIS and two-zone model using reduced order modeling and machine learning. The models will be trained using the same database used in developing the previous correlations, but will also include additional data and computational fluid dynamics (CFD) results. The models will be used to improve the known correlations and to discover new correlations between the machine performance and design variables. Keywords: Turbomachinery, Meanline Analysis, Machine Learning

NOMENCLATURE

C_p	Pressure recovery coefficient
C_{pi}	Ideal pressure recovery coefficient
C_f	Diffuser skin friction coefficient
DR_2	Impeller diffusion ratio, W_{1t}/W_{2p}
V	Absolute Velocity
U	Blade Velocity
W	Relative Velocity
η_a	Impeller inlet effectiveness
η_b	Impeller passage effectiveness
χ	Impeller secondary flow mass fraction
δ_{2p} δ_{2s}	Deviation for primary (2p) or secondary (2s) zones
w_{shaft}	Shaft work
Subscripts	
1	Inlet
2	Outlet
θ	Tangential
p	Primary zone
s	Secondary Zone
t	Tip

1. INTRODUCTION

Centrifugal flow impellers are commonly used in a wide variety of industrial machines, such as rocket turbopumps, refrigeration system compressors, and automotive turbochargers. Predicting the performance of these impellers over the entire operating range is key to the system development during the early design stages. A performance map is a useful tool for determining the pressure rise, mass flow rate, and efficiency for the impeller operating range. A performance map can either be measured experimentally or predicted. Generating accurate predictions of a performance map has proven to be a complicated process. One-dimensional meanline models were developed to give engineers a tool to model the performance of a pump or compressor before computational fluid dynamic (CFD) simulations were widely available. These models use a combination of physical laws and empirical data to calculate the change in fluid properties across the most important streamline. Even with the availability of CFD, meanline models remain a useful tool for preliminary design and design space exploration.

One of these meanline methods, known as the two element in series (TEIS) and two-zone model, has been shown to outperform other meanline models by generating impeller performance maps that more closely resemble results from CFD and experiments [1]. This model has several input parameters that must be accurately specified. Currently, these modeling parameters can only be accurately predicted by engineers with extensive experience in impeller design. This research will focus on creating an improved method for predicting the performance of a centrifugal impeller based on the TEIS and two-zone model using physical conservation laws and empirical data, as well as a large supplementary database of CFD simulations.

2. BACKGROUND

The background section discusses the current understanding of the flow physics through a radial impeller and the techniques that have been used to predict performance maps. This section will also cover several different predictive modeling methods that will be used to complete the research objectives.

2.1 Impeller Flow Physics

The physics of fluid flow through an impeller has been studied for over 90 years. Traditional models were developed by using a control volume analysis through a blade passage, analyzing the conservation of mass and a simplified version of the angular momentum equation, known as the Euler turbomachinery equation [2].

This approach was developed under the assumption that the flow at the inlet and at the outlet is thoroughly mixed with no forms of aerodynamic blockage. Aerodynamic blockage refers to a portion of the blade passage where flow is effectively blocked due to the presence of a low momentum boundary layer or a detached eddy. Dean et al. [4] proved through experiments that this assumption is flawed. They demonstrated that the flow exiting the impeller could be divided into two regions: a high energy jet zone and a low energy wake zone. Japikse built off this research to develop the two-zone model which is shown in Fig. 1. Japikse found that the jet, or primary zone, consists of near isentropic flow, while the wake, or secondary flow, consists of all non-isentropic flow. This discovery uncovered a flaw in the traditional modeling method. The velocity triangles of both outlet zones need to be considered [1].

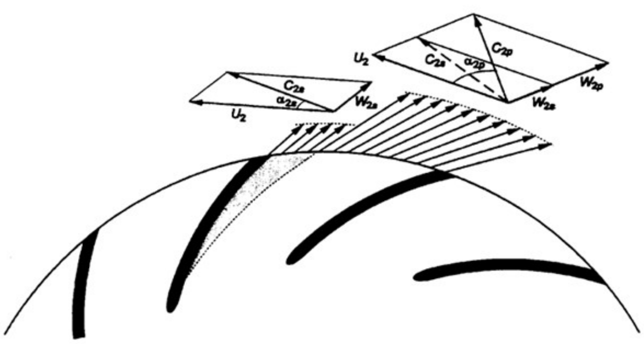

FIGURE 1: A CONCEPTUAL REPRESENTATION OF THE TWO-ZONE MODEL, SHOWING PRIMARY AND SECONDARY VELOCITIES. NO IMPLICATION OF THE PHYSICAL LOCATION OF THE TWO ZONES IS INTENDED BY THIS SKETCH [1].

The state of the primary zone flow is established using 15 equations, with an additional 8 equations establishing the secondary zone flow state. However, there is not an equal number of equations and unknown variables, which makes the two-zone modeling approach much more challenging than single-zone modeling. Despite this difficulty, there is useful simplicity that goes with two-zone modeling. Because the model is based on the actual flow physics, correctly implementing the model will result in a near exact match with results from higher-fidelity methods such as CFD and experiments. Single zone methods solve a simpler set of equations, making them easier to implement, but they neglect the physical conditions that cause the two-zone behavior. They also rely on data measured with a mixed outflow. These weaknesses prevent single-zone models from matching the performance of properly implemented two-zone models [5]. Additionally, many of the correlations, such as the Wiesner slip factor, are only valid within the design space that was used to define them. As new impellers are designed to operate outside of the validated design space with greater efficiency and higher loads, weaknesses in the correlations are being discovered [1].

As previously stated, the governing equations of the two-zone model are unable to be directly solved. Four variables need to be specified to solve the entire system of equations. They are $\delta_{2p}, \delta_{2s}, W_{2p}$, and χ. A downstream parameter describing the flow through the diffuser must also be specified. An impeller can have multiple output flow states depending on the downstream conditions. A parameter that sets the downstream conditions is necessary to accurately predict the impeller outlet flow state. A useful downstream parameter is C_f Given a C_f, an engineer with experience in impeller design can determine $\delta_{2p}, \delta_{2s}, W_{2p}$, and χ based on their understanding of the flow physics through the impeller. It is difficult for newer engineers to accurately predict the parameters. Models that predict $\delta_{2p}, \delta_{2s}, W_{2p}$, and χ will be discussed in later sections.

The Two Element in Series Model

Accurately predicting W_{2p} is important for determining the pressure rise across the impeller. Japikse developed a model to predict W_{2p} by treating the impeller as a series of rotating nozzles or diffusers with two elements that are shown in Fig. 2. The first element (element a) is a model of the impeller inlet as a variable geometry nozzle or diffuser. At high flowrates, the inlet reduces the flow area, acting as a nozzle to accelerate the flow. At low flowrates, the inlet increases the flow area, acting as a diffuser. The second element (element b) models the blade passage from the passage throat to the exit plane of the blades. This element acts as a fixed geometry diffuser. By analyzing the flow through each element, the TEIS model predicts the diffusion of the flow through the impeller. The diffusion is quantified as a diffusion ratio, defined in equation 1. Estimates for DR_2 can be made using equation 2, where C_{pai} and C_{pbi} are known from the geometry and η_a and η_b are set by the model user.

$$DR_2 = \frac{W_{1t}}{W_{2p}} \tag{1}$$

$$DR_2^2 = \frac{1}{1-\eta_a C_{pai}} \frac{1}{1-\eta_b C_{pbi}} \tag{2}$$

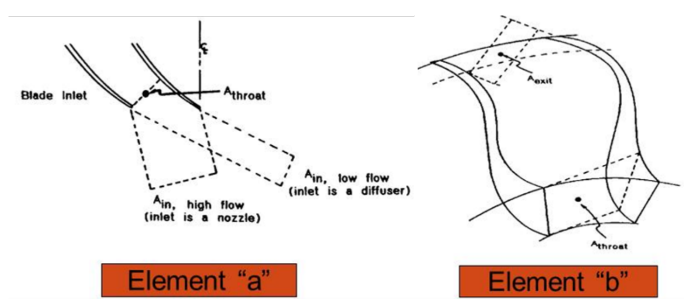

FIGURE 2: A VISUAL DESCRIPTION OF THE TEIS MODEL, SHOWING THE INLET (ELEMENT A) AND THE BLADE PASSAGE (ELEMENT B) [1].

Linear Regression and the Pelton Model

The TEIS and two-zone variables are difficult to predict without results from either CFD or experiments. Working with Japikse, Pelton [6] and Bitter [7] developed predictive models for the two-zone parameters using statistical regression and a database of experimental results. The database they used included data from approximately 125 centrifugal pumps and compressors. The work resulted in a set of equations that produce a reasonable curve fit for the modeling parameters of machines with similar geometric and flow conditions to those in the database. However, performance of machines with parameters outside of the validated range could not be accurately predicted. Pelton concluded that expanding the model's operating range would require more data and further analysis to determine whether additional parameters exist that could impact the model's performance.

Japikse et al. [8] also evaluated the use of neural networks for predicting the two-zone modeling parameters. They developed single layer neural networks for each of the parameters. They found that the models for δ_{2p}, and χ were able to produce reasonable results, but were unsuccessful in creating models for η_a and η_b. They concluded that future work would require more refined input parameters, as well as a larger dataset. They also state that error in the models could be the result of an unknown bias in the training dataset.

2.2 Predictive Modeling Methods

Newer methods for predictive modeling and design space exploration have been developed since Pelton developed his model. These methods, such as reduced order modeling, surrogate modeling, and machine learning, have allowed models to be created with increased accuracy and computational efficiency, as well as better fits for nonlinear datasets.

Reduced Order Models

Reduced-order models (ROMs) reduce the computational complexity of a high order system by projecting the system into a lower order space [9]. The end goal of a ROM is to generate a system solution for an arbitrary set of input parameters, k, using some surrogate model, $R(x)$. This is illustrated in equation 3 where Φ is a set of basis vectors, C is a trained set of expansion coefficients, and $f(k)$ is an interpolation function or surrogate model. The system solution is determined in a lower order space, then transformed into the original, high order space.

$$R(x) = \Phi C f(k) \tag{3}$$

Basis vectors for a system can be determined either through linear or nonlinear data reduction methods [10]. Linear projection methods for data reduction, such as principal component analysis (PCA), reduce the order of complex physical systems with minimal introduced error for linear datasets. These methods have been used in turbomachinery applications [9, 10, 11, 12, 13] to increase modeling accuracy and computational efficiency.

If nonlinearity exists in the dataset, using linear methods like PCA will introduce significant error when the data is reconstructed. Nonlinear methods were developed to generate basis vectors and reconstruct the data without introducing significant error. These methods include kernel principal component analysis (KPCA), isometric feature mapping (ISOMAP), and locally linear embedding (LLE) [10].

Machine Learning

Advances in machine learning have created frameworks that can be used to solve complex physics problems with data gathered through experiments and simulations. A common type of machine learning model is called a multilayer perceptron (MLP). This is a type of feedforward neural network (FFNN). These networks have a series of layers with a certain number of neurons per layer. Each neuron takes an input and performs a simple, linear operation to that input. The output is multiplied by a weight and passed to the next layer of neurons [16]. By repeatedly performing linear operations on the input, chains of composite functions that can recreate the response of nonlinear functions are created. The model is trained by adjusting the weights between each layer using an optimization algorithm. Training these models can be very computationally expensive and require large amounts of data, but the resulting models are highly efficient and accurate. Machine learning models are also limited when modeling out-of-sample scenarios and have the possibility to produce nonphysical predictions [17]. Integrating constraints based on conservation laws has shown promise in addressing these issues. Models trained with physical restraints have been shown to require less data while creating models that perform above the current state of the art. This method has been used to model complex fluid mechanics problems, such as flow across supercritical airfoils using the results from 600 CFD simulations [18], predicting drag force on particle suspensions in moving fluids using data from 5824 particles [17], and accelerating high fidelity CFD simulations using the results from five datasets of direct numerical simulation results [19].

The integration of physical constraints can also be used to overcome other limitations with ML models [18]. Machine learning models are limited when modeling out-of-sample scenarios and have the possibility to produce nonphysical predictions. Kochkov et al. applied conservation constraints to develop a model for accelerating CFD simulations [19]. This approach allowed their model to accurately resolve turbulence at scales outside of the range of the training data. Jia et al. applied conservation of energy when training a model to predict the temperature profiles at different depths in a lake [20]. The resulting model was able to be trained with fewer observed data points while outperforming a state-of-the-art physics-based model.

Machine learning frameworks have also been created to assist in the generation of new, unique data. Generative Adversarial Networks (GANs) are a class of deep learning models that generate new samples from a data distribution [18]. The model has two components: a generator that creates a new sample within the data distribution, and a discriminator that determines if the new sample belongs to the distribution. Models implementing this framework have successfully modeled steady state heat transfer and fluid flow [21]. Wu et al. showed that this model framework could accurately and efficiently generate images of the flow field around supercritical airfoils [18].

3. RESEARCH OBJECTIVES

The main objective of this research is to develop an improved model for predicting the flow state at the exit of a centrifugal impeller based on the TEIS and Two-Zone model using ROM, machine learning, and a sufficiently large database of experimental data and CFD simulations. It is hypothesized that the exit flow state can be predicted with the deviation at the exit blade within 5% of actual behavior and the fraction of primary to secondary flow predicted to also within 5%.

3.1 Create an Improved Model for Predicting Two-Zone Input Requirements

The first objective is to create a ROM for predicting $\eta_a, \eta_b, \delta_{2p}, \delta_{2s}$, and χ with the database used by Pelton and additional data provided by Concepts NREC, a corporate collaborator. Each impeller in the database has a performance map that was found through experiments. There is currently experimental data for 162 different compressors and 73 different pumps. This is 110 more cases than were available to Pelton. The data are of varying fidelity, with some data including pressure and temperature measurements at different stations and traverse pressure measurements along the impeller exit, while other data only include overall pressure rise and flow rates.

An equivalent match to the data using the TEIS and two-zone model will need to be made to determine values for $\eta_a, \eta_b, \delta_{2p}, \delta_{2s}$, and χ. This is done through an optimization algorithm that minimizes the error between the model and the data by varying the values of each modeling parameter. The properties considered in the minimization are total head rise, stage power, and diffusion ratio. The optimizer has been tested on the Eckardt O- Rotor (Eckardt, 1974), a rotor with well documented performance data. The resulting diffusion ratio, head rise, and stage power comparisons are included in Fig. 3. The initial testing is promising, but more testing is needed to ensure that the optimization is robust and can handle additional cases.

Once the database has been generated, a ROM will be constructed. First attempts will use PCA to generate a truncated set of basis vectors and a radial basis interpolation function. We recently performed PCA on the current, uncorrected database and found that 56 basis vectors were required to explain 99% of the variance in the database. The distribution of the total explained variance is shown in Fig. 4. However, reconstructing the reduced-order data introduced significant errors. We hypothesize that this error is being introduced because the dataset is nonlinear. PCA works very well when used on a linear dataset, but introduces significant error when used on nonlinear datasets. It is possible that the corrections we make to the database will improve linearity. If not, nonlinear dimensional reduction techniques, such as KPCA, ISOMAP, and LLE will be implemented.

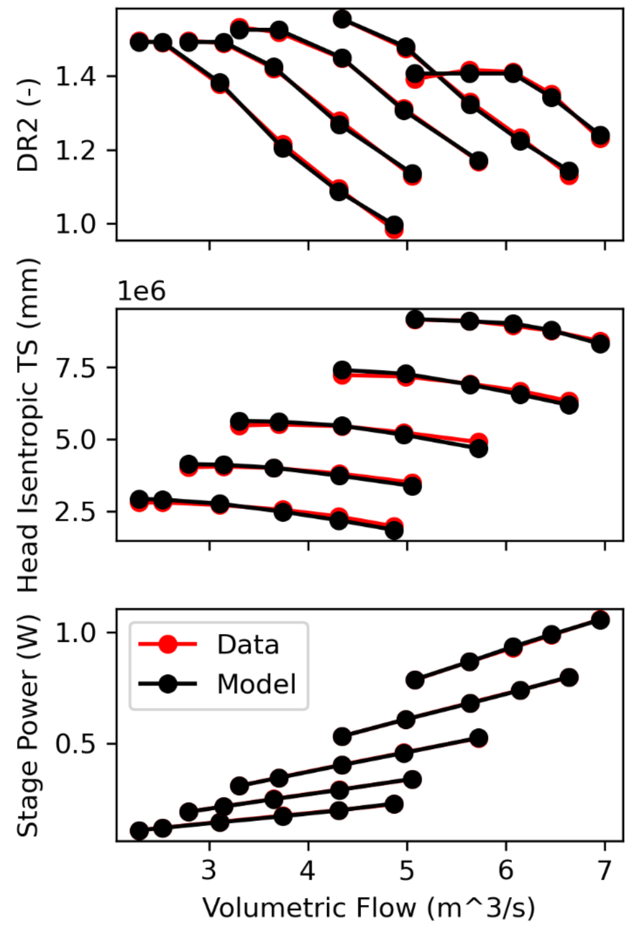

FIGURE 3: A COMPARISON OF THE DIFFUSION RATIO, THE HEAD RISE, AND THE STAGE POWER CALCULATED BY THE MODEL TO THE MEASURED VALUES.

Several different interpolation functions will be used to determine which will perform the best with the given data. These include Kriging, spline interpolation, and radial basis interpolation. The multi-fidelity nature of the data will influence which interpolation functions are tested. Frameworks for creating ROMs with multi-fidelity data have been developed, including Co-Kriging, linear regression-based multi-fidelity models, and Co-RBF.

The ROM input parameters will be determined by finding the covariance between the desired output variables and the potential input variables. The database currently has over 1000 variables associated with each case. Of those 1000 variables, approximately 200 could be reasonably determined based on the impeller geometry, inlet flow conditions, and the desired operating state. Preliminary covariance calculations show that some of the most important potential input variables are inlet relative Mach number, impeller area ratio, blade angles, and Reynold's number.

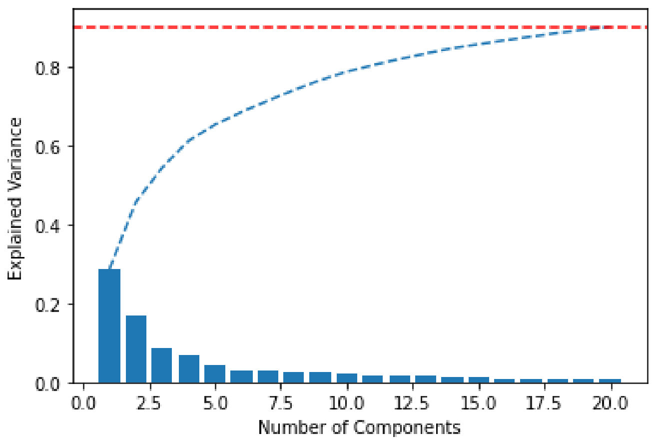

FIGURE 4: THE DISTRIBUTION OF THE EXPLAINED VARIANCE CAPTURED IN EACH COMPONENT GENERATED THROUGH PCA.

After different combinations of dimensionality reduction and interpolation have been explored to create an optimal ROM, a machine learning approach will be taken to create a separate model. FFNNs with a varying number of hidden layers will be trained to predict the desired variables from the input parameters. An initial ML model has been created for η_a using the prior, unexpanded database. The model used the same variables used in the linear regression for model inputs and consisted of 6 hidden layers. This model currently does not perform well compared to the former linear regression models, but this is to be expected given the small dataset used. We expect the model's performance to improve as more data becomes available.

3.2 Validate and Expand the ROM Through CFD

To validate the results of the ROM, steady state CFD simulations will be run on a select number of the new geometries to generate performance maps that can be compared to the results of the ROM. The CFD geometries will be generated by altering five different features of existing geometries to fill gaps in the design space. The five geometry features have will be altered are inlet and exit blade angles, blade thickness, Reynold's number scale, and the number of blades. Each geometry will be simulated at 2-4 different rotation rates and varying pressure differences. The simulations done at the same rotation rate constitute a speedline. The simulations will be done using Star CCM+ and the Fulton Supercomputing Lab. We are currently working on a framework to automatically generate a computational domain, create a suitable mesh, set the boundary conditions, run the simulation, and verify convergence. Values for $\eta_a, \eta_b,\ \delta_{2p}, \delta_{2s},$ and χ will be calculated from the CFD results and compared to the output of the ROM. The CFD results will then be added to the former dataset, the models will be retrained, and the process will be repeated until the ROM outputs are consistently within 10% of CFD. A random set consisting of 10% of the CFD simulations and class 1 or 2 data will be excluded from the training for validation. We estimate that at least 600 speedlines will need to be simulated to properly verify the results of the ROM and to provide enough data for the ML model.

All simulations will be run using Star CCM+ version 2206. Different turbulence models will be considered as the CFD framework is developed. The turbulence model must be able to account for variations due to streamline curvature and rotation effects. Possible candidates include the SST K-ω turbulence model [22] and the Spalart-Allmaras turbulence model with a rotation correction [23]. The simulations will be run in a way that minimizes the effects of non-impeller geometry. Each simulation will use a standard-length inlet and a standard-length vaneless diffuser.

It is not feasible to run a grid-independence study for each of the CFD simulations that will be used to expand the database. Instead, grid independence will be established for a small set of simulations during the development of the CFD framework. The meshing parameters that are found to work best will be applied to all the meshes generated using the framework. Several mesh statistics will be evaluated for each mesh to ensure that each mesh is high quality. The standards were determined based on the best practices outlined by the Star CCM+ documentation [24]. Meshes that do not meet the quality standards will be manually redone. The quality standards are shown in table 1. The Star CCM+ documentation also recommends a wall y+ value less than 5 for turbomachinery simulations with 10-12 prism layers.

TABLE 1: THE MESH STATISTICS THAT WILL BE USED TO DETERMINE THE QUALITY OF EACH MESH.

Statistic	Value	Definition
Volume Change	Greater than 1e-5	The ratio of the volume of a cell to that of its largest neighbor
Skewness Angle	Less than 85°	The angle between the face normal and the vector connecting the two cell centroids
Cell Quality	Greater than 0.1	A function of the geometric distribution of the cell centroids of the face neighbor cells

The CFD framework will be validated by simulating existing geometry and comparing the results to data. Simulations on two impellers have already been done; a low-speed impeller, commonly known as the Eckhardt O impeller [25], and a high-speed impeller. The low-speed impeller has a wheel radius of 7.88 inches, 20 blades, and was simulated at speeds of 10,000 and 12000 rpm. The high-speed impeller has a wheel radius of 1.35 inches, 7 full blades and 7 splitter blades, and was simulated at speeds of 80,000, 100,000, and 120,000 rpm. The quality of the match between the simulation and the data is determined by comparing the average pressure near the shroud at the blade exit (P2). This parameter is very sensitive to the modeling parameters and is often used for CFD validation. The low-speed impeller simulations show good agreement between the resulting flow field and the experimental measurements with the pressure at the blade outlet calculated from CFD being within 5% of the measurements across the entire operating range. This can be seen in Fig. 5. However, the simulations of the high-speed impeller do not match the measured data as well. More work is needed to ensure that simulations run at higher speeds will produce accurate results. This work will include mesh refinement, boundary condition selection, and geometry optimization.

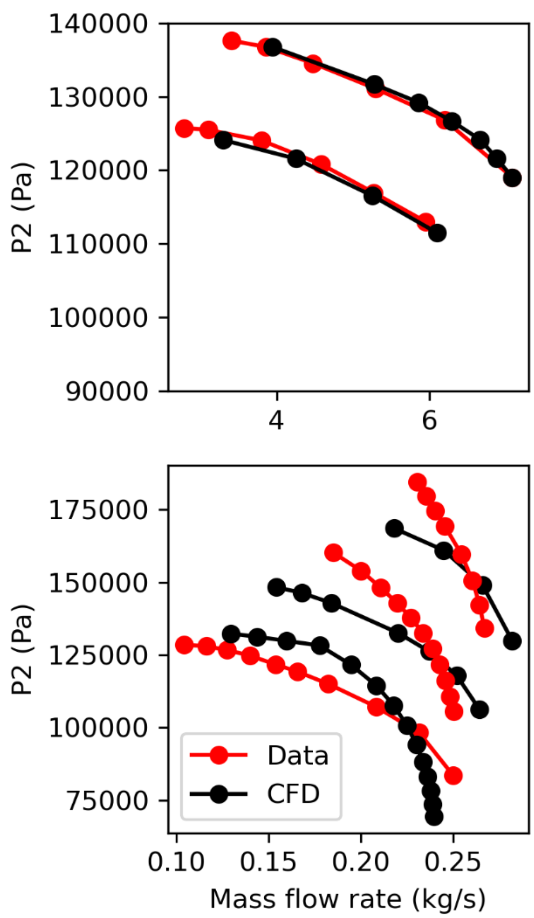

FIGURE 5: THE IMPELLER OUTLET PRESSURE AS A FUNCTION OF MASS FLOW RATE. VALUES SHOWN ARE FROM EXPERIMENTS AND CFD FOR A LOW-SPEED (TOP) AND HIGH-SPEED (BOTTOM) IMPELLER.

In addition to the steady state CFD runs used to expand the model, unsteady CFD will be run to validate the model at different points in the design space. Two to four impeller geometries will be chosen as candidates for the unsteady CFD. A full rotation will be simulated at three varying flow rates: near stall, near choke, and near design. We are also currently in discussions with Concepts NREC to use their test facility to measure additional data for the model validation.

3.3 Perform Real-Time Analysis of the Effect of Design Parameters on the Outlet Flow State

The linear regression model created by Pelton shows the influence that certain impeller design variables have on the two-zone modeling parameters [6]. These include relationships between η_a and inlet Reynold's number, η_b and the impeller exit velocity, χ and the impeller area ratio, and many others. As part of this work, the relationships discovered by Pelton will be validated and improved. Additionally, new correlations will be discovered using the model. Pelton assumed that variations in δ_{2s} did not have any significant effect on the model output, so no correlations were discovered for this variable. We would like to challenge this assumption and develop correlations for δ_{2s}. The highest performing model created in section 4.1 will be used to improve the known correlations and to discover new correlations between the outlet flow state and the impeller design. This will be accomplished by varying certain parts of the impeller geometry, such as blade angles, tip clearance, number of blades, blade thickness, and scale, inputting the new geometry to the model with the same boundary conditions, and comparing the resulting values for δ_{2p}, δ_{2s}, χ, η_a and η_b. The resulting values will be then used in the Two-Zone model to find correlations between the outlet flow state and the variations in design variables. By varying a combination of different geometric variables, over 125 unique geometries could be created from each impeller in the database. We anticipate that the reduced order model will be able to produce results near real-time, making it possible to run thousands of cases over the entire design space. Specific correlations of interest include:

- Exit blade angle and primary zone deviation
- Exit blade angle and secondary zone deviation
- Tip clearance and primary zone deviation
- Tip clearance and blade exit pressure
- Exit blade thickness and machine efficiency

Through this process, a better understanding of how changes to impeller geometry affect the outlet deviation and secondary zone mass fraction, both of which directly affect the power requirements, efficiency, and overall performance of the machine, will be gained.

4. CONCLUSION

Correlations for the TEIS and two-zone models will be improved. A large database of turbomachinery results will be generated through processing experimental data and supplementing with CFD results. The data will be used to train reduced-order and machine learning models that will predict the values of the required modeling parameters based on geometry and flow conditions.

ACKNOWLEDGEMENTS

The author extends a special thanks to the Utah NASA Space Grant Consortium and to Concepts NREC for funding this work. The author would also like to thank Dr. Steven Gorrell and Dr. Daniel Maynes for their help and guidance in performing this research.

REFERENCES

[1] D. Japikse, Centrifugal Compressor Design and Performance., Concepts ETI, 1996.

[2] A. L. Gerhart, J. I. Hochstein and P. M. Gerhart, Munson, Young and Okiishi's Fundamentals of Fluid Mechanics, 9th Edition, Hoboken: John Wiley & Sons, 2021.

[3] R. H. Aungier, Centrifugal Compressors: A Strategy for Aerodynamic Design and Analysis, ASME Press, 2000.

[4] R. C. Dean and Y. Senoo, "Rotating Wakes in Vaneless Diffusers," *Journal of Basic Engineering,* 9 1960.

[5] D. Japikse and K. N. Oliphant, "Turbomachinery Modeling: Explicit and Impliit Knowledge Capturing," in *ASME Turbo Expo 2005: Power for Land, Sea and Air*, Reno, 2005.

[6] R. J. Pelton, "One-Dimensional Radial Flow Turbomachinery Performance Modeling," 2007.

[7] J. J. Bitter, "One-Dimensional Performance Modeling of Centrifugal Flow Vaned Diffusers," 3 2007.

[8] R. Japikse, D. Japikse and R. Pelton, "An evaluation of neural networks for meanline model development," *Concepts ETI,* 2004.

[9] T. J. Blanc, M. R. Jones and S. E. Gorrell, "Reduced-Order Modeling of Conjugate Heat Transfer Processes," *Journal of Heat Transfer,* vol. 138, no. 5, 5 2016.

[10] G. D. Bird, S. E. Gorrell and J. L. Salmon, "Dimensionality-reduction-based surrogate models for real-time design space exploration of a jet engine compressor blade," *Aerospace Science and Technology,* vol. 118, p. 107077, 11 2021.

[11] S. Bunnell, S. Gorrell and J. Salmon, "Design Space Exploration of Rotor Blades Accounting for Vibratory Responses by Indirect Emulation," *Journal of Computing and Information Science in Engineering,* vol. 21, no. 4, 8 2021.

[12] R. A. Spencer, S. E. Gorrell, M. R. Jones and E. P. Duque, "Analysis and comparison of inlet distortion flow physics at design and near stall operating condition using proper orthogonal decomposition," *52nd AIAA/SAE/ASEE Joint Propulsion Conference, 2016,* 2016.

[13] T. J. Blanc, S. E. Gorrell, M. R. Jones and E. P. Duque, "Analysis and compression of time-accurate turbomachinery simulations using proper orthogonal decomposition," *49th AIAA/ASME/SAE/ASEE Joint Propulsion Conference,* vol. 1 PartF, 2013.

[14] R. Halder, K. Fidkowski and K. Maki, "Local Non-Intrusive Reduced Order Modeling Using Isomap," *AIAA SCITECH 2022 Forum,* 1 2022.

[15] A. I. J. Forrester, A. Sóbester and A. J. Keane, Engineering Design via Surrogate Modelling, John Wiley & Sons Ltd., 2008.

[16] S. T. Kelly and B. I. Epureanu, "Data-Driven Reduced-Order Model for Turbomachinery Blisks with Friction Nonlinearity," pp. 97-100, 2023.

[17] J. Willard, X. Jia, S. Xu, M. Steinbach and V. Kumar, "Integrating Scientific Knowledge with Machine Learning for Engineering and Environmental Systems," 3 2020.

[18] H. Wu, X. Liu, W. An, S. Chen and H. Lyu, "A deep learning approach for efficiently and accurately evaluating the flow field of supercritical airfoils," *Computers & Fluids,* vol. 198, p. 104393, 2 2020.

[19] D. Kochkov, J. A. Smith, A. Alieva, Q. Wang, M. P. Brenner and S. Hoyer, "Machine learning–accelerated computational fluid dynamics," *Proceedings of the National Academy of Sciences,* vol. 118, no. 21, 5 2021.

[20] X. Jia, J. Willard, A. Karpatne, V. Tech, J. S. Read, J. A. Zwart, U. S. G. Survey, V. Kumar, ;. J. Willard, M. Steinbach, V. Kumar, ;. A. Karpatne and M. Steinbach, "Physics-Guided Machine Learning for Scientific Discovery: An Application in Simulating Lake Temperature Profiles," *ACM Transactions on Data Science,* vol. 2, no. 3, 2021.

[21] A. B. Farimani, J. Gomes and V. S. Pande, "Deep Learning the Physics of Transport Phenomena," 2017.

[22] "Curvature and Rotation Sensitive Variants of the K-Omega SST Turbulence Model," in *Fluids Engineering Division Summer Meeting, vol. Volume 1: Symposia, Parts A, B and C*, 2009.

[23] M. L. Shur, M. K. Strelets, A. K. Travin and P. R. Spalart, "Turbulence Modeling in Rotating and Curved Channels: Assessing the Spalart-Shur Correction," 2000.

[24] Siemens Digital Industries Software, "Simcenter STAR-CCM+ User Guide v. 2206," Siemens, 2022.

[25] S. L. Brunton and J. N. Kutz, Data-Driven Science and Engineering, Cambridge University Press, 2019.