Turbocharging Seminar 2021

17-18 September 2021, Dalian, China

Using AI for Loss Prediction in a Hybrid Meanline Modelling Method to Deliver Improved Turbocharger Map Prediction

Pangbo Ren[1,*], Charles Stuart[1], Stephen Spence[1] ,
Ryosuke Inomata[2], Takayuki Kobayashi[2], Isao Morita[3]

1. *Trinity College Dublin, The University of Dublin, College Green, Dublin, Ireland*
2. *IHI Charging Systems International GmbH, Heidelberg, Germany*
3. *IHI Corporation, Yokohama, Japan*

**Corresponding author. renp@tcd.ie*

Abstract

Meanline modelling approaches remain attractive due to an unrivalled ability to predict full turbine performance maps quickly compared to high fidelity approaches such as CFD, especially in the preliminary design process. As improvements in performance on a component level approach a point of diminishing returns, the ability to efficiently optimise the complete charging system for a given duty is a topic that is attracting significant research interest. In order to achieve this aim, the current piece of work seeks to combine the advantages of machine learning techniques and physical meanline modelling to facilitate faster, more accurate predictions of complete turbocharger maps.

This paper presents a novel methodology for turbocharger turbine rotor performance prediction based on hybrid modelling. The radial turbine rotor for turbocharger applications is parameterized in order to conduct CFD simulations for a variety of turbine geometries. The results of the CFD simulations make up a database to be fed into an artificial neural network, by which the rotor losses can be predicted when a specific geometry and operating condition are given. The predicted losses are then utilized in the meanline code, substituting the conventional loss models to predict the turbine rotor performance. One novel aspect of this work is that the losses occurring in the rotor are split into two portions, before and after the throat respectively. Another novel aspect is that the blockage level was implemented as a variable value in this meanline model to reflect the changing secondary flow field at the rotor throat at different operating conditions. This hybrid meanline modelling method is evaluated on several unseen test cases and shows good agreement with the CFD results from the perspective of total-to-static efficiency and mass flow rate. The hybrid meanline modelling method displays great potential in wide range radial turbine performance prediction with enhanced accuracy in comparison to traditional approaches.

Keywords

artificial intelligence, radial turbine, wide range, performance, loss models, turbocharger

Nomenclature

AI	Artificial intelligence
ANN	Artificial neural network
b	Blade height (m)
CFD	Computational Fluids Dynamics
C_p	Specific heat capacity at constant pressure (J/(kg·K))
K	Loss coefficient
L	Loss
LE	Leading edge
Ma	Mach number
MFP	Mass flow parameter
N_{bl}	Blade number
P	Pressure (Pa)
PR	Pressure ratio
r	Radius (m)
t	Thickness (m)
TE	Trailing edge
U_7	Blade tip speed (m/s)
U/C	$\frac{U_7}{\sqrt{2C_pT_{01}\left[1-\left(\frac{P_{11}}{P_{01}}\right)^{\frac{\gamma-1}{\gamma}}\right]}}$
w	Relative speed (m/s)
x	x-coordinate, axial location
X	Streamwise location of the points at the blade angle Bezier curve
y	y-coordinate, radial location
Y	Blade angle value of the points at the blade angle Bezier curve
Z	Axial distance from LE to TE (m)
η	Efficiency
β	Relative flow angle (◦)
ε	Clearance (mm)

Subscripts

1	Housing inlet
2	Housing throat
3	Housing outlet
4	Nozzle inlet
5	Nozzle throat
6	Nozzle outlet

7	Rotor inlet
8	Rotor inlet after incidence loss
9	Rotor throat
11	Rotor outlet
13	Diffuser outlet
B	Second point of blade angle Bezier curve
c	Clearance
C	Third point of blade angle Bezier curve
D	Fourth point of blade angle Bezier curve
inc	Incidence
geom	Geometrical
H	Hydraulic
m	Mean
opt	Optimum
p	Passage
P	Control point of hub meridional profile Bezier curve
Q	Control point of shroud meridional profile Bezier curve
r	Radial direction
s	Isentropic process
te	Trailing edge
ts	Total-to-static
tt	Total-to-total
x	Axial direction

1. Introduction

There is a continuing need for the rapid, reliable and accurate generation of overall performance predictions for turbomachinery design work, especially for turbocharger design, in which overall performance rather than primarily design point operation is the key consideration. In the case of turbocharging applications, existing engine and powertrain simulations require turbine maps to calculate the turbine performance, which are usually obtained from experimental testing. Unfortunately, the need for extrapolation is unavoidable because of the dearth of the testing data, leading to inaccuracies especially at off-design conditions. Therefore, low fidelity turbine and compressor models need to be incorporated into existing one-dimensional engine simulation tools to enable intensive modelling and optimisation of complete vehicle powertrains for different drive cycles using turbine models that allow for geometrical changes with some physical modelling.

A low fidelity turbine modelling method should deliver a rapid solution and require the minimum amount of geometric data. These aims are typically achieved through the use of empirical loss models calibrated against experimental characteristics. The meanline method is especially advantageous in the preliminary design process, when detailed three-dimensional blade and flow channel profiles have not been determined. Meanline turbine modelling has been possible for many years [1][2], although the conventional empirical loss models require calibration for specific turbine designs and are not robust at off-design operating conditions. Vehicle powertrain models currently predict performance using interpolated and extrapolated turbine and compressor maps, with inevitable uncertainty [3][4]. Improved powertrain optimisation methods need to be able to alter the turbocharger geometry and predict the impact on compressor and turbine maps through physically based models in order to achieve a fully optimised solution. The current meanline models are not able to give sufficiently accurate performance prediction across a wide range of operating conditions. Improvements in meanline modelling are specifically needed to allow better calibration of the loss models for extreme off-design performance prediction, which is important for modelling pulsed flow and for vehicle transient performance [5]. In recent years, research has been conducted to extend the meanline method into mixed flow turbine applications [6][7], variable geometry turbine applications [8][9], turbine pulsating flow modelling [10][11], twin-scroll turbine applications [12] and so forth. However, the meanline modelling method always requires a calibration process for the loss models by comparing with experimental or high-fidelity numerical results at design point. The reason behind this is the limited accuracy and the limited range of applicability of the conventional loss models.

It is unrealistic to expect models that are essentially based on the assumption of one-dimensional flow to be able to accurately represent the losses arising from the complex three-dimensional flow field in a real turbine, particularly when dealing with simulations across a wide operating range. Loss modelling inevitably results in simplifications which have to be compensated for by the introduction of empirical data from turbine tests. Many different loss modelling systems are in use by turbine designers. Probably the most widely used is based on the work of NASA. The NASA models [1][13] were originally published in the 1960s and later expanded upon, but were actually based on a very limited number of radial turbine designs originally developed for a single application of a closed-cycle gas turbine. Commonly, rotor loss models applied in the meanline method can be classified into incidence loss, passage loss, tip clearance loss, trailing edge loss and windage loss. The individual loss models are usually configured to be related to flow kinetic energy with different empirically derived coefficients.

While surrogate based artificial intelligence (AI) approaches offer the ability to include a lot of data in the metamodel training, the lack of fundamental flow physics can prove problematic from a design perspective. By comparison, while one-dimensional meanline modelling incorporates the necessary physics, the loss models employed can be limited in application and frequently rely on designer experience to generate satisfactory results. By utilising a hybrid methodology whereby the loss used in the meanline model is predicted through an AI surrogate technique trained using an extensive CFD database, the current work seeks to deliver improvements in both speed and accuracy of prediction of complete performance maps.

Structure of the paper

The main objective of this research is to explore the possibility and potential of a hybrid meanline modelling technique using AI / machine learning for loss prediction to deliver wide range turbine performance map prediction with an acceptable trade-off between accuracy and calculation time.

To begin with, this paper introduces the experimentally validated turbine rotor CFD simulation study method. The meanline modelling is presented afterwards, followed by a brief introduction to the AI method adopted in this study. The rotor geometry parameterization and loss extraction methods are then described, followed by presentation and discussion of the results generated by the novel hybrid meanline model. Finally, the conclusions are drawn to summarize the capability of the new hybrid meanline method.

2. Experimentally validated CFD method

A numerical simulation model was set up using the commercial CFD code ANSYS CFX 19.2. A steady-state model comprising the full rotor blade passages (360°) of a turbocharger turbine stage was employed, as is shown in Figure 1. The Shear Stress Transport (SST) turbulence model was used due to its numerical stability over a range of operating conditions and its effectiveness for predicting turbomachinery flows [14]. Structured hexahedral grids for the rotor and diffuser and the extended sections at inlet and outlet were created using TurboGrid. An unstructured mesh was created in Ansys ICEM for the volute component. A grid independence study was conducted to ensure that the simulation was independent of the grid refinement. This resulted in a mesh cell count for the volute and rotor passages and inlet and outlet sections was 16.1 million. The y+ value was maintained below five in the rotor domain and at less than ten in the volute domain. A frozen rotor interface (multiple reference frame) was utilized at the interface of the rotating and stationary components.

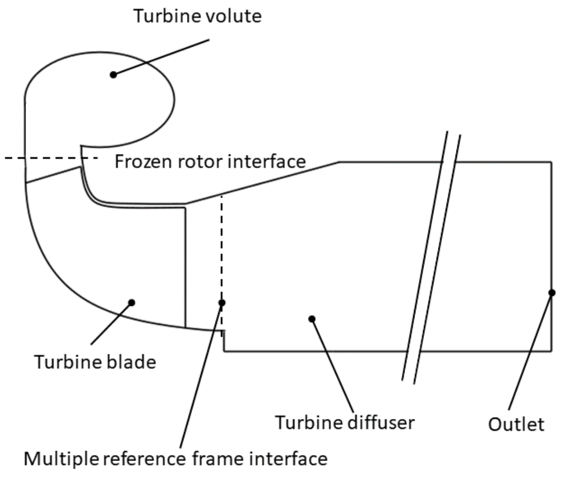


Figure 1 Full passages CFD model

All wall boundary conditions were modelled as being no-slip and adiabatic, and the fluid was defined as 'Air Ideal Gas'. Stagnation pressure and temperature were defined at the inlet while the static pressure was given at the outlet.

Experimental validation of the numerical approach was conducted using a turbine dynamometer test facility. Efficiency calculations were based on temperature and pressure measurements taken upstream of the turbine inlet and the at the outlet section. The turbine dynamometer allowed a much wider range of turbine operating conditions to be tested than is possible on a conventional hot gas stand.

The corrected MFP and efficiency predictions from the CFD simulation are compared with the experimental measurements at medium rotating speed with different inlet stagnation pressure in Figure 2. These data are chosen for clarity of presentation as the data is representative of the correlation achieved across the operating range. The experimental and numerical efficiencies and mass flow parameters achieved good agreement and followed similar trends. Overall, the maximum error of mass flow parameter and efficiency is 1.47% and 1.3% respectively. With a very good agreement, this CFD setup is considered sufficiently validated by experimental test data.

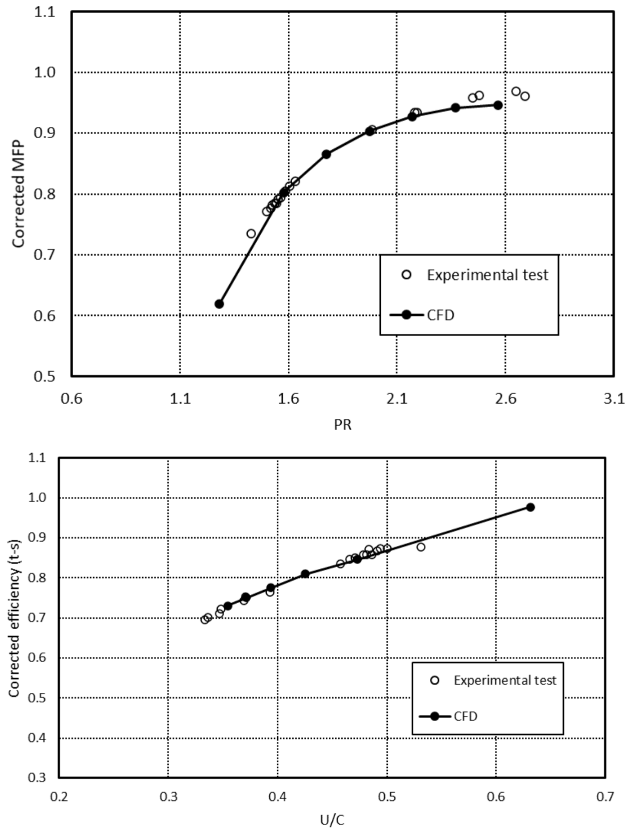


Figure 2 CFD results compared with experimental test

3. Meanline method

In the meanline method, the blade-to-blade and meridional variations of the flow pattern are inevitably neglected so that any turbomachinery device can be considered as a set of ducts in which the basic geometrical parameters are defined. The meanline method is a very powerful tool to predict the turbine performance because of the capability to simulate a complete turbine map in seconds which would take hours even with a very coarse mesh in CFD on comparable computing resources.

In this study, the flow path of the turbine stage was divided into 13 stations from the housing inlet to diffuser outlet, as is shown in Figure 3, which incorporates more calculation stations that most conventional meanline models so as to allow for more implementation of the more comprehensive loss and blockage models.

The reliability of the iteration method used to obtain a converged mass flow rate at a given operating point is fundamental to the success of the model. The method employed in the current work is based on [2] and ensured a converged solution for a wide range of flow conditions. It should be noted that turbines can operate at extremely high expansion ratios, resulting in choking at one or more stations within the stage. Therefore, the expansion ratio is a very sensitive function of mass flow rate close to the choke condition. In the iterative scheme, the exit static pressure was used to influence assumed values of static pressure at the blade trailing edge station. This value of static pressure was then used in turn to compute local values of mass flow rate in the iteration process until the mass flow rate at each station converged upon a single value. The architecture of the meanline is illustrated in Figure 4. More detailed information of the iterative calculation method can be found from [2].

Loss models are essential for meanline model performance prediction. A wide range of loss models and correlations is available in the published literature. The rotor losses are generally divided into the following categories: passage loss, incidence loss, tip clearance loss, trailing edge loss and windage loss. Loss models adopted in the meanline model here are listed as followings:

Incidence loss [1]:

$$L_{inc} = \frac{K_{inc} w_7^2 \sin(i_7)^2}{2} \tag{1}$$

Where

$$i_7 = \beta_7 - \beta_{7,opt}, \tag{2}$$

And

$$\tan\beta_{7,opt} = \frac{-1.98\tan\alpha_7}{N_{bl}(1-1.98/N_{bl})} \tag{3}$$

Passage loss [15]:

$$L_p = K_p \left\{ \left(\frac{L_H}{D_H}\right) + 0.68 \left[1 - \left(\frac{\bar{r}_{11}}{r_7}\right)^2\right] \frac{\cos\beta_{b11}}{\left(\frac{b_{11}}{l_c}\right)} \right\} \frac{1}{2} (w_7^2 + w_{11}^2) \tag{4}$$

Tip clearance loss [15]:

$$L_c = \frac{U_7^2 N_{bl}}{8\pi} \left(K_x \varepsilon_x C_x + K_r \varepsilon_r C_r + K_{xr} \sqrt{\varepsilon_x \varepsilon_r C_x C_r}\right) \tag{5}$$

Trailing edge loss [16]:

$$L_{TE} = \frac{\rho_{11} w_{11}^2}{2} \left[\frac{N_{bl} t_{11m}}{\pi r_{11m} \cos\beta_{11}}\right]^2 \tag{6}$$

The input information to the model comprised of one-dimensional turbine geometry, pressure ratio, stagnation temperature at inlet, rotational speed, exit static pressure and gas properties.

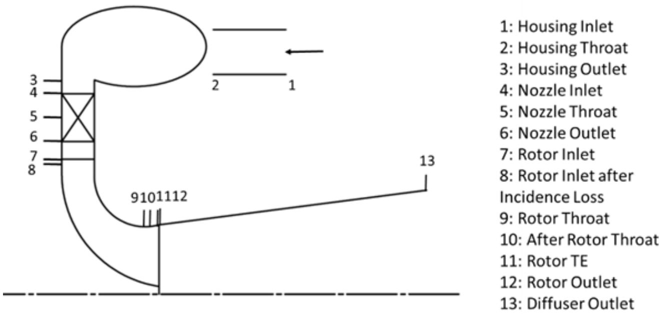


Figure 3 Turbine flow path

The meanline model was coded using the Python 3 computer language. The input data was saved in a Microsoft Excel file with several sheets including geometry, operating conditions and loss model coefficients. The output was also a Microsoft Excel file with two sheets, one for overall performance and a second covering detailed results at each station for every operating condition point.

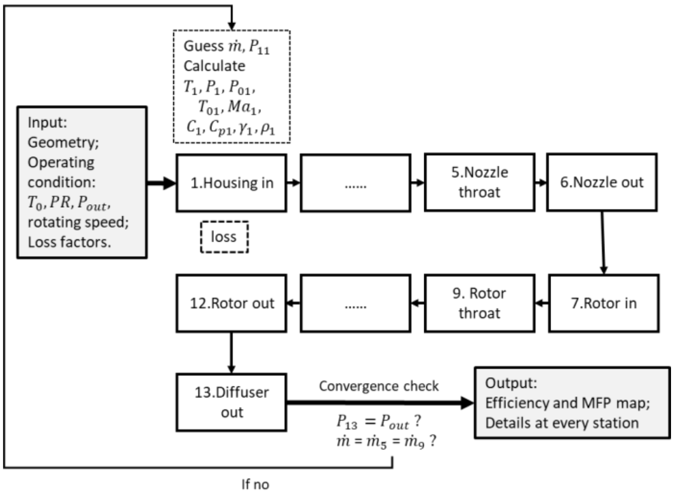


Figure 4 Meanline model structure

4. Hybrid model

In a conventional meanline method, the contribution of individual sources of loss throughout the stage are calculated by several separate loss models during the iteration process, as discussed in Section 3. The problem with this approach is that the loss coefficients must be tuned for a given family of turbines through use of testing data to give satisfactory performance prediction. For turbines without testing data, the accuracy of the conventional meanline method cannot be ensured. However, the lack of wide range testing data for radial turbines is a well-known limitation for the low fidelity model development and validation. This problem arises due to the range of stable compressor operation limiting the range of turbine power absorption that can be achieved at any given speed on a turbocharger hot gas test stand. Therefore, there is low confidence in calibrating and using the meanline method for predicting the performance map for wide range applications, especially during the preliminary design process. Furthermore, it is difficult to separate the losses out into individual, independent headings when they are all interrelated to a greater or lesser extent.

Machine learning is a branch of artificial intelligence (AI) which focuses on the use of data to imitate the way that humans learn. It can be used to make predictions by uncovering key insights within large data sets. Applications of machine learning have been proposed in turbomachinery design and optimization [17][18]. Artificial neural network (ANN) is just one of several possible methods of machine learning to train a surrogate model. The motivation for choosing ANN as the method to train an AI model is that ANN has been widely used for many years in a variety of applications, so that the robustness has been validated and the related training experiences are available [19][20]. Besides, the Keras module in Python provides a user-friendly environment for ANN model development.

In this section, the ANN-meanline hybrid model is introduced to explore an accurate, fast, and wide-ranged turbine performance prediction model. The ANN was configured to be trained by CFD simulation data and then predict the loss for a certain turbine geometry at a given operating point. Large numbers of CFD cases were conducted to set up the ANN training database. Different from the conventional meanline method, losses were presented as two packages rather than several conventional loss models. Therefore, there is no need to tune the loss coefficients and the accuracy of performance prediction is independent from the loss coefficients.

4.1. Summary of AI method

For performance prediction in high-dimensional problems, neural networks are employed using regression techniques to approximate the highly non-linear functions. An artificial neural network consists of an input layer, a number of hidden layers and an output layer. There are some neurons in each layer. The non-linearity is created by introducing complex non-linear activation function of each neuron. The learning process is actually driven by seeking to minimize the error throughout the entire function from input layer to output layer. The process is monitored or evaluated via a loss function, which is analogous to the cost function of a traditional optimization process. (Note that this is a measure of the accuracy of the numerical process and does not physically relate to the thermodynamic losses in the turbine stage.) The performance of an ANN is highly dependent on the tuning parameters, which are learning rate, the number of hidden layers, the number of neurons, the activation function and the loss function.

4.2. Rotor geometer parameterization

The ANN training database should contain diversity of turbine geometries and operating points in order to effectively predict the performance of unseen turbines. To begin with, the turbine geometry needs to be parameterized.

DesignModeler in ANSYS 19.2 was utilized to design rotor blades. Blade angle distribution was described by a cubic Bezier curve that was defined by four points, as illustrated in Figure 5. With point A fixed at the LE (leading edge) of the rotor blade, the curve could be determined by the other three points, where five parameters X_B, Y_B, X_C, Y_c, Y_D are set up. X_C was fixed constant at the 70% streamwise location of the blade. Four parameters were chosen to describe the blade thickness at hub and shroud side, LE and TE (trailing edge) respectively.

As is depicted in Figure 6, the meridional profile was also parameterized by the blade height at the LE, b_7, hub and shroud diameter at the TE, D_{hub11} and $D_{shroud11}$, rotor axial length Z. The hub and shroud profiles were both defined as cubic Bezier curves. To reduce the number of parameters, some correlations were assumed:

$$x_{P_{II}} = x_{P_I} \tag{7}$$
$$y_{P_{III}} = y_{P_{IV}} = D_{hub11}/2 \tag{8}$$
$$x_{QII} = x_{QI} \tag{9}$$
$$y_{Q_{II}} = y_{Q_{III}} = y_{Q_{IV}} = D_{shroud11}/2 \tag{10}$$
$$y_{P_I} = y_{Q_I} = D_7/2 \tag{11}$$
$$y_{P_{II}} = \left(y_{P_I} + y_{P_{IV}}\right) * 40\% \tag{12}$$
$$x_{P_{III}} = \left(x_{P_I} + x_{P_{IV}}\right) * 50\% \tag{13}$$
$$x_{Q_{III}} = \left(x_{Q_I} + x_{Q_{IV}}\right) * 50\% \tag{14}$$

In addition, shroud tip clearance and blade number were also considered. The blade geometrical parameters are shown in Table 1.

There were 14 geometry parameters in total. Four operating condition parameters were also defined, namely inlet stagnation pressure P_{0in}, inlet stagnation temperature T_{0in}, flow direction and rotor rotational speed *N*. Therefore, 18 variable parameters were used to investigate the design space in this study. This equates to a high degree of dimensionality and therefore the number of specific values for each dimension was limited in this study. Three rotational speed values were chosen to cover the low, medium and high-speed operating conditions. Six different pressure ratios were chosen as the sampling points for each speed line to cover a wide U/C range. Static pressure at outlet was held constant at 101325 Pa.

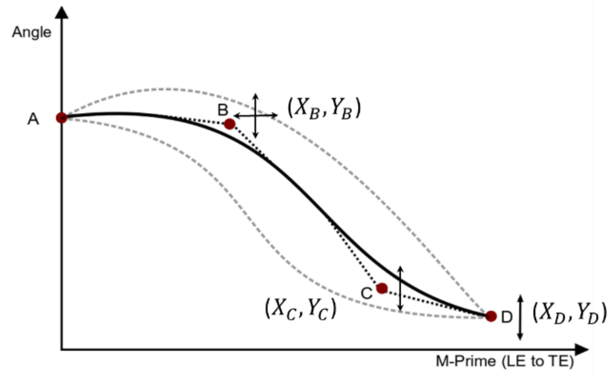


Figure 5 Blade angle distribution defined by a cubic Bezier curve

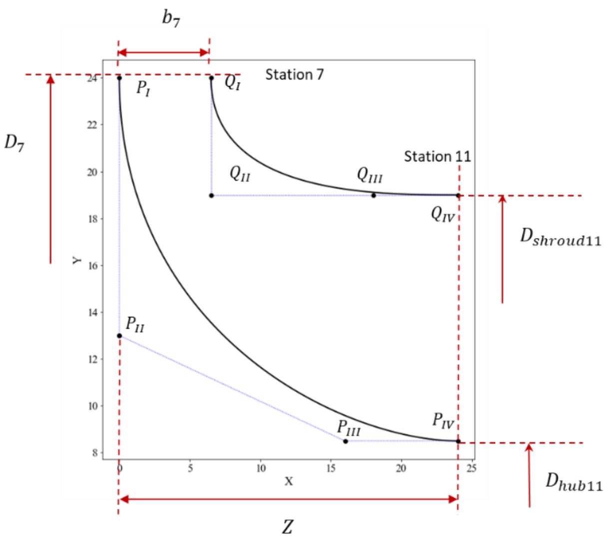


Figure 6 Meridional profile definition

One geometry within the design space was chosen as the 'baseline case' and then different turbine rotor geometries were obtained by adjusting the various parameters detailed in Table 1. The parameters of baseline case were defined as '1' and the range was non-dimensionalized. The sampling method adopted here is based on the concepts of HDMR (high-dimensional model representation), detailed information of which can be found in [21]. As a result, a total of 75 turbine geometries were generated. For this study, the geometries were within the range of typical automotive turbocharger designs, but the method could be extended to include less conventional designs. A selection of the rotors is presented in Figure 7.

Table 1 Blade geometry parameters

Parameter	Range
b_7	(-23.7%, +29.5%)
D_{hub11}	(-38.0%, +38.7%)
$D_{shroud11}$	(-14.4%, +7.0%)
Z	(-53.1%, +63.3%)
Blade number	(-20%, +10%)
Tip clearance	(-45.5%, +45.5%)
X_B	(-60%, +60%)
Y_B	(-100%, +100%)
Y_C	(-100%, +100%)
Y_D	(-33.3%, +33.3%)
Thickness_hub_LE	(-26.3%, +26.3%)
Thickness_hub_TE	(-16.3%, +16.3%)
Thickness_shroud_LE	(-27.3%, +27.3%)
Thickness_shroud_TE	(-17.6%, +17.6%)

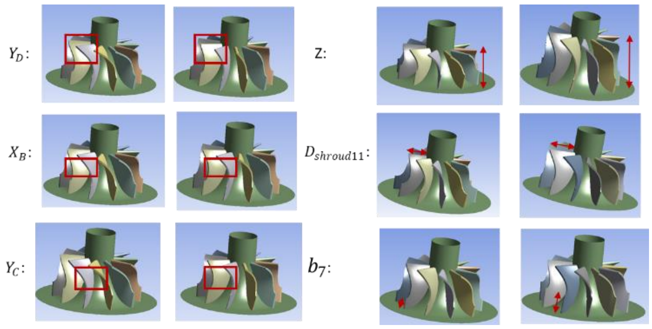


Figure 7 Examples of different rotor designs due to varying geometry parameters

4.3. Numerical model

A simplified CFD model was developed based on the validated CFD model in Section 2. The CFD model of the rotor only utilized a steady-state single passage with periodic boundary conditions, as shown in Figure 8. Similarly, the SST turbulence model was used and structured hexahedral grids for the rotor were created using TurboGrid. A target y+ value less than five was assigned to maintain the accuracy of the boundary layer flow simulation. The grid topology also followed the same method used for the validation of the full stage model in Section 2. A mixing-plane interface was adopted at the interface of the rotating and stationary domains. A flow direction was specified as an inlet boundary condition. To validate the efficacy of the simplified approach, the performance calculation from the simplified model was compared with the full-passages model that was described in Section 2. Inflow direction, stagnation temperature and pressure were extracted from the full stage model and then used as boundary conditions for the simplified model. It turns out that the simplified model could predicted the rotor efficiency with a deviation of less than 0.2%pts compared to the full-passage model, which verified that a good agreement was obtained.

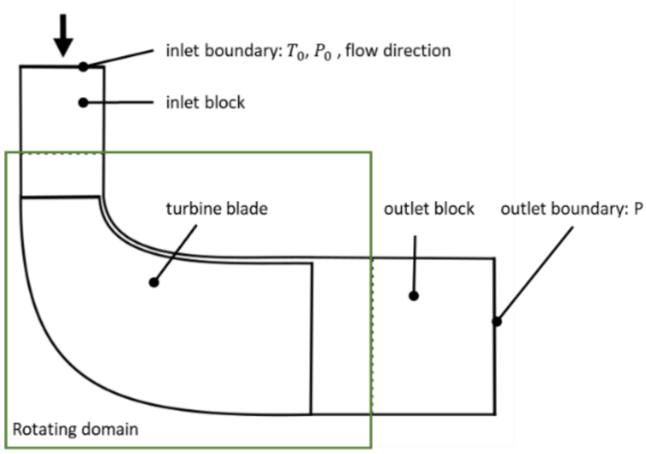


Figure 8 Simplified rotor-only CFD model

4.4. Hybrid meanline model

In a conventional meanline method, the contribution of individual sources of loss throughout the stage are calculated by a group of loss models during the iteration process, as discussed in Section 3. The process to tune the coefficients is always essential to give good performance prediction. However, in the current work, losses incurred in the rotor could be calculated directly from CFD, then predicted by ANN.

Turbine isentropic efficiency is defined by

$$\eta_{tt} = \frac{h_{01}-h_{013}}{h_{01}-h_{013ss}} \tag{15}$$

$$\eta_{ts} = \frac{h_{01}-h_{013}}{h_{01}-h_{13ss}} \tag{16}$$

Where the isentropic final enthalpies h_{013ss} and h_{13ss} are the values obtained in an isentropic expansion to the same final stagnation and static pressures as the actual process. The enthalpy deviation between the isentropic process and the real expansion process was used to quantify the loss in this study. If only the rotor part is investigated (without exhaust diffuser), the loss is then the difference between h_{011ss} and h_{011ss}, as is shown in Figure 9.

In conventional loss models, losses are often correlated to the relative velocity at different stations, in which the different categories of losses are added in successively. However, in the real flow field the losses are highly interdependent. For example, the occurrence of separation due to incidence at the LE of the rotor would affect the downstream pressure field and the development of the tip leakage flow and subsequent leakage vortex. A meanline model cannot capture these three-dimensional flow features, but it is clear that the various conventional categories of losses in the rotor are not independent of each other. In this approach, the rotor losses were not broken down into different categories but were collected as a single package. The meanline calculation method adopted in this study reached a converged mass flow rate by comparing the mass flow at the throat and other stations through the stage, as discussed in Section 3. Since the loss estimation affects the mass flow calculation, the overall rotor loss was split into two parts and then introduced into the meanline method separately, namely upstream and downstream of the rotor throat.

At the rotor throat (Station 9), static enthalpy was calculated by

$$h_9 = h_{9s} + Loss1 \tag{17}$$

Where

$$Loss1 = C_{p7}T_{07} \times \left(\frac{p_9}{p_{07}}\right)^{\frac{\gamma-1}{\gamma}} - C_{p9}T_9$$

While at the rotor trailing edge (Station 11), the stagnation enthalpy was calculated by

$$h_{011} = h_{011s} + Loss2 \tag{18}$$

Where

$$Loss2 = C_{p7}T_{07} \times \left(\frac{p_{011}}{p_{07}}\right)^{\frac{\gamma-1}{\gamma}} - C_{p11}T_{011}$$

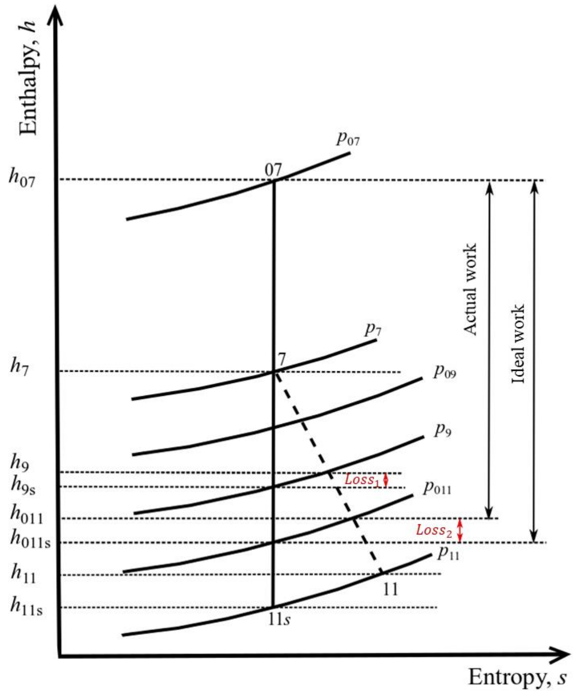


Figure 9 h-s diagram of the turbine

Blockage factor is usually included in conventional meanline modelling to correct the rotor throat area to account for boundary layer thickness and reduced localised meridional velocity due to secondary flows. In traditional meanline calculation methods, this factor has been introduced as a constant value, without considering the possibility of blockage varying at different operating conditions due to the inherently changing secondary flows and accumulated losses approaching the exducer throat. One of the features of this hybrid model is that the blockage factor was treated as a variable value that changed with operating condition.

In order to implement a variable blockage factor in the hybrid model, the first step involved derivation of a blockage factor calculation method. From aerodynamic theory, the effective area at the throat was calculated by

$$A_{eff} = \frac{\dot{m}\sqrt{RT_0}}{p_o\sqrt{\gamma}M}\left(1+\frac{\gamma-1}{2}M^2\right)^{\frac{\gamma+1}{2(\gamma-1)}} \tag{19}$$

Then the blockage factor can be estimated by

$$B = \left(1-\frac{A_{eff}}{A_{geom}}\right) \text{x}100\% \tag{20}$$

Where A_{geom} stands for the geometrical area.

Determining the three-dimensional geometric throat area for each rotor generated within the Design-Modeler software was a complex manual process that was not suitable for dealing with many geometries. To automate the post processing for more than 1000 CFD cases, a simplified 'throat area' calculation process was created in Ansys CFD-Post by defining the minimum area annulus turbo-surface along the streamwise direction as the throat plane, as depicted in Figure 10. In this case, the predominant flow direction was not perpendicular to this annular plane at the throat. Therefore, an angle between the meanline velocity and the plane was introduced in the effective area calculation, as shown in equation (19). A comparison was made between this simplified annulus plane method and the three-dimensional throat plane to illustrate the adequacy of the method. The three-dimensional throat plane was created in Design-Modeler and then imported to CFD-Post. The blockage factor calculated by these two methods is compared for several rotor geometries at the same rotating speed but different PR and one of the geometries is shown in Figure 11, which shows that the simplified method has sufficient accuracy in the respect of blockage factor calculation. Especially when the difference in evaluation / setup time and the consistency in application across different geometries is factored in, the simplified method is the clear choice for moving forward. Thus, the blockage factor was calculated by an automated process utilizing the simplified method in the CFD post-processing. It can also be seen from Figure 11 that the blockage factor varies appreciably according to the operating condition, validating the necessity of including variable blockage factors in a meanline calculation.

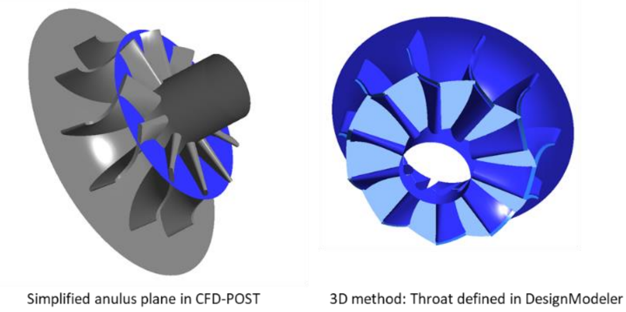


Figure 10 Simplified and 3D throat plane definition

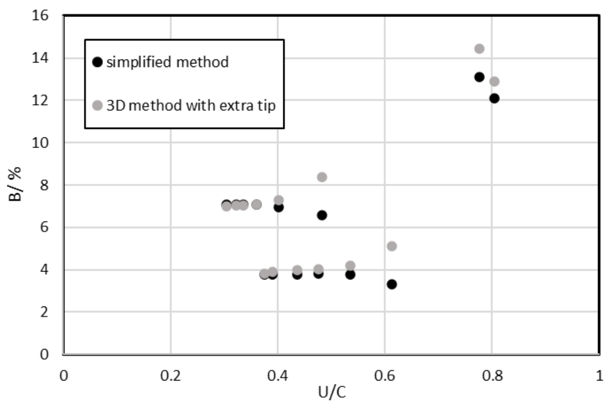


Figure 11 Comparison of blockage factor calculated by two methods

Data created through simulation of the parametric rotor model described in Section 2.2 in CFD was fed into an ANN model. The structure of the hybrid model is shown in Figure 12. There were 1400 groups of data in the database. Three ANNs were trained to predict the two portions of the loss and the blockage factor. The inputs to the ANN were rotor geometry and operating condition information, while the output of the ANNs were the rotor loss and the blockage factor respectively for a specific geometry and operating condition. Since only the rotor component was investigated in this paper, the performance relating to only the rotor component was extracted from the meanline model.

As far as calculation time is concerned, using a desktop computer workstation with 24 processors, it only required several minutes to train the ANN models. Once trained, the ANN model did not need to be retrained except if new cases were added to the training database to extend the range of geometries or operating conditions covered. The ANN model required only seconds to calculate and output the rotor loss and blockage factor for use in the meanline model. The meanline calculation took around 20 seconds to produce a turbine performance curve at one speed-line. Compared to the high-fidelity CFD modelling method, this hybrid-meanline model was around 700 times faster once it had been initially trained, illustrating tremendous superiority in terms of calculation time, thus fulfilling the role of design tool for full powertrain optimization.

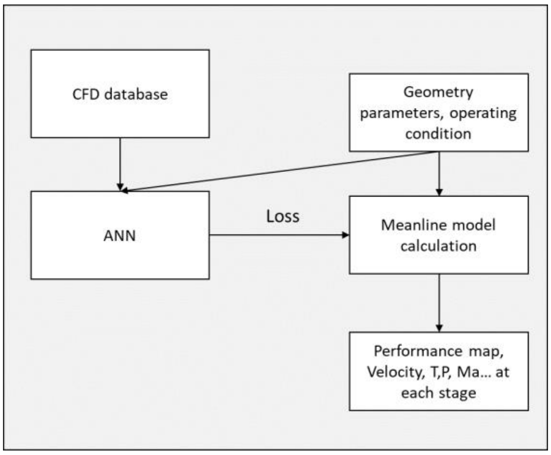


Figure 12 The structure of the hybrid meanline model

5. Results and Discussion

The hybrid meanline model was tested on four unseen designs, which were not included in the ANN training database. The geometric parameters and operating parameters of these unseen cases were within the design space. The hybrid model was evaluated by comparing the efficiency and mass flow rate predictions with the CFD simulations and the conventional meanline model with default loss coefficients, as illustrated in Figure 13.

Compared to the baseline case, Test case A was obtained by decreasing the blade angle distribution parameter Y_B, which resulted in the minimum value of this parameter. Test case B had a smaller blade angle distribution parameter Y_C compared to the baseline. Test case C was the case with a different meridional profile, characterized by a shorter axial length. Test case D was obtained by reducing the baseline blade number by 1.

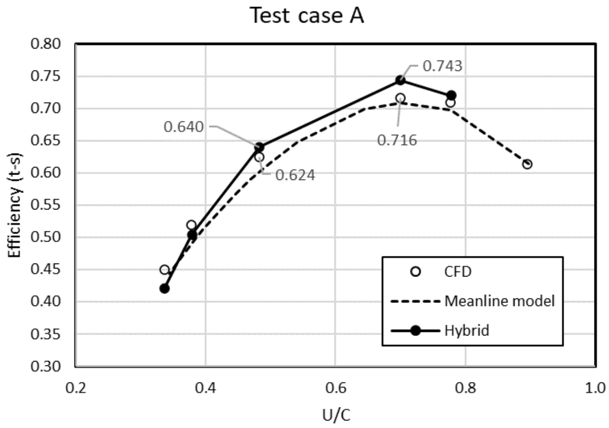
The efficiency predictions at U/C equals 0.48 and 0.70 of four test cases are labelled in Figure 13. It can be seen that at the off-design point, where U/C=0.48, the hybrid efficiency prediction errors for the four test cases are +1.6%pts, +2.4%pts, -1.7%pts, -0.8%pts respectively compared to the CFD results. The average error is 1.63%pts. When U/C=0.70, the efficiency prediction errors are +2.7%pts, +2.1%pts, -2.0%pts, -3.4%pts respectively, resulting in an average error of 2.55%pts. These results indicate that the hybrid model achieves greater accuracy at lower U/C operating conditions. The reason behind this is that enthalpy difference across the turbine at the design point is much smaller than that of off-design points, which makes the efficiency more sensitive to any error in loss prediction. Similar issues are observed at higher U/C operating points, where the enthalpy change is much less compared to the low U/C points. Therefore, it is understandable that the hybrid model appears not to work as well at high U/C operating conditions, since the same accuracy of loss prediction has a proportionally larger impact on efficiency than at operating points exhibiting a larger enthalpy drop across the turbine. Further enhancements to the accuracy of the ANN model could be helpful to overcome this problem, which will be one of the future works of this research.

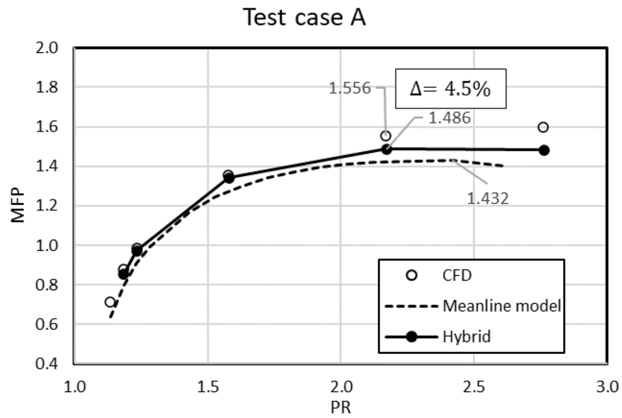
It can be seen from the efficiency map of Test case A and C that the hybrid model achieves better accuracy than the conventional meanline model, especially at U/C between 0.4 and 0.8. Considering that Test case A has a different streamwise blade angle distribution and that Test case C has a different axial length, it can be concluded that the additional aspects of rotor geometry included in the hybrid model (as depicted in Figure 6) allow the loss to be better captured than traditional meanline modelling methods.

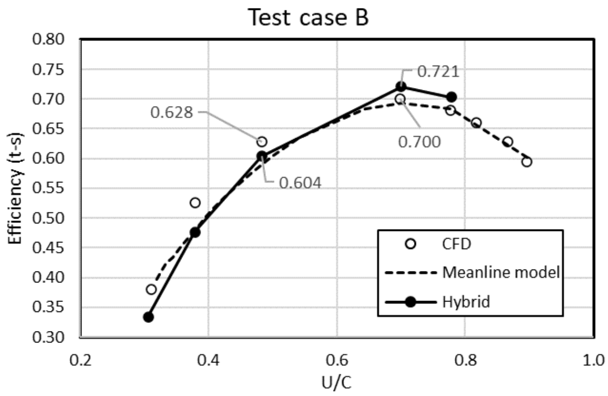
As for the MFP prediction, the errors are given by comparing the hybrid model predictions and the CFD results at choke condition, as is shown in Figure 13, which are 4.5%, 4.1%, 4.7%, 5.4% for Test case A, B, C, D respectively. The average error is 4.68%. Comparing to the CFD results, the conventional meanline model predicts the mass flow parameter with errors of -8.0%, -8.7%, -5.2%, -10.7% for test case A, B, C, D at choke conditions. The average error is 8.15%. Overall, the mass flow rate prediction achieved by the hybrid approach was significantly better than that of the conventional meanline method, which verifies the effectiveness of the variable blockage factor. By incorporating variable blockage and splitting the rotor loss into two components (before and after the throat), the hybrid model better captures the reality of the flow field and hence results in a more accurate mass flow rate prediction. In addition, in conventional loss models, the streamwise blade angle distribution was not included, in which rotors with different blade angles resulted in the same performance map. However, the hybrid model is capable of calculating the loss difference when blade angle varies between LE and TE in the streamwise direction and capturing the performance difference.

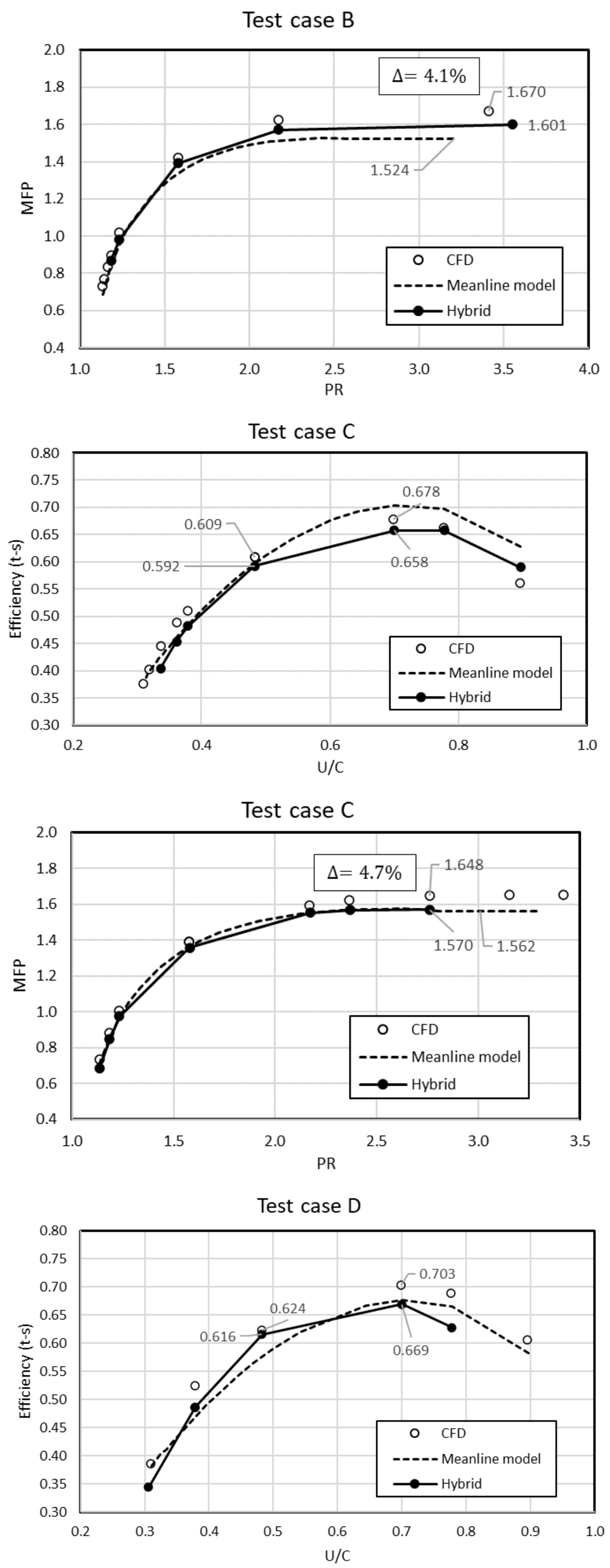
The results indicated that this hybrid meanline model could predict wide range performance maps, from the perspectives of geometry diversity and operating condition.

Significantly, it should be noted that the hybrid meanline model dispensed with the requirement for the loss calibration process compared to the conventional meanline method, which was vital for turbine design in the preliminary design stage and the overall optimization process in the 1-D engine modelling. The hybrid model also showed its advantage in greatly reduced computational time in comparison to CFD method, as described in Section 4.4.









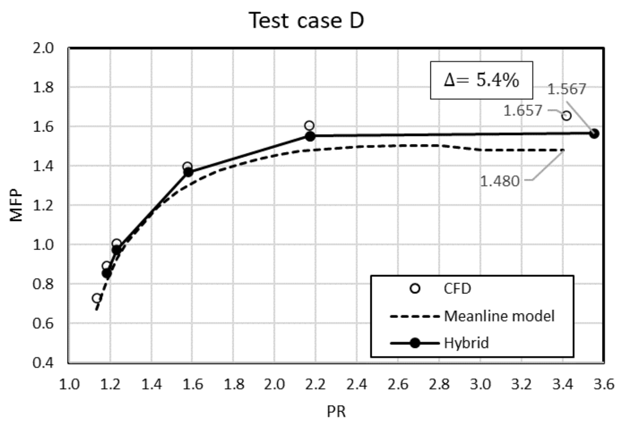


Figure 13 Validations of the hybrid model

6. Conclusions

This paper explored a new hybrid meanline modelling method for wide range performance prediction of radial turbine rotors with trained ANN using results from validated CFD on 75 geometries. The hybrid model was evaluated on four test cases spanning the design space. It indicated that this hybrid modelling method, independent of empirical factors and loss model calibration, has enhanced capability in wide range turbine MFP and efficiency predictions compared to conventional approach and was 700 times faster than CFD modelling, making it a viable design tool for turbocharger and full powertrain optimization. Compared to conventional meanline method, this hybrid model can capture the loss difference caused by different streamwise blade angle distribution and different axial length. When coupled with the inclusion of a variable throat blockage factor and splitting rotor loss into two components, marked improvements in prediction of turbine efficiency and MFP were witnessed. It can be expected that the extension of the method to include stators would result in an accurate and efficient low fidelity modelling method for turbine stages. Overall, this method is an effective reference for the turbomachinery research on utilizing machine learning technology.

Acknowledgement

The authors would like to thank ANSYS Inc. for the use of their CFD software and their technical support in this research. The use of the turbomachinery test facilities at Queens University Belfast in support of this project are gratefully acknowledged.

References

[1] Futral S.M., Wasserbauer C.A., 1965, “Off-design performance prediction with experimental verification for a radial-inflow turbine”, NASA TN D-2621.

[2] W. A Connor, D Flaxington, 1994, “A one-dimensional performance prediction method for radial inflow turbines’’, Instn. Mech. Engrs., Conference Publications, Turbocharging, Vol. 6:

271-282.

[3] https://www.gtisoft.com/gt-suite-applications/propulsion-systems/gt-power-engine-simulation-software/ [Accessed 20/05/2021]

[4] Pesiridis, A., Salim, S.I.W. and Martinez-Botas, R.F., 2012, May. Turbocharger matching methodology for improved exhaust energy recovery. In Proc. of the 10th Int. Conf. on Turbocharging and Turbochargers (pp. 203-218).

[5] Serrano, J.R., Arnau, F.J., García-Cuevas, L.M., Dombrovsky, A. and Tartoussi, H., 2016, "Development and validation of a radial turbine efficiency and mass flow model at design and off-design conditions", Energy Conversion and Management, Vol. 128, pp. 281-293.

[6] Qi, J., et al., 2016, "Supercritical CO2 radial turbine design performance as a function of turbine size parameters", In ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition. American Society of Mechanical Engineers Digital Collection.

[7] Gao Y, Petrie-Repar P, 2018, "Validation of meanline performance prediction method for radial and mixed flow turbine",In 13th IMECHE conference transactions.

[8] Meitner, P. L., Glassman, A. J., 1980, "Loss model for off-design performance analysis of radial turbines with pivoting-vane, variable area stators," SAE Paper No. 801135.

[9] Qiu, X., Anderson, M.R. and Baines, N.C., 2009, "Meanline modeling of radial inflow turbine with variable area nozzle", In ASME Turbo Expo 2009: Power for Land, Sea, and Air (pp. 1185-1191). American Society of Mechanical Engineers.

[10] Bin Mamat A.I., Martinez-Botas R.F., 2010, "Mean Line Flow Model of Steady and Pulsating Flow of a Mixed-Flow Turbine Turbocharger", ASME. Turbo Expo: Power for Land, Sea, and Air, Volume 7: Turbomachinery, Parts A, B, and C. (44021): p. 2393-2404.

[11] Yang, B., Martinez-Botas, R., 2019. "TURBODYNA: Centrifugal/Centripetal Turbomachinery Dynamic Simulator and Its Application on a Mixed Flow Turbine". Journal of Engineering for Gas Turbines and Power, 141(10), 101012.

[12] M.S. Chiong, S. Rajoo, R.F. Martinez-Botas, A.W. Costall, 2012, "Engine turbocharger performance prediction: One-dimensional modeling of a twin entry turbine", Energy Convers. Manag. 57 (2012) 68–78.

[13] Todd, C.A. and Futral, S.M., 1969. A Fortran IV program to estimate the off-design performance of radial-inflow turbines. NASA TN D-5059.

[14] Gibson L., Galloway L., Kim S., and Spence S., 2017, Assessment of turbulence model predictions for a centrifugal compressor simulation. Journal of the Global Power and Propulsion Society. 1: 142–156, https://doi.org/10.22261/2II890

[15] Baines, N. C., 1998. "A Meanline Prediction Method for Radial Turbine Efficiency in Axial and Radial Turbines". In IMechE 6th International Conference on Turbocharging and Air Management Systems, London, November, pp. 3–5.

[16] Glassman, A.J., 1995. Enhanced Analysis and users manual for radial-inflow turbine conceptual design code RTD.

[17] Li, Z. and Zheng, X., 2017. Review of design optimization methods for turbomachinery aerodynamics. Progress in Aerospace Sciences, 93, pp.1-23.

[18] Qin, R., Ju, Y., Spence, S., and Zhang, C., 2021, "Metamodel-driven data mining model for centrifugal compressor three-dimensional design", ASME J. Turbomach. December 2021, 143(12): 121013.

[19] Liu, Z. and Karimi, I.A., 2020. Gas turbine performance prediction via machine learning. Energy, 192, p.116627.

[20] Karsoliya, S., 2012. Approximating number of hidden layer neurons in multiple hidden layer BPNN architecture. International Journal of Engineering Trends and Technology, 3(6), pp.714-717.

[21] Ju, Y., Qin R., Kipouros T., Parks G., Zhang C., 2016, "A high-dimensional design optimisation method for centrifugal impellers", Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 230(3), pp. 272–288.