DEVELOPING GENERIC DESIGN EXPERTISE FOR GAS TURBINE ENGINES

Robust Design of a Micro Centrifugal Compressor

Proefschrift

ter verkrijging van de graad van doctor aan de Technische Universiteit Delft, op gezag van de Rector Magnificus Prof. Ir. K.C.A.M. Luyben, voorzitter van het College voor Promoties, in het openbaar te verdedigen op 30 juni 2014 om 12.30 uur

door

Adeel Javed

Master of Science in Thermal Power
Cranfield University, UK

geboren te Rawalpindi, Pakistan
Dit proefschrift is goedgekeurd door de promotor:

Prof. Ir. J. P. van Buijtenen

en de copromotor

Dr. R. Pecnik

Samenstelling promotiecommissie:

Rector Magnificus voorzitter
Prof. Ir. J. P. van Buijtenen Technische Universiteit Delft, promotor
Dr. R. Pecnik Technische Universiteit Delft, copromotor
Prof. Dr. Ir. B.-J. Boersma Technische Universiteit Delft
Prof. Dr. P. Colonna Technische Universiteit Delft
Prof. Dr. P. Pilidis Cranfield University
Dr. Ir. N. P. Kruyt Universiteit Twente
Ir. W. P. J. Visser Micro Turbine Technology BV

This research has been financially supported by AgentschapNL/NIVR (Project SRP 59624), Micro Turbine Technology BV, and Sulzer Eldim BV.

Copyright © 2014 by Adeel Javed

All rights reserved. No part of the material protected by this copyright notice may be reproduced or utilized in any form or by any means, electronic or mechanical, including photocopying, recording or by any information storage and retrieval system, without the prior permission of the author.


Printed by Ipskamp Drukkers, The Netherlands.

---

1Author’s email: adeel.ja@gmail.com
In the name of God, the Most Gracious, the Most Merciful.

Read! In the Name of your Lord, Who has created (all that exists), He has created man from a clot of blood. Read! And your Lord is the Most Generous, Who has taught (the writing) by the pen. He has taught man that which he knew not.

Al-Alaq, Verses 1-5, Holy Qur’an
To the memory of my dearest mother.

Tanzila Naheed (1958 – 2013)

To my father, my wife and my family.
Summary

The design of complex mechanical parts, especially for gas turbines and turbochargers, encompass a number of disciplines. These disciplines can be distinguished as functional design (aero-thermodynamics), mechanical design (structural integrity and rotordynamics) and design for manufacturing. Unfortunately, the various functions in these disciplines are carried out by different people, in different departments of a company, or even in different companies.

Traditionally, the attitude of the designers has been, “we design it, you build it”. This notion has been termed as “over the wall” approach, where the design engineers are sitting at one side of the wall and throwing designs over to the other side to the manufacturing engineers. The manufacturing engineers then have to deal with various manufacturing problems, for instance, the design not being manufacturable or too strictly tolerated for controlling the inherent manufacturing uncertainties and the resulting dimensional deviations. This is mainly a consequence of lack of manufacturing knowledge in the design office, while manufacturing engineers are not being involved in the design effort. Manufacturing costs can substantially increase if a complex design carrying stringent tolerance margins has to be manufactured. Therefore, following the “over the wall” approach may result in a design that will require a reassessment for its manufacturability and production costs.

Recent developments in concurrent engineering have led to a greater cooperation between the traditionally separate design and manufacturing disciplines. Concurrent engineering is being adapted in industry, which represents a philosophy for product design that relies on simultaneous evaluation of a design by the design and manufacturing engineers in order to achieve a high quality product having low production costs.

The goal of this thesis is to support concurrent engineering in the gas turbine industry. A robust design methodology has been presented, which considers both the functionality and manufacturability of a product during the preliminary design phase, thus negating the “over the wall” approach. In simple terms, a robust design methodology leads to a robust product for which the output performance of the product is insensitive to a limited variability of the input sources, without having to minimize the sources of variation. Applying the robust design methodology to optimize a design would, therefore, considerably improve the product quality in terms of functionality and costs of manufacturing.
Robust engineering requires uncertainty quantification (UQ) in a system. Manufacturing uncertainties are stochastic in nature and cause random dimensional deviations in the manufactured parts. These deviations eventually propagate into the functionality of the parts as variability in their performance. Therefore, quantification of manufacturing uncertainties for their influence on part performance is fundamental.

UQ begins with the data assimilation process, in which the uncertainties of the input quantities are characterized as explicit probability distributions. Monte Carlo simulation (MCS) is a widely used method to model uncertainty propagation in a system. It is a comprehensive probabilistic sampling technique for simulating a randomly occurring process, given the stochastic properties of one or more input variables, with a focus on characterizing the statistical nature of the output response. Monte Carlo samples are evaluated deterministically in a model, which emulates the behavior of a system and determines its output performance, subjected to a set of inputs. The model is required to be computationally inexpensive in order to simulate a large number of Monte Carlo samples for UQ. Moreover, the model also has to accurately emulate the system’s behavior in a sufficient detail. Nonetheless, computational cost and detail level of a model are two conflicting aspects, which are impossible to satisfy simultaneously. Finally after UQ, the robust design optimization is carried out by coupling the model based Monte Carlo simulator to an optimizer. The optimizer locates the robust designs in a predefined search space for which the output performance is immune to a limited variability of the inputs.

A small-scale turbocharger compressor has been used as a test case in the present study. The turbocharger is part of a microturbine under development by Micro Turbine Technology BV (MTT). Main focus has been given to the impeller since it is a complex design, and also responsible for the transfer of work to the compressor flow. Initially, a one-dimensional (1D) meanline model was prepared using the two-zone methodology with two main objectives:

- To model the compressor performance with a reasonable accuracy and detail using very basic geometric and operational inputs.
- The modeling process is sufficiently quick to perform a probabilistic evaluation of manufacturing uncertainties and robust design optimization.

The compressor was simulated in the 1D meanline model and results were compared against the available test data for validation. For a more detailed understanding of the compressor performance, a three-dimensional (3D) performance evaluation was also carried out using computational fluid dynamics (CFD). The characteristic two-zone flow was modeled and its behavior was quantified at different operating conditions. The CFD outcomes were also used to improve the empirical modules in the 1D meanline model.

Despite the ability to swiftly compute compressor performance, the 1D meanline model lacked the required accuracy to predict the variation in performance
with deviations in impeller geometry. Therefore, the model was discarded for UQ application. A metamodel (also known as surrogate) was considered instead. A metamodel can interpolate the outputs from pre-evaluated high fidelity data very accurately and efficiently. Metamodels are, therefore, widely used to perform UQ and robust design optimization. However, preparation of a suitable metamodel is a difficult and time consuming task.

A metamodel was constructed for the impeller using design of experiments (DoE) samples and used to perform MCS. A large number of Monte Carlo samples have been used in the evaluation to determine the mean and standard deviation of compressor performance, along with the probability distribution. For the given tolerance range, a significant number of impellers manufactured under uncertainty could be functionally unacceptable as they are found to be located well away from the mean performance.

In order to reduce the variability in compressor performance, the robust design optimization has been performed using the multi-objective genetic algorithm (MOGA) coupled with the metamodel based Monte Carlo simulator. Consequently, three robust impeller designs were obtained and evaluated for their advantages and disadvantages compared to the baseline impeller design.

Adding more value to the research, an experimental UQ was performed at Mitsubishi Turbocharger and Engine Europe BV (MTEE). To construct an experimental metamodel, a sample of impellers were manufactured specifically for DoE. The DoE samples were tested on the turbocharger test bench at MTEE and the output responses were used to fit a polynomial response surface. MCS was performed using the experimental metamodel and the variability in performance was determined. The results obtained from computational and experimental UQ showed a good agreement, thereby confirming the methodology and tools applied in the research.

All in all, the robust design methodology, and the resulting robust impellers have symbolized the advantages of concurrent engineering, where any product could be designed considering its functionality and manufacturability simultaneously. It is very likely that the manufacturing costs are reduced as UQ and robust design methodology allow the identification and removal of unnecessarily strict tolerance margins, tolerance relaxation for difficult dimensions and reduction in part rejections.
Samenvatting

Het ontwerp van complexe mechanische onderdelen, met name voor gasturbinen en turboladers, omvat een aantal disciplines. Deze disciplines kunnen worden onderverdeeld in functioneel ontwerp (aero-thermodynamica), mechanisch ontwerp (structurele integriteit en rotordynamica) en ontwerp voor produceerbaarheid. Helaas worden de verschillende functies binnen deze disciplines door verschillende mensen uitgevoerd op verschillende afdelingen van een onderneming of zelfs door verschillende bedrijven.

De traditionele houding van ontwerpers was veelal “wij ontwerpen het, u bouwt het”. Deze houding wordt dikwijls aangeduid als de “over de muur” aanpak waarbij de ontwerpers aan n zijde van de muur zitten en hun ontwerp over de muur gooien naar de producenten aan de andere zijde. Daarbij krijgen deze producenten vaak te maken met een variatie aan productieproblemen zoals een ontwerp dat niet produceerbaar is of een te strikte toepassing van toleranties kent. Veelal is dit het gevolg van het ontbreken van fabricagekennis bij de ontwerpers enerzijds en het niet betrekken van de producenten bij het ontwerpprocès anderzijds. Fabricagekosten kunnen substantieel toenemen wanneer een complex ontwerp volgens te strikte tolerantiemarges geproduceerd dient te worden. Het gevolg van de “over de muur” aanpak resulteert dus mogelijk in een herziening van het ontwerp wat betreft de produceerbaarheid en fabricagekosten.

Recente ontwikkelingen in parallel ontwerp (ook wel ‘concurrent engineering’ genaamd) hebben geleid tot een verbeterde samenwerking tussen de traditioneel gescheiden ontwerp- en productiedisciplines. Parallel ontwerp wordt geadopteerd door de industrie als een filosofie waarbij productontwerp gebaseerd wordt op een simultane ontwerpevaluatie door ontwerpers en producenten met als doel een hoge kwaliteit te realiseren bij lage fabricagekosten.

Het doel van deze thesis is het ondersteunen van concurrent engineering in de gasturbinen-industrie. Een *robuuste ontwerpmethode* is gepresenteerd welke beide functionaliteit en produceerbaarheid in ogenschouw neemt in een vroeg ontwerpstadium. De “over de muur” aanpak is dus genegeerd. Simpel gezegd, een robuuste ontwerpmethode leidt tot een robuust product waarbij de productprestaties ongevoelig zijn voor variantie van een beperkt aantal ingangsvariabelen zonder de noodzaak deze variantie te minimaliseren. Toepassing van een robuuste ontwerpmethode ter optimalisatie van een ontwerp zou om deze reden een tot significante kwaliteitsverbetering moeten leiden wat betreft functionaliteit en fabricagekosten.
Robuuste productontwikkeling vereist een onzekerheidskwantificatie (ook wel ‘uncertainty quantification’ of UQ) van een system. Fabricageonzekerheden zijn van nature stochastisch en veroorzaken willekeurige dimensionele afwijkingen in vervaardigde onderdelen. Deze afwijkingen werken uiteindelijk door in de functionaliteit van onderdelen, zich uiteindelijk als een variantie in prestaties. Om deze reden is een kwantificatie op productieonzekerheden en de invloed daarvan op prestaties van fundamenteel belang.

UQ begint met dataverwerking waarbij de onzekerheden van ingangsvariabelen worden gekarakteriseerd als kansverdelingen. Monte Carlo-simulatie (MCS) is een wijds toegepaste methodiek om spreiding van onzekerheden in systemen te modelleren. Het is een veelomvattende techniek voor het bemonsteren van waarschijnlijkheden ten behoeve van het simuleren van willekeurige processen. Gegeven zijn hierbij de stochastische eigenschappen van n of meer ingangsvariabelen met de focus op het karakteriseren van de statistische aard van uitgangsvariabelen. Monte Carlo-monsters worden deterministisch gevalueerd in een model dat systeemgedrag emuleert en daarbij worden de uitgangsprestaties bepaald op basis van een set ingangsvariabelen. Van dit model wordt vereist dat het niet rekenintensief is vanwege het grote aantal Monte Carlo-monsters dat gesimuleerd dient te worden ten behoeve van UQ. Tevens moet het model in staat zijn accuraat en met voldoende detail het systeemgedrag te emuleren. Echter, rekenintensiviteit en detailering zijn tegenstrijdige modelaspecten welke niet simultaan gerespecteerd kunnen worden. Tenslotte wordt na de UQ een robuuste ontwerpopimalisatie uitgevoerd door de modelgebaseerde Monte Carlo-simulatie te koppelen aan een optimalisatie-algoritme. Het optimalisatie-algoritme lokaliseert robuuste ontwerpen in een voorgedefinieerde ontwerpruimte waarvoor geldt dat de uiteindelijke uitgangsprestaties immuun zijn voor een begrensde variantie in ingangsvariabelen.

Een compressor van een kleine turbolader is als een casus voor het huidige onderzoek gebruikt. De turbolader is een onderdeel in de microturbine die Micro Turbine Technology BV (MTT) in ontwikkeling heeft. Daarbij is de nadruk gelegd op de impeller aangezien het ontwerp complex is en dit onderdeel verantwoordelijk is voor de overdracht van arbeid aan de compressorstroom. Aanvankelijk is een n-dimensionaal (1D) meanline model voorbereid op basis van de twee-zone-methodiek met twee hoofddoelen:

- Het met voldoende nauwkeurigheid en detail modelleren van de compressorprestaties gebruikmakend van versimpelde geometrie en gebruiksccondities.
- Het modelleerproces is voldoende snel om een waarschijnlijkheidsstudie van fabricage-onzekerheden en een robuuste ontwerpopimalisatie uit te kunnen voeren.

De compressor is gesimuleerd met het 1D meanline model en de verkregen resultaten zijn vergeleken met de beschikbare testdata ter validatie. Met behulp van computational fluid dynamics (CFD) is een drie-dimensionaal (3D) onderzoek gedaan om de compressorprestaties beter in detail te kunnen begrijpen. De
karakteristieke twee-zone stroming is gemodelleerd en het gedrag is gekwantificeerd voor verschillende bedrijfsoorstantigheden. De CFD-resultaten zijn ook gebruikt om de empirische modules in het 1D meanline model te verbeteren. Ondanks de mogelijkheid om snel compressorprestaties te berekenen, miste het 1D meanline model de noodzakelijke verfijning. Rekeninghoudend met afwijkingen in impeller-geometrie, is de variantie in prestaties niet te voorspellen. Het model werd daarom niet geschikt geacht voor UQ-toepassing. In plaats hiervan is een metamodel (ook wel surrogaat genoemd) gebruikt. Een metamodel kan de uitgangsvariabelen snel en efficiënt interpoleren vanuit een vooraf gevalueerde natuurgetrouwe dataset. Metamodellen worden om deze reden veel gebruikt voor UQ en het uitvoeren van robuuste ontwerp-optimalisatie. Echter, de voorbereiding van een geschikt metamodel is een tijdrovende en lastige taak.

Een metamodel voor de impeller is samengesteld door gebruik te maken van experimentontwerp (ook wel design of experiments of DoE). Dit metamodel is gebruikt om MCS uit te voeren. Een groot aantal Monte Carlo-monsters zijn gebruikt in het onderzoek om de verwachtingswaarde en de standaarddeviatie van compressorprestaties te kunnen bepalen tezamen met de kansverdeling. Voor het gegeven tolerantiebereik is er een groot aantal met onzekerheid gemaakte impellers die functioneel onacceptabel zouden kunnen zijn. Dit omdat ze ver buiten het bereik van de gemiddelde prestaties liggen.

Robuuste ontwerp-optimalisatie is benut, gebruik makend van het multi-objectief genetisch algoritme (MOGA) gekoppeld aan het op Monte Carlo-simulator metamodel, met als doel de variantie in compressorprestaties te reduceren. Zo-doende zijn er drie robuuste impellers verkregen en beoordeeld op hun voor- en nadelen vergeleken met het basis impeller-ontwerp. Om meer waarde aan het onderzoek te geven is een experimenteel UQ uitgevoerd bij Mitsubishi Turbocharger and Engine Europe BV (MTEE). Een specifieke set impellers (DoE-monsters) zijn puur gefabriceerd voor het DoE onderzoek om een experimenteel metamodel te kunnen construeren. De DoE-monsters zijn getest op de turboladertestbank bij MTEE waarbij de uitgangsrespons passend is gemaakt op een polynoom responsopervlak (ook wel response surface). MCS is uitgevoerd met behulp van het experimentele metamodel en de variantie in prestaties is vastgesteld. De verkregen resultaten van het berekende en experimentele UQ laten een goede overeenkomst zien en bevestigen daarmee de methodiek en de in het onderzoek benutte gereedschappen.

Samengevat, de robuuste ontwerpmethodiek en de resulterende robuuste impellers hebben het voordeel van parallelle ontwerpen aangetoond. Hiermee kan elk product simultaan op functionaliteit en maakbaarheid geoptimaliseerd worden. Het is hoogstwaarschijnlijk dat de fabricagekosten zijn verlaagd omdat UQ, tezamen met de robuuste ontwerpmethodiek, de onnodig strikte toleranties identificeert en verwijdert. Tevens wordt het tolerantiebereik verruimd op lastige dimensies en is er een reductie van afgekeurde onderdelen.
## 3 1D Meanline Performance Evaluation

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1 Preface</td>
<td>35</td>
</tr>
<tr>
<td>3.2 Introduction to Centrifugal Compressors</td>
<td>35</td>
</tr>
<tr>
<td>3.2.1 Fundamental Design and Operation</td>
<td>35</td>
</tr>
<tr>
<td>3.3 Impeller Performance Characteristics</td>
<td>41</td>
</tr>
<tr>
<td>3.3.1 Velocity Triangles and Energy Transfer</td>
<td>41</td>
</tr>
<tr>
<td>3.3.2 Rothalpy</td>
<td>42</td>
</tr>
<tr>
<td>3.3.3 Slip Factor</td>
<td>43</td>
</tr>
<tr>
<td>3.3.4 Impeller Flow Physics</td>
<td>43</td>
</tr>
<tr>
<td>3.4 Turbomachinery Design and Development</td>
<td>46</td>
</tr>
<tr>
<td>3.4.1 Levels of 1D Design and Analysis</td>
<td>48</td>
</tr>
<tr>
<td>3.4.2 Selection of an Appropriate Model</td>
<td>49</td>
</tr>
<tr>
<td>3.5 1D Meanline Model Development</td>
<td>50</td>
</tr>
<tr>
<td>3.5.1 Inducer Analysis</td>
<td>50</td>
</tr>
<tr>
<td>3.5.2 Impeller Exducer/Tip-State Modeling</td>
<td>52</td>
</tr>
<tr>
<td>3.5.3 Empirical Loss Models</td>
<td>63</td>
</tr>
<tr>
<td>3.5.4 Vaneless Diffuser Modeling</td>
<td>67</td>
</tr>
<tr>
<td>3.5.5 Discharge Volute Modeling</td>
<td>68</td>
</tr>
<tr>
<td>3.6 Compressor Performance Evaluation in 1D</td>
<td>69</td>
</tr>
<tr>
<td>3.6.1 Compressor Geometry and Specifications</td>
<td>69</td>
</tr>
<tr>
<td>3.6.2 Stage Performance Mapping and Validation</td>
<td>71</td>
</tr>
<tr>
<td>3.6.3 Study of the Compressor Loss Mechanisms</td>
<td>72</td>
</tr>
<tr>
<td>3.7 Additional 1D Meanline Modeling</td>
<td>77</td>
</tr>
<tr>
<td>3.8 Conclusions and Recommendations</td>
<td>77</td>
</tr>
</tbody>
</table>

## 4 Performance Evaluation in 3D CFD

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1 Preface</td>
<td>83</td>
</tr>
<tr>
<td>4.2 Introduction to CFD</td>
<td>83</td>
</tr>
<tr>
<td>4.3 CFD Preprocessing and Solution setup</td>
<td>84</td>
</tr>
<tr>
<td>4.3.1 Geometry Definition</td>
<td>84</td>
</tr>
<tr>
<td>4.3.2 Grid Processing</td>
<td>86</td>
</tr>
<tr>
<td>4.3.3 Boundary Conditions</td>
<td>87</td>
</tr>
<tr>
<td>4.3.4 Turbulence Model Selection</td>
<td>88</td>
</tr>
<tr>
<td>4.4 Performance Evaluation and Validation</td>
<td>89</td>
</tr>
<tr>
<td>4.5 Two-Zone Flow Investigation</td>
<td>90</td>
</tr>
<tr>
<td>4.5.1 Two-Zone Flow Quantification</td>
<td>92</td>
</tr>
<tr>
<td>4.6 Improvement of the 1D Meanline Model</td>
<td>97</td>
</tr>
<tr>
<td>4.6.1 Design Space Definition</td>
<td>97</td>
</tr>
<tr>
<td>4.6.2 Impeller Parameterization and CFD Evaluation</td>
<td>98</td>
</tr>
<tr>
<td>4.6.3 Calibration of the 1D Meanline Model</td>
<td>99</td>
</tr>
<tr>
<td>4.6.4 Validation of the 1D Meanline Model</td>
<td>101</td>
</tr>
<tr>
<td>4.7 Conclusions and Recommendations</td>
<td>103</td>
</tr>
</tbody>
</table>
List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Example of a typical “over the wall” approach in the industry</td>
<td>2</td>
</tr>
<tr>
<td>1.2</td>
<td>Schematic of a typical turbofan engine [courtesy CFM]</td>
<td>3</td>
</tr>
<tr>
<td>1.3</td>
<td>Case studies from different Dutch part manufacturers</td>
<td>6</td>
</tr>
<tr>
<td>2.1</td>
<td>A typical turbocharger and its main components</td>
<td>13</td>
</tr>
<tr>
<td>2.2</td>
<td>MTT microturbine for CHP application</td>
<td>15</td>
</tr>
<tr>
<td>2.3</td>
<td>A typical normal probability distribution for presenting random geometric deviations in manufactured parts (the terms LSL and USL represent the lower and upper specification limits, respectively)</td>
<td>19</td>
</tr>
<tr>
<td>2.4</td>
<td>Output variability obtained from sensitive and robust designs</td>
<td>22</td>
</tr>
<tr>
<td>2.5</td>
<td>Generic three-step robust design methodology</td>
<td>23</td>
</tr>
<tr>
<td>2.6</td>
<td>Illustration of uncertainty propagation</td>
<td>24</td>
</tr>
<tr>
<td>2.7</td>
<td>Impeller die casting process [courtesy Furukawa-Sky Aluminum Corp]</td>
<td>26</td>
</tr>
<tr>
<td>2.8</td>
<td>Illustration of different machining operations on the impeller</td>
<td>27</td>
</tr>
<tr>
<td>3.1</td>
<td>Main components of a turbocharger centrifugal compressor</td>
<td>36</td>
</tr>
<tr>
<td>3.2</td>
<td>Modern turbocharger compressor impellers</td>
<td>37</td>
</tr>
<tr>
<td>3.3</td>
<td>Types of diffusers</td>
<td>39</td>
</tr>
<tr>
<td>3.4</td>
<td>Volute configurations</td>
<td>40</td>
</tr>
<tr>
<td>3.5</td>
<td>Velocity triangles at impeller (a) inducer and (b) exducer sections</td>
<td>41</td>
</tr>
<tr>
<td>3.6</td>
<td>Flow structure in a centrifugal compressor impeller [1]</td>
<td>44</td>
</tr>
<tr>
<td>3.7</td>
<td>Velocity measurements at various sections of an Eckardt compressor impeller [10]</td>
<td>45</td>
</tr>
<tr>
<td>3.8</td>
<td>Turbomachinery design process and flow of information</td>
<td>47</td>
</tr>
<tr>
<td>3.9</td>
<td>Compressor stage with station numbers</td>
<td>51</td>
</tr>
<tr>
<td>3.10</td>
<td>Conceptual representation of the TEIS model [2]</td>
<td>53</td>
</tr>
<tr>
<td>3.11</td>
<td>Secondary zone mass flux fraction $\chi$ vs. area fraction $\epsilon$ [2]</td>
<td>57</td>
</tr>
<tr>
<td>3.12</td>
<td>Illustration of the impeller mixed-out state</td>
<td>61</td>
</tr>
<tr>
<td>3.13</td>
<td>The test case turbocharger compressor</td>
<td>70</td>
</tr>
<tr>
<td>3.14</td>
<td>1D compressor performance plots and comparison with experimental data at 220,000rpm rotational speed</td>
<td>71</td>
</tr>
<tr>
<td>3.15</td>
<td>1D internal loss plots at different rotational speeds</td>
<td>73</td>
</tr>
<tr>
<td>3.16</td>
<td>1D external loss plots at different rotational speeds</td>
<td>74</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
<td>------</td>
</tr>
<tr>
<td>3.17</td>
<td>1D stationary component loss plots at different rotational speeds</td>
<td>75</td>
</tr>
<tr>
<td>3.18</td>
<td>Compressor loss grading at the best-efficiency operating point</td>
<td>76</td>
</tr>
<tr>
<td>4.1</td>
<td>Fluid domain and boundary conditions</td>
<td>85</td>
</tr>
<tr>
<td>4.2</td>
<td>Structured mesh of the fluid domain</td>
<td>86</td>
</tr>
<tr>
<td>4.3</td>
<td>Compressor map comparison for 220,000rpm speed line</td>
<td>89</td>
</tr>
<tr>
<td>4.4</td>
<td>Relative velocity streamlines representing the secondary flow vortex structure at different operating points for 220,000rpm speed line</td>
<td>91</td>
</tr>
<tr>
<td>4.5</td>
<td>Mass flow distribution at impeller trailing edge for stall, best-efficiency and choke operating conditions at 220,000rpm rotational speed</td>
<td>95</td>
</tr>
<tr>
<td>4.6</td>
<td>Secondary flow illustration at impeller trailing edge for different operating conditions at 220,000rpm rotational speed (the terms FB and SB represent the full and splitter blades, respectively)</td>
<td>96</td>
</tr>
<tr>
<td>4.7</td>
<td>$\chi$ vs. $\epsilon$ correlation for the turbocharger compressor obtained from CFD in comparison to reference centrifugal compressor data given in ref. [5]</td>
<td>97</td>
</tr>
<tr>
<td>4.8</td>
<td>Impeller parameterization</td>
<td>99</td>
</tr>
<tr>
<td>4.9</td>
<td>Performance evaluation of impeller geometric deviations using the 1D meanline model and comparison with the CFD data</td>
<td>102</td>
</tr>
<tr>
<td>5.1</td>
<td>Metamodeling scheme applied for the test case impeller</td>
<td>108</td>
</tr>
<tr>
<td>5.2</td>
<td>DoE response scatter along mean value for test case impeller</td>
<td>111</td>
</tr>
<tr>
<td>5.3</td>
<td>Performance evaluation of impeller geometric deviations using the response surface model and comparison with the CFD data</td>
<td>117</td>
</tr>
<tr>
<td>5.4</td>
<td>Performance evaluation of impeller geometric deviations using the kriging model and comparison with the CFD data</td>
<td>119</td>
</tr>
<tr>
<td>5.5</td>
<td>Sensitivity ranking of impeller performance due to geometric deviations predicted by the response surface models</td>
<td>121</td>
</tr>
<tr>
<td>5.6</td>
<td>Probability distributions of variation in impeller pressure ratio and isentropic efficiency predicted by the metamodel based MCS</td>
<td>124</td>
</tr>
<tr>
<td>5.7</td>
<td>Scatter plots from Monte Carlo simulation showing the effects of geometric variation on impeller performance</td>
<td>125</td>
</tr>
<tr>
<td>5.8</td>
<td>Uncertainty ranking of the impeller geometric parameters decomposed by ANOVA</td>
<td>126</td>
</tr>
<tr>
<td>6.1</td>
<td>An example of the Pareto front in case of a multi-objective optimization. Design $D_1$ is dominated by designs $D_2$ and $D_3$, while $D_2$ and $D_3$ are non-dominating for each other; hence they are the Pareto optimal solutions</td>
<td>131</td>
</tr>
<tr>
<td>6.2</td>
<td>Local and global minima [1]</td>
<td>132</td>
</tr>
<tr>
<td>6.3</td>
<td>Illustration of the tournament selection method with $s = 2$ [9]</td>
<td>134</td>
</tr>
<tr>
<td>6.4</td>
<td>The basic cycle of genetic algorithms [1]</td>
<td>135</td>
</tr>
<tr>
<td>6.5</td>
<td>Flowchart for robust design optimization methodology</td>
<td>136</td>
</tr>
</tbody>
</table>
LIST OF FIGURES

6.6 Pareto optimal solutions obtained from MOGA optimizations for short and extended search spaces ........................................ 138
6.7 Schematic comparison between the trims of the baseline impeller and the optimized robust impeller designs ......................... 139
6.8 Probability distributions of variation in performance of robust impeller designs and comparison with the baseline (denoted by ‘Bsl’ in the plots) .................................................. 141

7.1 Manufacturing of the DoE impellers for experimentation ............ 147
7.2 Illustration of impeller geometry as machine-readable parameters . 148
7.3 MTEE turbocharger test bench ............................................. 149
7.4 Comparison between the measured DoE responses and the experimental response surface model outcomes for the test case compressor (For confidentiality, $t_{clr,1}$ and $t_{clr,2}$ are represented as -1, 0 and +1 levels of factorial design) .............................................. 151
7.5 Performance sensitivity evaluation and comparison between the CFD, computational metamodel and the experimental metamodel results (the term ‘RSM’ represents a response surface model) ............. 153
7.6 Performance variation under manufacturing uncertainties predicted by metamodel assisted MCS ........................................... 155
## List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>Specifications of the turbocharger Compressor</td>
<td>70</td>
</tr>
<tr>
<td>4.1</td>
<td>Selected impeller geometric parameters and their deviations</td>
<td>98</td>
</tr>
<tr>
<td>5.1</td>
<td>Response surface model diagnostics</td>
<td>116</td>
</tr>
<tr>
<td>5.2</td>
<td>Sensitive impeller geometric parameters and their tolerances</td>
<td>122</td>
</tr>
<tr>
<td>6.1</td>
<td>Geometric and performance comparison between the baseline and robust impeller designs obtained from metamodel based MOGA optimization</td>
<td>140</td>
</tr>
<tr>
<td>6.2</td>
<td>Validation of the optimized impellers in CFD</td>
<td>142</td>
</tr>
<tr>
<td>7.1</td>
<td>Selected impeller geometric parameters and their deviations</td>
<td>146</td>
</tr>
<tr>
<td>7.2</td>
<td>Experimental response surface model diagnostics</td>
<td>150</td>
</tr>
</tbody>
</table>
Nomenclature

Symbols

\( A \) \quad \text{Area, } m^2
\( A_k \) \quad \text{Axial velocity ratio, } -
\( AR \) \quad \text{Area ratio, } -
\( b \) \quad \text{Height, } m
\( B \) \quad \text{Blockage, } -
\quad \text{Log-law ordinate intercept}
\( c_f \) \quad \text{Skin friction coefficient, } -
\( c_p \) \quad \text{Specific heat capacity at constant pressure, } J/(kg.K)
\( C \) \quad \text{Absolute velocity, } m/s
\( C_p \) \quad \text{Process capability index, } -
\( D \) \quad \text{Pressure recovery coefficient, } -
\( D_f \) \quad \text{Design}
\( DR \) \quad \text{Diffusion Ratio, } -
\( D_f \) \quad \text{Diffusion factor, } -
\( e \) \quad \text{Residual}
\( E \) \quad \text{Energy, } J
\( f \) \quad \text{Sum of body forces}
\quad \text{Function}
\( f_{us} \) \quad \text{Secondary flow tangential velocity parameter, } -
\( g \) \quad \text{Inequality constraint}
\( G \) \quad \text{Machining parameter, } m
\( h \) \quad \text{Specific enthalpy, } J/kg
\quad \text{Equality constraint}
\( i \) \quad \text{Incidence, } ^\circ
\( I \) \quad \text{Rothalpy, } J/kg
\( k \) \quad \text{Von Karman constant}
\quad \text{Number of design variables}
\( K \) \quad \text{Total pressure loss coefficient, } -
\( L_b \) \quad \text{Average path length of impeller flow, } m
\( L_\theta \) \quad \text{Impeller meridional length, } m

\( \dot{m} \) \quad \text{Mass flow rate, } kg/s

\( M \) \quad \text{Mach number, } -
  \quad \text{Inlet Diameter, } m

\( n \) \quad \text{Number of design variables}

\( n_s \) \quad \text{Number of samples}

\( N \) \quad \text{Rotational speed, } rpm

\( p \) \quad \text{Number of regression coefficients}

\( P \) \quad \text{Pressure, } Pa

\( p_c \) \quad \text{Crossover probability}

\( p_m \) \quad \text{Mutation probability}

\( q \) \quad \text{Output quantity, } -

\( Q \) \quad \text{Exit diameter, } m

\( r \) \quad \text{Radius, } m

\( R \) \quad \text{Specific gas constant, } J/(kg.K)
  \quad \text{Correlation factor}
  \quad \text{Trim radius, } m

\( R^2 \) \quad \text{Coefficient of determination}

\( \mathbf{R} \) \quad \text{Correlation matrix}

\( \mathbf{R}_N \) \quad \text{Rotation number, } -

\( R_e \) \quad \text{Reynolds number, } -

\( s \) \quad \text{Specific entropy, } J/(kg.K)
  \quad \text{Tournament size}

\( SS \) \quad \text{Sum of squares}

\( t \) \quad \text{Thickness, } m
  \quad \text{Tip}

\( T \) \quad \text{Temperature, } K

\( TQ \) \quad \text{Stress tensor matrix}

\( U \) \quad \text{Blade speed, } m/s

\( w \) \quad \text{Weight}

\( W \) \quad \text{Relative velocity, } m/s

\( \dot{W} \) \quad \text{Power, } J/s

\( x \) \quad \text{Input variable}

\( \mathbf{x} \) \quad \text{Input variable vector}

\( X \) \quad \text{Area parameter, } -
  \quad \text{Design matrix}

\( y \) \quad \text{Output variable}

\( \hat{y} \) \quad \text{output at an untried value}

\( y^+ \) \quad \text{Non-dimensional distance, } -

\( Z \) \quad \text{Blade number, } -
  \quad \text{Realization of a stochastic process}
**Greek Symbols**

- \( \alpha \): Absolute angle, \(^\circ\)
- \( \beta \): Relative angle, \(^\circ\)
- \( \chi \): Regression coefficients
- \( \Delta \): Difference
- \( \epsilon \): Secondary zone mass flux fraction, 
- \( \phi \): Primary zone area fraction, 
- \( \phi \): Random error
- \( \phi \): Diffuser inclination angle, \(^\circ\)
- \( \gamma \): Specific heat ratio, 
- \( \eta \): Efficiency, %
- \( \eta \): Effectiveness, 
- \( \lambda \): Swirl factor, 
- \( \mu \): Dynamic viscosity, Pa.s
- \( \nu \): Slip or deviation, m/s
- \( \nu \): Mean
- \( \nu \): Kinematic viscosity, m\(^2\)/s
- \( \Pi \): Pressure ratio, 
- \( \theta_k \): Hyperparameter
- \( \rho \): Density, kg/m\(^3\)
- \( \sigma \): Entropy gain function, 
- \( \sigma \): Standard deviation, 
- \( \sigma^2 \): Variance, 
- \( \Sigma \): Summation
- \( \tau \): Shear stress, Pa
- \( \omega \): Angular velocity, rad/s
- \( \xi \): Input quantity

**Subscripts**

- 0: Total or stagnation state
- 1: Impeller inlet
- 2: Impeller outlet
- 5: Diffuser exit
- 7: Volute outlet
- \( \infty \): Refers to tangential velocity for zero slip velocity
- \( a \): Element ‘a’
- \( adj \): Adjusted
- \( b \): Blade
- \( \text{Element ‘b’} \)


**Abbreviations**

ANOVA       Analysis of variance  
CETI        Concepts ETI, Inc (Concepts NREC)
<table>
<thead>
<tr>
<th>Acronym</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CAD</td>
<td>Computer-aided design</td>
</tr>
<tr>
<td>CCD</td>
<td>Central composite design</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational fluid dynamics</td>
</tr>
<tr>
<td>CHP</td>
<td>Combined heat and power</td>
</tr>
<tr>
<td>CNC</td>
<td>Computer numerical control</td>
</tr>
<tr>
<td>DACE</td>
<td>Design and Analysis of Computer Experiments</td>
</tr>
<tr>
<td>DoE</td>
<td>Design of Experiments</td>
</tr>
<tr>
<td>FB</td>
<td>Full blade</td>
</tr>
<tr>
<td>FEA</td>
<td>Finite element analysis</td>
</tr>
<tr>
<td>GA</td>
<td>Genetic algorithm</td>
</tr>
<tr>
<td>HPT</td>
<td>High pressure turbine</td>
</tr>
<tr>
<td>ISA</td>
<td>International Standard Atmosphere</td>
</tr>
<tr>
<td>LHS</td>
<td>Latin hypercube sampling</td>
</tr>
<tr>
<td>LPT</td>
<td>Low pressure turbine</td>
</tr>
<tr>
<td>LSL</td>
<td>Lower specification limit</td>
</tr>
<tr>
<td>MCS</td>
<td>Monte Carlo simulation</td>
</tr>
<tr>
<td>MOGA</td>
<td>Multi-objective genetic algorithm</td>
</tr>
<tr>
<td>MSE</td>
<td>Mean square error</td>
</tr>
<tr>
<td>MTEE</td>
<td>Mitsubishi Turbocharger and Equipment Europe BV</td>
</tr>
<tr>
<td>MTT</td>
<td>Micro Turbine Technology BV</td>
</tr>
<tr>
<td>OEM</td>
<td>Original equipment manufacturer</td>
</tr>
<tr>
<td>PCA</td>
<td>Principal-component analysis</td>
</tr>
<tr>
<td>PDF</td>
<td>Probability distribution function</td>
</tr>
<tr>
<td>PMF</td>
<td>Probability mean function</td>
</tr>
<tr>
<td>PVF</td>
<td>Probability variation function</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds averaged Navier-Stokes</td>
</tr>
<tr>
<td>RSM</td>
<td>Response surface model</td>
</tr>
<tr>
<td>SA</td>
<td>Simulated annealing</td>
</tr>
<tr>
<td>SB</td>
<td>Splitter Blade</td>
</tr>
<tr>
<td>SNR</td>
<td>Signal-to-noise ratio</td>
</tr>
<tr>
<td>SST</td>
<td>Shear stress transport</td>
</tr>
<tr>
<td>TEIS</td>
<td>Two-elements-in-series</td>
</tr>
<tr>
<td>TIT</td>
<td>Turbine inlet temperature</td>
</tr>
<tr>
<td>TU Delft</td>
<td>Delft University of Technology</td>
</tr>
<tr>
<td>UQ</td>
<td>Uncertainty quantification</td>
</tr>
<tr>
<td>USL</td>
<td>Upper specification limit</td>
</tr>
<tr>
<td>VGT</td>
<td>Variable geometry turbine</td>
</tr>
</tbody>
</table>
There is nothing impossible to him who will try.

Alexander the Great
1 Introduction

1.1 Background and Scope

The foremost responsibility of a design engineer is to design a product that performs the specified function efficiently. Modern design methods also necessitate the product to satisfy various other objectives including life, weight, maintainability and lower manufacturing costs. Although a purposeful and an efficient design is of paramount importance, a modern design is not complete if it does not satisfy the above mentioned objectives. Likewise, a manufacturing engineer is responsible to provide the necessary tooling, equipment, operation plan and other technical resources for manufacturing and assembly of a product. A manufacturing engineer is also tasked to ensure that the design is manufacturable, taking into account the production costs and process capability of the manufacturing facility.

Technically, a design represents an application of a procedure or a methodology for detailing the shapes, materials and tolerances of the parts composing a system. The shape of the parts are parameterized into dimensions and provided to the manufacturer as drawings, representing the nominal or basic sizes of the part and other features. Manufacturing, on the other hand, is the application of various processes to alter the geometry, properties and appearance of a given material to achieve the fabrication of the designed part, also including the assembly of different parts forming a system. Ideally, manufacturing should deliver the part with exact dimensions and features as specified by a designer in the part drawing. In reality, however, it is impossible to manufacture the part precisely due to inherent manufacturing uncertainties, which result in systematic and random dimensional deviations in its geometry. These manufacturing uncertainties are the errors which can be caused by any aspect of a manufacturing process such as geometric errors in the production machines, wear in the cutting tools, varying workpiece properties, irregularity in the flow of the coolant, environmental conditions, etc. Consequently, some degree of variation in part geometry has to be allowed by the
CHAPTER 1. INTRODUCTION

Figure 1.1: Example of a typical “over the wall” approach in the industry

The limits defined to regulate such dimensional variations are known as the manufacturing tolerances. High quality products, for instance the products for the aerospace and automotive applications, require stringent tolerances, which can only be achieved by a high process capability. The process capability is a statistical measure of the ability of a manufacturing process to achieve the required tolerance margins, as defined by the design engineer. A functional design (optimum for performance) and a production design (optimum for manufacturing) are, therefore, very much associated and cannot be handled separately [1].

Traditionally, the attitude of the designers has been, “we design it, you build it”. This notion has been termed as “over the wall” approach, where the design engineers are sitting on one side of the wall and throwing designs over to the other side of the wall to the manufacturing engineers. The manufacturing engineers then have to deal with various manufacturing problems, for instance the design not being manufacturable or having too strict tolerances as a consequence of not being involved in the design effort [2]. Manufacturing costs can substantially increase if a complex design carrying stringent tolerance margins is manufactured. Therefore, following such an approach may result in a design that will require a reassessment eventually, for its manufacturability and production costs. Figure 1.1 presents a generic example of the “over the wall” approach in industry, where the design and development of a product take place in various disciplines separated by a virtual wall, representing an insufficient interdisciplinary interaction.

Recent developments in concurrent engineering have led to a greater cooperation between the traditionally separate, design and manufacturing disciplines. Concurrent engineering is being adapted more and more in industry, which represents a philosophy for product design that relies on the design being simultaneously evaluated by the design and manufacturing engineers in order to achieve high quality at low production costs [3].

The goal of this thesis is to support concurrent engineering by designing a robust design methodology, which simultaneously considers the functionality and manufacturability of a product during the preliminary design phase, thus negating the “over the wall” approach. In simple terms, a robust design methodology leads
to a robust product for which the output performance is insensitive to a limited variability of the input variables, without minimizing the sources of variation by incorporating a higher process capability. Applying the robust design methodology to optimize a design would, therefore, considerably improve the product quality in terms of functionality and costs of manufacturing.

1.2 The Gas Turbine Industry Perspective

The gas turbine industry comprises of the original equipment manufacturers (OEM) and the part manufacturers. A gas turbine OEM simply refers to a company responsible for the complete product development and supply to the end user or the customer. For instance Rolls Royce, General Electric and Pratt & Whitney are considered to be gas turbine OEMs since they develop, manufacture and supply gas turbine engines to the aviation and power generation sectors. However, an OEM outsources a lot of its manufacturing to different part manufacturers. Such outsourcing is beneficial for an OEM in saving the set up and overhead costs required for establishing separate manufacturing facilities and know-how for some of their parts, thus saving overall production costs. The part manufacturers have specialized manufacturing skills and process capability, which allows them to manufacture high-quality gas turbine parts at relatively low costs. These include turbomachinery and parts thereof. More advanced part manufacturers also supply complete subsystems, such as the low pressure compressor or fan assembly for a turbofan engine. Moreover, part manufacturers also supply parts directly to the gas turbine users, specially for the older gas turbine engines, which are no longer in production at the OEM. Figure 1.2 presents a typical turbofan engine, and shows the complex architecture of the machine, comprising of hundreds of parts and many subsystems.
CHAPTER 1. INTRODUCTION

The “over the wall” approach in gas turbine industry forms the virtual wall between an OEM and the part manufacturer due to lack of concurrent engineering. In this case, an OEM is responsible for overall gas turbine design and development, including supply to the customer while the part manufacturers only manufacture and supply specific parts to the OEM. Before production, the part manufacturers are provided with the part drawings and tolerance information. The design is assessed by the part manufacturers from a manufacturing perspective and cost analysis is performed. Being high-quality parts for aerospace application, it is highly likely that the designs are complex and specified with strict tolerances. A high process capability is, therefore, required to meet the tolerance margins, which can incur significant manufacturing costs. Most of the time, the design is acceptable for production. However, it is also possible that the manufacturability of some designs is not optimal due to their complexity and unacceptable tolerance margins, thus exceeding the manufacturer’s process capability. Regular dimensional deviations can occur with considerable number of parts being rejected for rework or scrap. To counter the manufacturing problems, often the part manufacturers have to request dimensional and tolerance relaxations for which the OEM has to evaluate and redesign the parts. As the development time is vital, along with the manufacturing costs, any design reconsideration may not be ideal for the OEM.

The success of gas turbine part manufacturers is dependent on their ability to produce high-quality gas turbine parts at the lowest cost. In striving for lower production costs, while maintaining superior quality of the product, it may be worthwhile to focus on the concept phase of the product design and development from a manufacturing point of view. Inclusion of the manufacturing concerns of the part manufacturers in the preliminary design phase could result in a design having superior manufacturability. Applying the concurrent engineering methodology is, therefore, advantageous for the gas turbine OEM and part manufacturers.

1.2.1 Benefits for the Dutch Gas Turbine Industry

Dutch gas turbine industry mainly comprises of part manufacturers, which supply gas turbine parts to the OEM, and to the gas turbine users directly. Dutch part manufacturers such as DutchAero, Aeronamic, Stork Turbo Blading, Sulzer, etc, possess high-tech manufacturing expertise and production facilities for various gas turbine parts. State of the art machining and manufacturing practices allow the Dutch part manufacturers to meet the critical tolerances and high production standards required by the OEM. However, there is an increasing concern in the community regarding tolerance levels demanded by the customers and their effect on the manufacturing costs. The Dutch part manufacturers have, therefore, realized the importance of implementing concurrent engineering together with the OEM in order to achieve a more significant role in the international gas turbine market. Doing so will allow them to comprehensively scrutinize the relationship between functionality, tolerances, manufacturing uncertainty and the resulting deviations, and costs, while optimizing gas turbine parts for better manufacturability.
1.3 Case Studies

A number of cases have been studied from preliminary to detailed level. These include a variety of gas turbine parts, under production by the Dutch gas turbine industry. The following sections present more information on these parts and their evaluation from a manufacturing perspective:

- **Axial turbine seal from Sulzer Eldim.** A case received from Sulzer Eldim BV was to evaluate the effect of manufacturing deviations in an axial turbine seal on the low pressure turbine (LPT) performance. Figure 1.3a shows a segment of the LPT seal manufactured by Sulzer Eldim. The LPT turbine seal forms a peripheral ring of abradable material around the rotating blade tips to minimize the leakage of flow over the rotors. Any manufacturing irregularity can vary the tip-clearance gap between the turbine seal and the rotor, thus influencing the turbine performance in general. Hence, the aim of this study was to investigate the possible correlations between the tip-clearance in the LPT and the overall engine performance. A CFM56-7 turbofan engine was used as a reference and its LPT stage was modeled. Once all the thermodynamic data were available for the LPT, the variation in turbine isentropic efficiency subjected to tip-clearance deviations was evaluated using different empirical correlations available in the literature. Finally, the effect of LPT tip-clearance variation on overall engine performance was determined. More information on this study can be found in ref. [4].

- **Axial compressor casing from DutchAero.** A case was provided by DutchAero BV to evaluate the influence of axial compressor and turbine casing designs on gas turbine performance. Casings are complex designs which have to satisfy many functional requirements such as:
  
  - Containment of the pressurized air and gases in the gas turbine core, as well as the debris, in case of component failure.
  
  - Sustain the reaction forces due to aerodynamic loads from the stator vanes, bearing loads and maneuver loads.
  
  - Support the active clearance control mechanism to achieve optimum tip-clearance by closely matching the thermal and centrifugal growth of the rotor under steady and transient operating conditions.

Moreover, all the above mentioned requirements have to be met with the casing made as light as possible. Figure 1.3b shows an axial compressor casing manufactured by DutchAero. Dimensional deviations in the casings, as a result of manufacturing uncertainties, can affect the gas turbine performance considerably. A comprehensive evaluation was required in this case; however, only a preliminary study was made.
• **Axial compressor blade from Stork Turbo Blading.** A preliminary study was made on an axial compressor blade manufactured by Stork H&E Turbo Blading BV for industrial gas turbine engine application. Figure 1.3c shows various kinds of axial turbomachinery manufactured by Stork Turbo. A design optimization of the axial blades for the front three compressor stages was carried out by the National Aerospace Laboratory (NLR). The effect of design changes on overall engine performance were then evaluated using the Gas Turbine Simulation Program (GSP).

• **Micro centrifugal compressor impeller from MTT.** A recuperated micro gas turbine or a microturbine is under development at Micro Turbine Technology BV (MTT). It produces an electric and thermal power output of 3kWe and 14kW respectively. The microturbine is intended to be used domestically as a micro combined heat and power (CHP) unit and as an auxiliary power unit/parking heater for trucks. An automotive turbocharger has been adapted as the turbogenerator unit for the MTT microturbine application [5].

Centrifugal compressors are largely used in automotive turbochargers, where a high pressure ratio is demanded at low mass flow rates. Figure 1.3d shows a cut out of a typical turbocharger especially the compressor. The main
technical challenges in a turbocharger compressor development are:

- Low Reynolds number flows resulting in significant viscous losses and, subsequently, lower efficiencies than the larger equivalents.
- Manufacturing issues due to small component sizes, strict tolerance margins, reliability requirement and cost effectiveness.
- Large heat transfer from hot to cold sections, which affects the overall turbocharger performance.

Keeping in view the challenges mentioned above, quality demands that a turbocharger compressor is manufactured with high precision in order to meet the strict tolerance margins and, therefore, lower the variation in performance due to manufacturing deviations. However, manufacturability of the turbocharger compressor requires the production costs to be low. In order to satisfy both the performance and manufacturability requirements, a detailed evaluation has been performed on the turbocharger compressor from a manufacturing perspective. The focus has been given to the impeller since it is a complex design to manufacture, and also the component responsible for energy transfer to the flow in a centrifugal compressor. A robust design optimization has been carried out, following a comprehensive manufacturing uncertainty quantification, using different performance modeling tools and statistical methods. A robust impeller design can permit a definition of wider tolerances for critical dimensions, since variation in performance due to manufacturing deviations is reduced without incorporating a higher process capability. Overall quality is expected to increase, along with a reduction in production costs of the turbocharger using the robust design methodology.

1.4 Thesis Outline

In this chapter, the importance of concurrent engineering and its application in the gas turbine industry has been introduced. The scope and motive of the study have been described along with the information about the test cases supplied by various gas turbine part manufacturers located in the Netherlands. The micro centrifugal compressor impeller case from MTT has been selected for detailed evaluation and the thesis presents different sections of this evaluation.

In chapter 2, the problem has been formulated in detail. The main characteristics of a turbocharger have been illustrated, along with their novel application in a microturbine. The manufacturing processes involved in the production of a turbocharger compressor have been described. The manufacturing uncertainties and their sources have been illustrated. Lastly, the methodology for a robust compressor design has been explained, along with the tools and techniques considered to evaluate, and counter the influence of manufacturing uncertainties on compressor performance through uncertainty quantification (UQ).
In chapter 3, the complex fluid physics attributed to centrifugal compressors has been illustrated, and a one-dimensional (1D) meanline model for simulating the centrifugal compressor performance has been described in detail. The 1D meanline model has been used to calculate the performance map of the baseline turbocharger compressor using its basic geometric information. The advantages and disadvantages of the 1D meanline model have been identified by comparing the 1D results with the available test data. The last part of the chapter presents a study on losses related to different components in the turbocharger compressor.

In chapter 4, the comprehensive three-dimensional (3D) evaluation of the turbocharger compressor using computational fluid dynamics (CFD) has been detailed; from pre-processing of the compressor geometry to the solution and post-processing of the results. The CFD evaluation has been used to study the flow properties of the turbocharger compressor in detail and improve the 1D meanline model by correcting its empirical modules using the CFD drawn results. The 1D meanline model did not predict the correct variation in the compressor performance caused by selective deviations introduced in the impeller geometry. Therefore, the 1D model has been discarded from further application, with the meta-model recommended as a suitable replacement to perform the UQ and robust design optimization.

In chapter 5, quantification of the influence of manufacturing uncertainties on turbocharger compressor performance has been described. Two widely used metamodels — response surface model and kriging model, have been evaluated and described in this chapter. The models have been constructed for the turbocharger compressor and evaluated against the CFD sensitivity data. The response surface model has been selected in the end and used to perform the manufacturing uncertainty quantification by applying the Monte Carlo simulation (MCS).

In chapter 6, a robust design optimization of the impeller has been illustrated. The optimization has been performed by integrating an optimization algorithm (multi-objective genetic algorithm in this case) with the metamodel based MCS setup. The optimization process aims at reducing the variation in the impeller performance due to manufacturing deviations in the geometry, while maintaining or improving the nominal performance. Three robust impeller designs have been evaluated in the end and compared with the baseline impeller to reveal the relative advantages and disadvantages.

In chapter 7, an experimental validation of the modeling of impact of manufacturing uncertainties on turbocharger compressor performance has been detailed. The experimental validation has been carried out by testing a number of impeller samples as part of a design of experiments (DoE) on a turbocharger test bench. The experimental evaluation has given interesting results and a reasonable validation for modeling of manufacturing uncertainties.

Finally in chapter 8, general conclusions have been made, along with the recommendations for future analyses. The chapter concisely mentions the accomplishment of the research objective, added value and application of the work in support of concurrent engineering.
References


2

Micro Centrifugal Compressor Impeller

2.1 Preface

The association between the design and manufacturing engineering disciplines has been described in further detail in this chapter. The origin of manufacturing uncertainties, their management through tolerance specification, and the overall impact on manufacturing costs has been scrutinized. Moreover, a comprehensive methodology has been devised to quantify the effect of manufacturing uncertainties on the turbocharger compressor performance, and eventual robust optimization of the impeller design.

2.2 Turbocharger Compressor Case

Small-scale turbomachinery have miniature dimensions and tolerance specifications. High-precision manufacturing is, therefore, required to produce them. For a turbocharger compressor, a number of manufacturing processes are involved in its production, all of which can induce dimensional deviations caused by the manufacturing uncertainties. Introduction to the turbocharger compressor and its novel application in a microturbine is necessary. The following sections describe the application in a brief detail.

2.2.1 Introduction to Turbochargers

Automotive turbochargers are used to compensate for the performance trade-off in the internal combustion (IC) engines, as they are downsized in order to meet the stringent emission regulations. Although being a small machine consisting of
a few components, the turbocharger is a critical subsystem, which determines the final output performance of a downsized IC engine.

Basic Operation

Turbochargers are used to charge pre-compressed air into an internal combustion engine with an objective to boost its power output and fuel economy. A turbocharger mainly consists of a single compressor and turbine coupled on a common shaft. The turbine uses the extra energy in the hot exhaust gas (that would normally be wasted) to drive the compressor, which in turn, supplies compressed air to the engine cylinders. Figure 2.1 gives an overview of a typical turbocharger for automotive applications, highlighting its different components and gas path.

Turbomachinery

Radial or centrifugal type turbomachinery (for both compressor and turbine) is widely used for automotive turbochargers. A single-stage subsonic centrifugal compressor can be designed to operate at any pressure ratio up to 3.5:1 with reasonable efficiency and good reliability, out of heat-resistant aluminum alloys [1]. Axial compressors are more efficient than the centrifugal type. However, for a similar pressure ratio, an axial compressor will be longer in construction (due to multi-stage compression), heavier and more expensive to build. Moreover, axial compressors have a comparatively smaller operating range. Similarly, centrifugal type turbines are mostly used in small-scale turbochargers due to their simplicity and cheaper manufacturing; provided that the impeller wheel can be cast. Axial flow turbines are generally not used in automotive turbochargers for being less efficient due to large tip-leakage flows and boundary layer blockage. However, the Honeywell dual-boost turbocharger concept [2] uses an axial turbine, and delivers a better transient response time than its centrifugal counterpart, but at an expense of reduced peak turbine efficiency.

Bearing Configuration

The bearings are mounted inboard, located towards the center of the shaft with the compressor and turbine impeller wheels overhanging at each end of the shaft. Such arrangement is simple, light and cheap for automotive turbocharger applications. The center casing or the cartridge houses the two bearings, and a lubrication oil inlet and drainage system. A disadvantage of this layout is the inherently unstable running of the rotor assembly due to the short distance between the bearings and the heavy weight of the overhanging turbine wheel. Hence for such an assembly, well-damped bearings and careful balancing of the rotor assembly is required.

Both sleeve and rolling bearings are used, depending on the nature of the application and the user’s preference. For automotive applications, the sleeve bearings (also known as journal bearings) are widely used since the low-cost rolling bearings cannot meet the durability requirements at very high speed operations.
The sleeve bearings have an almost indefinite life, are capable of accepting larger out-of-balance loads, and are insensitive to vibrations and shocks transmitted from outside. However, large oil flow at high pressure is needed for sufficient lubrication and cooling. Additionally, a single thrust bearing is also installed in an automotive turbocharger for managing the axial loads. Lubrication is required for the thrust bearing as well, and it is normally located inboard of the impeller but outboard of the journal bearings.

**Auxiliary Components**

An exhaust waste gate is typically used in automotive turbochargers to control the boost pressures in an engine. It consists of a simple flap valve, controlled by a pneumatic actuator or an electronic boost control system, allowing the engine exhaust gas to by-pass the turbine. The output power is, thereby regulated by limiting the turbine flow rate. By-passing the exhaust gas allows an application of a smaller turbocharger (particularly the turbine), which is able to provide sufficient boost at low speed when the waste gate is closed. A smaller turbocharger also reduces the lag due to its low inertia, especially when the waste gate is closed during acceleration. However, waste gate systems are costly and potentially unreliable due to the very high temperature, transient and corrosive environment they operate in. Apart from the waste gate systems, a variable geometry turbine (VGT) is also used to control the boost pressures. A VGT matches itself to the exhaust gas flow rate at any operating condition, thereby not allowing a loss of energy as in a waste gate system. A variable guide vane assembly is commonly used to control the turbine power in a VGT configuration.
2.2.2 Introduction to Micro Gas Turbines

Advances in energy technology and the general trend towards smaller unit sizes of power plants have led to an increased interest in micro cogeneration or small combined heat and power generation (CHP) units, with the hope of ultimately developing units that can provide electricity and heat for individual buildings [3]. A micro CHP is a process of producing both electricity and usable thermal energy (heat and/or cooling) at high efficiency, and near the point of use [3, 4]. The technological core of micro cogeneration system is an energy conversion unit; which can be an IC engine, stirling engine, fuel cell, organic Rankine cycle system or a micro gas turbine, that allows the simultaneous production of electricity and heat. In addition, various other components are also included in a micro cogeneration system, such as a well developed grid access, including possible monitoring and control devices.

Micro gas turbines or microturbines in short, are small-scale energy conversion units developed from technologies originally used in auxiliary power systems for aircrafts or automotive turbochargers. A microturbine can be divided into three primary systems:

- Mechanical system – turbine, compressor, generator and recuperator.
- Fuel system – fuel delivery and combustion chamber.
- Control system – main control software, inverter and power firmware.

Microturbines have an advantage over the piston engine generators in terms of higher power to weight ratio, extremely low emissions and fewer moving parts. They accept most commercial fuels, such as natural gas, propane, diesel and kerosene. A microturbine thermodynamic cycle is similar to a conventional gas turbine. The inlet air is first compressed in the compressor and preheated in the recuperator using heat from turbine exhaust. Heated air from the recuperator is then mixed with fuel in the combustor and burned. The hot combustion gas is expanded in the turbine, which produces the mechanical power to drive the compressor and the electric generator. The waste heat in the exhaust is subsequently supplied for domestic or light industrial thermal needs.

Microturbines operate at high rotational speeds and lower pressure ratios compared to larger gas turbines. Achieving the efficiency levels closer to the larger industrial gas turbines is the development target. Simple scaling of larger turbomachinery would not suffice for a microturbine application. As the size of the turbomachinery decreases, different technical challenges originate, which are related to manufacturing, materials, bearings, and high heat transfer due to large temperature differences over small distances. Particularly below 100kW, many developments have failed to obtain sufficient efficiency, reliability and cost effectiveness to be successful for the market [5].

Turbine inlet temperature (TIT) in a microturbine is generally limited to 950 °C or below to enable the use of relatively inexpensive materials for the turbine.
2.2. TURBOCHARGER COMPRESSOR CASE

(b) Microturbine demonstrators

Figure 2.2: MTT microturbine for CHP application

- Demand for higher thermal efficiency and limitation of TIT dictates the requirement of an efficient compressor. A single stage centrifugal compressor is an ideal choice for microturbine application due to low its manufacturing cost, design simplicity, compactness and performance characteristics, such as wide surge margin with high tolerance to inlet flow distortion [6].

2.2.3 Turbocharger Adaptation for MTT Microturbine

Development of efficient turbomachinery optimized for a particular cycle is very expensive, and in the micro power generation market, can only be justified with very large production volumes [5]. Turbocharger quality and performance has consistently been improved over the past decades. The similarity of a turbocharger to the radial turbomachinery configuration required in a microturbine makes it a readily available option for new companies entering in the micro cogeneration market, thus saving valuable time and development costs.

MTT has employed a commercial turbocharger in a microturbine CHP unit for domestic and vehicular applications. Figure 2.2 shows the MTT microturbine
demonstrator and the modified turbocharger integrated with a generator. The microturbine will be able to produce 3kWe of electric power along with 14kW of usable thermal power. The target turbo generator efficiency is 16%, which is largely dependent on the turbomachinery efficiencies, recuperator effectiveness and the TIT.

Although the rotating assembly of a turbocharger has much in common with the single shaft radial flow recuperated microturbine, the initial design requirements are significantly different, hence impairing the basic concept of convertibility [7]. The performance demanded from a turbocharger is governed by the vehicle engine specifications. A wide surge margin and flow range is usually required from a turbocharger to ensure normal operations during rapid acceleration/deceleration. On the contrary, the performance required from a microturbine is controlled by the connected generator. The turbogenerator in a microturbine should also have an adequate surge margin to handle rapid load changes. However, a wide flow range is not a strict design requirement as the microturbine is intended to operate at peak efficiency, away from surge limit most of the time.

Modern day turbocharger compressors have reached efficiencies of 75% while the TIT of 1050° C is possible with advanced materials [8]. However, because the turbocharger compressor is developed for a wider flow range application, the design could be adapted for more efficiency. An attempt has been presented in ref. [9] to gain more efficiency from the turbocharger compressor by applying and optimizing a set of vanes in the diffuser.

### 2.3 Sources of Manufacturing Uncertainties

All manufacturing processes introduce manufacturing uncertainties, which appear in form of dimensional deviations in a product, having different sources of origin. However, before coming to the sources of manufacturing uncertainties, some fundamentals of manufacturing have to be described and understood. The following sections illustrate different aspects of manufacturing in some detail.

#### 2.3.1 Defining Manufacturing

In technological terms, manufacturing is an application of physical and chemical processes to alter the geometry, properties and appearance of a given starting material to a pre-defined state. It also includes the assembly of multiple parts to build the final product. Manufacturing processes employ a combination of machinery, tools, power and labor. Moreover, manufacturing is always carried out as a sequence of operations, where each operation brings the material closer to the desired final state. In economic terms, manufacturing is a transformation of materials into items of greater value by means of one or more processing and assembly operations. Hence, manufacturing adds value to the material by changing its shape or properties.
2.3. SOURCES OF MANUFACTURING UNCERTAINTIES

2.3.2 Manufacturing Process

A manufacturing process is a designed manufacturing procedure, that results in physical and chemical changes to a starting work material. The intention is to increase the value of the work material by changing its geometry and properties. Manufacturing process consists of processing operations and assembly of components. A processing operation transforms the work material from one state of completion to a more advanced state that is closer to the final desired product. A processing operation uses energy (mechanical, thermal, electrical and chemical), which is applied in a controlled way by means of machinery and tooling. Human energy is also required to control the machines, oversee the operations, and load and unload parts before and after each cycle of operation. Ultimately, a finished part exits the process. An assembly operation, on the other hand, joins two or more components to create a new entity, called an assembly, sub-assembly, or some other term that refers to the joining process either permanently or semi-permanently (e.g., a welded assembly). Permanent joining processes include welding, brazing, soldering and adhesive bonding. The use of screws, bolts and other threaded fasteners are the traditional methods in the mechanical assembly category. More permanent mechanical assembly techniques include the use of rivets, press fitting, and expansion fits.

2.3.3 Manufacturing Capability

A manufacturing plant generally facilitates the application of many manufacturing processes and systems designed to transform a certain limited range of materials into products of increased value. Manufacturing capability of a manufacturing plant refers to the technical and physical limitations in processing capability, product size and production capacity. Technological processing capability of a plant is determined by its available set of manufacturing processes, which can fabricate a part precisely according to design requirements. A plant with a given set of processes is also limited in terms of size and weight of the products that can be accommodated. For instance, a plant must be equipped with cranes to move large and heavy products. Larger machines are also used to process larger parts. Moreover, a plant is limited by the number of parts that can be produced in a given time period. This limitation in quantity is commonly called the plant capacity or production capacity, defined as the maximum rate of production that a plant can achieve under given operating conditions.

2.3.4 Manufacturing Costs

Manufacturing processes require many different inputs for processing a given material to fabricate and assemble a part. All of these inputs add up as costs of a manufacturing process. Some of these inputs include the costs of material, overhead cost of the machines, tooling, quality inspection and scrapping. Materials
Normally have the most contribution to the manufacturing costs. Material properties also determine the manufacturing processes to be used, thus influencing the costs per process used. Manufacturing capability is improved by applying machines with higher manufacturing precision. However, such machines are expensive to acquire and maintain. Costs of fixture and special tooling also have a significant impact upon overall costs of manufacturing. For quality control purposes, more frequent sampling is necessary, which results in an increase in the overall costs. Any scrap produced during manufacturing will directly affect the manufacturing costs as well.

2.3.5 Manufacturing Uncertainties

Uncertainties can be classified into two groups — *aleatoric* uncertainties, which are random or stochastic in nature, unpredictable, and cannot be controlled, and *epistemic* uncertainties, which are systematic and can be controlled. The hysteresis error of a pressure sensor and random variations of the chip-forming process are two examples of random or aleatoric uncertainties. Aleatory uncertainty is normally characterized by probabilistic approaches, where it can be represented by a finite number of random variables with some known probability distribution. The systematic or epistemic uncertainties arise solely due to lack of knowledge regarding the probability distributions of the quantities of interest. It is typically specified as intervals, estimated as the lower and upper bounds of the input parameters through expert opinion, in conjunction with the available data. Consequently, the epistemic uncertainty can be considered as providing (conservative) bounds on an underlying aleatory uncertainty \([10]\). Epistemic uncertainty can be quantified by sampling based approaches. An example of the epistemic uncertainty is the systematic shift in part dimensions due to tool and fixture wear over a period of time, which can be measured and compensated for.

A designer would like the part to be manufactured exactly as it has been designed. Dimensions are the linear or angular sizes of a part specified in a part drawing, illustrating its nominal shape and features. However, it is impossible to manufacture a part precisely due to uncertainties in the manufacturing processes. Uncertainties appear as dimensional deviations in the part, and eventually propagate as performance or functional variations during operation. Manufacturing uncertainties are both aleatory and epistemic and can be caused by almost any aspect of a manufacturing process. For instance, manufacturing uncertainties in a casting process can be caused by shrinkage defects, gas porosity, pattern wear, improper core, etc. Similarly the sources of variation during machining of parts can be due to small errors in the machine, cutting tool wear, fixture location and wear, change in the work piece physical and chemical properties, coolant flow variation and degradation, operator or human error, environmental conditions, and so on. Apart from manufacturing uncertainties, there are also operational uncertainties in parts which develop over time due to the operating conditions, shape deflections caused by external damages, repairs, etc.
2.3. SOURCES OF MANUFACTURING UNCERTAINTIES

Since manufacturing uncertainties cannot be eliminated, tolerances are defined to control or limit them. American National Standards Institute (ANSI) defines tolerance as “the total amount by which a specific dimension is permitted to vary. The tolerance is the difference between the maximum and minimum limits.” [11]. If the dimensional deviations are maintained within the specified tolerances, the manufactured parts are functionally acceptable to the designer. The ability to achieve a certain tolerance is, however, a function of the process capability of the manufacturer.

2.3.7 Process Capability

All manufacturing processes produce dimensional deviations in manufactured parts. The deviations, if measured for each dimension, can be represented by a statistical model. A common statistical model used to describe manufacturing uncertainties is a normal or Gaussian distribution, as shown in Fig. 2.3. The mean $\mu$ of the distribution marks the highest point on the curve and tells how close the process is to the target dimension, while the spread of the distribution, expressed by its standard deviation $\sigma$, indicates the precision or process capability. A manufacturing process is said to be in a state of control if the statistical distribution of a dimension, represented by $\mu$ and $\sigma$, does not change over time.

In Fig. 2.3, parameters USL and LSL mark the upper and lower specification limits, respectively for part dimensions, as set by the design requirements in shape of tolerances. If USL and LSL correspond to the $\pm 3\sigma$ process capability, a few parts will be rejected (about 3 per 1000). Here, process capability can be defined...
as the ability of a manufacturing process to produce a part within the specification limits or tolerances set by a designer. A high level of process capability, in conjunction with a stringent quality assurance system, will result in a consistent production of good quality products, thereby reducing the costs of scrap and rework of substandard parts, wasted materials and labor hours. Process capability is measured by the process capability index \( C_P \), which is a widely used criterion in the manufacturing industry and given as,

\[
C_P = \frac{\text{allowable process spread}}{\text{actual process spread}} = \frac{USL - LSL}{6\sigma} \tag{2.1}
\]

A \( C_P \) of 1.0 indicates that a process is “capable”. However, customers can demand higher levels of quality such as the Six Sigma, for which a minimum \( C_P \) of 2.0 is required.

### 2.3.8 Design for Manufacturability

A good design not only delivers functionally, it is also easily manufacturable and economical to produce. The challenge is to counter the manufacturing uncertainties and the resulting dimensional deviations, which are in turn, deeply associated to the product application and design methodology. In support to design for manufacturability, some guidelines can be outlined:

- **Simplicity.** A product having a simple shape, least number of parts and with fewer precision adjustments would be the least costly to produce. Therefore, one of the key requirements for concurrent engineering is to incorporate an easy to manufacture design. However, for high performance designs, such as in aerospace and automotive industry, design complexity is inherent and unavoidable.

- **Moderate tolerances.** Tolerances have a profound effect on the selection of manufacturing processes, functional quality of a product, and the overall production costs. It would be advantageous, therefore, to identify and classify various part dimensions according to their sensitivity to output performance. Doing so, will prevent unnecessarily strict tolerance allocation for less sensitive parameters.

- **Avoiding secondary operations.** Complex designs, allocated with strict tolerances, normally result in an increase in part rejections unless sufficient process capability is ensured, for e.g., Six Sigma. Many of the rejected parts will eventually undergo refurbishing by applying secondary manufacturing operation, thus incurring additional manufacturing costs. Therefore, avoiding secondary operations is a requirement for design for manufacturability.

- **Know your manufacturer well.** If the capabilities and limitations of the manufacturer are realized, an ideal producible product can be designed. An effective teamwork is, however, necessary in this case.
2.4. ROBUST DESIGN

These guidelines are very much associated to each other, and would be applicable to nearly all manufacturing processes in support of concurrent engineering and design for manufacturability. As mentioned, fulfillment of most of these guidelines may not be possible for high performance designs. Nonetheless, robust design engineering provides an opportunity to counter the influence of manufacturing uncertainties by designing a smart product, which may be sufficiently immune performance-wise, to a limited variation in geometry.

2.4 Robust Design

By robustness, it is meant that the product or a process performs consistently on target and is relatively insensitive to factors that are difficult to control [12]. Controlling the variability in output performance against uncertain inputs by employing tolerances is a commonly used technique, and is known as the method of tolerance design. On the other hand, robustness introduces flexibility in the design of a system by exploiting the input parameters, such that the sensitivity gradients are reduced. This difference can be explained by the variation transmission equation given below, provided the set of input parameters \( x = x_1, x_2, \ldots x_k \) are statistically independent:

\[
\sigma_y^2 = \sigma_{x_1}^2 \left( \frac{\partial y}{\partial x_1} \right)^2 + \sigma_{x_2}^2 \left( \frac{\partial y}{\partial x_2} \right)^2 + \ldots + \sigma_{x_k}^2 \left( \frac{\partial y}{\partial x_k} \right)^2 \tag{2.2}
\]

Equation 2.2 has been derived from the Taylor series expansion, which is one of the regularly used methods to propagate uncertainty of input parameters \( x = x_1, x_2, \ldots x_k \) into a system for evaluating the variability in output performance \( y \) (where \( y = f(x) \)). The equation shows that the output variance \( \sigma_y^2 \) is proportional to the variance of the inputs \( \sigma_x^2 \) and the sensitivity gradients \( \partial y / \partial x \). Assigning tolerances to the input parameters necessarily reduces the variance of the input parameters, but at increased manufacturing costs. However, by reducing the sensitivity gradients, variance of the output performance can be limited without controlling the input uncertainties through strict tolerances.

Figure 2.4 illustrates the difference between sensitive and robust designs. The sensitive design represents the probability distribution of a generic output parameter obtained via propagation of uncertainties assimilated in its input parameters. The design shows a large output variation, which could result in a good number of parts to exceed the permissible limits, thereby causing part rejections due to lack of sufficient functionality. Improving the quality by defining stricter tolerances and/or secondary manufacturing operations can reduce the number of defective parts, but at an added cost. This is obviously not an ideal situation for any product developer. On the contrary, a robust design will ensure reduced output variability with similar magnitudes of uncertainty in the input parameters. More and more parts will be able to deliver performance close to the target, thus reducing the number of defective parts substantially. Furthermore, a robust design
Figure 2.4: Output variability obtained from sensitive and robust designs also allows tolerance relaxation since the output variability is being regulated by reduced sensitivity gradients.

2.4.1 Taguchi Methods

It is important to mention the work of Genichi Taguchi towards uncertainty management and robust design of a system. Taguchi methods [13] are commonly used to reduce the variations in product performance. The methods consist of three stages – system design, parameter design and tolerance design. A preliminary system configuration is conceptualized in the system design phase. This phase is innovative as new methods and technologies are explored, which would eventually become a part of the system. The parameter design stage, also referred to as robust design stage, involves the identification of the sensitive parameters, which influence the system performance to uncertainties. A signal-to-noise ratio (SNR) is used to define a quantitative measure of the performance variability. With maximization of SNR, a reduction in output performance variation can be obtained. Finally, the tolerance design phase includes fine-tuning of the optimal design obtained in the parameter design stage.

Taguchi methods are widely accepted and used in industry for uncertainty management. However, the technique of modeling SNR has been criticized for leading to non-optimal solutions. The summation of data into a single SNR function, and the consequential loss of important information regarding the influence of individual uncertainty sources on variability in performance, has been considered a shortcoming of Taguchi methods.
2.4. ROBUST DESIGN

2.4.2 Generic Methodology for Robust Design

A three-step generic methodology has been conceived to achieve design for robustness against manufacturing uncertainties. Figure 2.5 illustrates the proposed methodology and its description has been detailed in the following sections of the chapter.

Sensitivity Analysis

It is important to identify the input parameters having the most influence on system output. The objective is to select the most sensitive input parameters for further evaluations, which can potentially save a designer from unnecessary design effort. Furthermore, a sensitive input parameter that also carries most of the uncertainty is important for robust design, compared to a non-sensitive input parameter with a similar magnitude of uncertainty. A parameter-wise sensitivity evaluation is needed for this purpose. Sensitivity analysis investigates the connection between inputs and outputs with an objective to identify how the variability in an output quantity of interest $q$ is connected to an input $\xi$ in a system. This exercise determines the sensitivity derivatives $\partial q/\partial \xi$ for each input. A sensitivity ranking is finally built to identify and select the most sensitive input parameters, dominating the system response.

Uncertainty Quantification

The sensitivity analysis is followed by an uncertainty quantification (UQ), which measures the variability in the output caused by the presence of manufacturing uncertainties in the inputs. UQ initiates with data assimilation process, where the uncertainties in the inputs are characterized. Fundamentally, data assimilation consists of a study of the system of interest that aims at identifying the properties, physical processes and other factors required to fully characterize it [14]. Data related to manufacturing uncertainties is accumulated and represented in form of an explicit probability distribution.

![Figure 2.5: Generic three-step robust design methodology](image)
Following data assimilation, the input uncertainties are required to be propagated, as illustrated in Fig. 2.6, through the system for modeling the output variability. Monte Carlo simulation (MCS) is a widely used uncertainty propagation technique for simulating a random process, given the stochastic properties of one or more random variables, with a focus on characterizing the statistical nature (mean, variance, range, distribution type, etc) of the responses (outputs) of interest [15]. Monte Carlo methods have long been recognized as the most exact method for all calculations that require knowledge of the probability distribution of responses of uncertain systems to uncertain inputs [16]. To implement a MCS, a pre-defined number of input samples (100,000 samples generally recommended for a good resolution) are randomly generated and analyzed in a model, which describes the complete system characteristics. The model determines the output performance variation represented by its respective probability distribution. Necessary statistical analysis is eventually performed to determine the output mean and variability either as standard deviation or variance.

Similar to the sensitivity analysis, it is important to determine which of the input parameters have contributed the most to variability in the output. The analysis of variance (ANOVA) is a commonly used method to identify the most uncertain input parameters by decomposing the variance in a measured outcome to the input sources. A selection can then be made for the input parameters that accommodate the most sensitivity and uncertainty at the same time for the robust design optimization.
Robust Design Optimization

Application of the robust design methodology leads to a robust product or a robust process for which performance is insensitive to a limited variability of given sources without eliminating those sources [17]. Traditional methods of deterministic design ignore the presence of uncertainty, which can lead to undesirable variations in the design performance [18]. Robust design optimization can be carried out using a stochastic optimization algorithm coupled with a model-based Monte Carlo simulator. MCS are performed on each individual design produced by the optimizer and mean and variability of the output parameter of interest are obtained. Robustness is achieved when a configuration is obtained that minimizes the output variability, while delivering a satisfactory mean output, as illustrated in Fig. 2.4. The design satisfying the functional requirements, along with the specified constraints is finally selected.

2.5 Robust Design of the Impeller

Several studies have been performed on axial turbomachinery to quantify the effects of manufacturing uncertainties on performance using probabilistic techniques. Effects of manufacturing tolerances on gas turbine cooling have been studied in ref. [19]. The study has been performed to identify and quantify the effects of manufacturing, as dictated by the tolerances allowed in the finished product, over the resulting cooling design of a high pressure turbine (HPT) airfoil. A comprehensive Monte Carlo analysis with 20,000 trials has been made using a simplified cooled airfoil model involving one-dimensional (1D) heat transfer modeling. As a result, the probabilistic distribution of maximum metal temperature variation is determined. In ref. [20], a probabilistic methodology to quantify the impact of geometric variability on axial compressor aerodynamic performance has been presented. The principal-component analysis (PCA) based blade geometric model has been coupled with a quasi-two-dimensional (Q2D) compressible viscous blade-passage analysis tool. Later in ref. [21], three different compressor airfoils have been optimized both deterministically for minimum loss, and stochastically for minimum mean loss and minimum loss variability. The stochastic airfoil designs gave 30% to 40% lesser polytropic efficiency variability. A similar study has been performed in ref. [22], but using a metamodel-based robust design optimization. Another robust design optimization study has been presented in ref. [18, 23], involving a metamodel-based Bayesian MCS applied on two-dimensional (2D) axial compressor blades.

It can be observed that due to the tremendous computational costs involved in carrying out an uncertainty propagation with MCS, industries and academia have been focusing on the application of low-fidelity models. Metamodels that are constructed over the responses obtained from the high-fidelity models, following a dedicated design of experiments (DoE), are regularly used for deterministic design optimization problems.
2.5.1 Impeller Manufacturing Processes

Turbocharger compressor impellers are generally manufactured by an aluminum die casting process. This method is a molding process in which molten aluminum is injected into cavities formed by reusable molds called dies, and allowed to solidify. Aluminum alloys have low density, good corrosion resistance, are relatively easy to cast, and have good mechanical properties and dimensional stability. Typical die casting process involves various machines consisting of several elements, such as the die mounting and clamping system, die, metal pumping and injection system, metal melting and storing system, and any auxiliary equipment for mechanization of such operations as part extraction and die lubrication [24].

The process begins with the fabrication of an impeller master model, machined with high precision as shown in Fig. 2.7a. The master model is used to prepare the die, which can either be a rubber or a metal female mold depending on the...
2.5. ROBUST DESIGN OF THE IMPELLER

Figure 2.8: Illustration of different machining operations on the impeller

manufacturer’s preference. A metal die is shown in Fig. 2.7b. Replicas or patterns are produced using the die by filling the female mold with silicone rubber or wax. These patterns have a shape identical to the master impeller as shown in Fig. 2.7c. The pattern is then placed in a frame and filled with slurry of water and plaster. Plaster is an inorganic substance, to which if water is added and stirred, becomes hardened due to cement hydration reaction. The pattern is later removed and the plaster casting die is recovered as shown in Fig. 2.7d. Low-pressure casting process is generally applied to cast the impeller using the plaster die. Air pressure is applied to the surface of the melt that is contained and heated in a sealed vessel in the lower portion of the plaster casting mold. The melt is forced through a stalk connected to the mold and inserted into the melt allowing the melt to flow up into the mold. Simple gravity die casting is also commonly used. After cooling, the plaster is removed from the metal casting. The impeller castings are then quenched and then trimmed to remove the unnecessary material as shown in Fig. 2.7e. A raw impeller casting, as shown in Fig. 2.7f, is finally obtained.

Secondary machining and surface finishing operations may also be performed as illustrated in Fig. 2.8. These operations involve machining of the shaft hole, backface machining, boss cutting (boss is the protrusion in front of the impeller casting) and machining of the lateral surface of the blades to achieve the required impeller trim. Moreover, the impellers are also rotationally balanced by removing some amount of material usually from the boss. For the turbocharger application, a single type of die is commonly used to cast impellers of a standard shape and size. The standard size casted impellers are later trimmed using appropriate machining to obtain impellers with different flow capacities and pressure heads. Two types of trims are normally performed on the casted impellers:
• Height Trims to obtain impellers with different flow capacities by removing excess material from the impeller blade tips.

• Diameter Trims to allow the same casting to be used for lower boost pressure applications.

Impeller blades are generally tapered for structural reasons. As the impeller height is progressively trimmed, the average thickness and hence the blade blockage increases. This eventually can impact the impeller efficiency sufficiently enough to require an additional casting with thinner blades. In addition, the diameter trim will impact both the blade thickness at the trailing edge and the backsweep angle. Therefore, if the diameter trims are required, both the blade angle and thickness variations must be considered during impeller design.

Fine-grained structure with a minimum amount of porosity and good mechanical properties can be achieved from die casting of small turbocharger impellers. For products like impellers, die casting holds an economic advantage with lower cycle or processing time. Thin blades with a high quality surface finish can be reliably and economically manufactured by using the rubber patterns, in place of the wax patterns used previously for casting. The flexibility of rubber allows it to be pulled even from the complex dies. One rubber pattern can be used for the production of many moulds before the accumulated wear takes it out of tolerance. Therefore, the rubber pattern has to be replaced in the stipulated time for precise manufacturing.

The complex geometry and very high running speeds of impellers create high stresses at locations such as blade roots and around the bore. With increasing angles of backsweep, higher blade speeds are required to maintain the pressure ratio. Hub failures are most likely to occur through low cycle fatigue. For cast impellers, a high quality of casting, inspection techniques to eliminate castings with imperfections, and an excellent quality of bore finish are all important to life [25]. Therefore, it is vital to minimize the defects while manufacturing the turbocharger compressor impellers.

Impellers for turbocharger compressors are also manufactured by machining from forged billets. Despite the advancements in casting technology, impellers manufactured through machining have better and consistent finishing, along with better quality control of the finished part. However, all the surfaces of the impeller including the complex blade designs have to be machined individually, which is a time consuming and costly process. Hence, for production in large numbers, machining is not an ideal choice. Nonetheless, the fatigue life of machined impellers is substantially better than those produced by castings. Machining is also suited for fabricating large high-performance centrifugal compressors, where the structural integrity (stress, fatigue, and frequency response) demands the use of forged billets.

Since many processes are involved in manufacturing of the turbocharger compressor impellers, it can be established that the underlying requirements — small size, strict tolerances and quality while having low production costs, necessitate
the impellers to be designed for high manufacturability. A robust impeller design will, therefore, be a favorable solution in the presence of these manufacturing uncertainties.

2.5.2 Performance Modeling

For the turbocharger compressor under focus, the three-step robust design methodology proposed in section 2.4.2 has been applied using different modeling methods briefly described in the following sections of the chapter.

High Fidelity Modeling

As a first step, the complete module or a relevant part of the module containing the component of interest requires an analysis in detailed or high fidelity models, for instance, computational fluid dynamics (CFD) and finite element analysis (FEA) models. For the test case compressor, CFD has been used effectively to model the performance and internal flow structure of the compressor. CFD tools are a great help for studying the structure of a flow field and deducing details concerning the creation of losses, blockage, and deviation in turbomachines [26]. Compressor map has been simulated and compared with the available experimental data for validation. Apart from modeling the overall compressor performance, the internal flow structure has been studied in detail, especially in the impeller passage. The flow in the impeller passage is complex and highly three-dimensional (3D) due to strong Coriolis and centrifugal forces, resulting in secondary flows. The contribution of the impeller to overall aerodynamic loss in the compressor is a direct consequence of the presence of the secondary flows, which carry most of the irreversibility.

Using high fidelity models, performing UQ and robust design optimization is computationally very expensive and practically impossible. Uncertainty propagation using MCS requires an evaluation of a large number of input samples. For high fidelity probabilistic modeling in CFD, every sample would require a definite computation in order to predict the final output variability. In addition, robust design optimization also requires a large number of computational evaluations, thus making the application of high fidelity models infeasible. Nonetheless, this drawback can be sufficiently managed with low fidelity models.

Low Fidelity Modeling

To reduce the computational costs, low fidelity models can be used effectively. These models are not equally detailed as the high fidelity models, but are able to evaluate the outputs fairly accurately without being computationally expensive. Low fidelity models can be fundamentally divided into two types – hierarchical and metamodels. Hierarchical models analyze a problem by reducing the complexity of the modeling procedure, while retaining a certain level of physics. Hierarchical models use analytical methods coupled with empirical correlations in order
to represent the physical reality and properties of a system under consideration. Such models do not entail the complete physical detail compared to the higher fidelity counterparts, nonetheless, they are an efficient simulation alternative. For instance, Navier-Stokes equations used to model fluid flow in CFD can be replaced by simplified Euler equations to reduce the modeling complexity and overall computational effort. Similarly, 1D meanline and 2D throughflow models can replace a 3D CFD model for quicker evaluations. The 1D and 2D model optimization problems do not take into account the blade shape, but consider more generalized parameters such as blade height, mean diameter, effective angle at mean radius, twist, etc [27]. The correlations used within these models have been systematically cleaned up and made easier to operate and more reliable [28]. During design optimization, however, each optimized design has to be re-evaluated with the high fidelity model for additional confirmation and tuning of the low fidelity models.

On the contrary, metamodels (also known as surrogates) are not based on any physics, but on an interpolation of already analyzed results by the higher fidelity models [29]. Metamodeling uses the basic idea of analyzing an initial set of design points to generate data, which can be used to construct approximations of the original high fidelity model [18]. A metamodel can also be developed using data obtained from a hierarchical model, as shown in ref. [17], for reducing computational costs, where an axial compressor throughflow model has been used to create a response surface for robust design optimization. Metamodels perform the same tasks as high fidelity models, but at a very low computational cost. However, preparing a good metamodel can be complicated and time consuming.

For centrifugal compressors in particular, two types of 1D models are in general use – the single-zone model and the two-zone models. Single-zone model considers the entire passage flow between stations as a single average flow process with the calculation of impeller efficiency and slip factor using a set of empirical correlations. The two-zone model, on the other hand, considers the passage flow to be separated into two distinct primary and secondary zones, where the primary zone flow is assumed isentropic, while the secondary zone carries all the irreversibility. The flow properties in the two zones are calculated separately, followed by a mixing calculation to estimate the impeller performance. The diffuser and volute meanline modules are subsequently included to complete the 1D compressor performance modeling. Both single-zone and two-zone models are widely used and their selection is user dependent. However, looking at the two modeling schemes critically, one can easily say that the concept behind a two-zone model is in agreement with the actual flow physics inherent to centrifugal compressor impellers, with the passage flow actually divided in two separate zones. The two-zone model also uses less empiricism compared to its counterpart.

For the turbocharger compressor, a 1D meanline model based on the two-zone methodology has been constructed to conduct basic performance analyses. Once tuned and validated, the model could be used to perform UQ and robust design optimization. A novelty has been introduced in the two-zone scheme by blending in the empirical loss correlations, generally used in a single-zone model, with the
secondary zone calculation scheme. Thus, the new two-zone model contains the features of both of the standard single-zone and two-zone models, while delivering additional information in the form of a loss breakdown together with the overall compressor performance. Chapter 3 contains the comprehensive information on the 1D meanline model prepared for the turbocharger compressor analysis.

During the study, the 1D meanline model was found incapable to carry out the UQ and robust design optimization. The empirical modules required comprehensive tuning or replacement by new ones, specifically derived from the high fidelity model analysis and test data for the particular turbocharger compressor. Some effort has been made to improve the quality of the 1D model, but the required level of accuracy could not be achieved. Alternatively, a metamodel has been constructed as a replacement for the 1D model to carry out the UQ. Two widely used metamodels, quadratic response surface and kriging, have been constructed over a high fidelity data set obtained from a DoE performed in CFD. The models have been evaluated and the quadratic response surface is finally selected as 1D model replacement. Chapter 5 contains detailed information on the metamodels and their use in UQ for the turbocharger compressor.

**Experimental Validation**

Suitable experimental information is generally required for two purposes: firstly to validate the high and low fidelity model results in order to achieve a reasonable assurance of their accuracy, and secondly to experimentally verify the performance of the final optimized designs. Conducting an experimental UQ and the subsequent validation of computational UQ is advantageous, however, it is an expensive procedure. Initially, the experimental data in form of a compressor map has been used to validate both the 1D meanline and CFD performance modeling of the turbocharger compressor. Later, an experimental validation of the computational UQ has also been made possible by carrying out an experimental DoE. A selected number of impellers have been manufactured and tested on a turbocharger test bench to obtain the responses for constructing an experimental metamodel. MCS has been performed using the experimental metamodel for UQ and the results have been compared with the computational counterpart. Chapter 7 illustrates the experimental validation in detail.

**References**


REFERENCES


REFERENCES


3

1D Meanline Performance Evaluation

3.1 Preface

Uncertainty quantification (UQ) through Monte Carlo simulation (MCS) and subsequent design optimization for robustness requires an evaluation of a considerably large number of samples. It would be impractical to perform MCS using a high fidelity model due to limitations in computational resources and time. Therefore, it is imperative that a low fidelity model is employed instead. A one-dimensional (1D) meanline model has been developed to assess the performance variation of the test case compressor subjected to geometric deviations. The present chapter elaborates the 1D modeling procedure in detail and presents a comprehensive performance evaluation of the compressor. The advantages and disadvantages of 1D modeling have been highlighted in the process.

3.2 Introduction to Centrifugal Compressors

Before proceeding to performance modeling of centrifugal compressors, it is important to comprehend its characteristics related to design and flow physics. The different components and the working principle of centrifugal compressors have been introduced in this section with a special focus on the impeller.

3.2.1 Fundamental Design and Operation

Centrifugal compressors are used in situations, where mass flow requirements are low to moderate and the pressure rise is high [1]. As the name suggests, a centrifugal compressor’s task is to compress a certain mass of air from ambient to
higher pressures. Such an operation is desired to be performed as efficiently as possible with a minimum loss in the output performance. Generally, a centrifugal type compressor is made of four basic components as shown in Fig. 3.1. These components are:

- Stationary inlet casing
- Rotating impeller
- Stationary diffuser
- Collector or volute

The inlet casing simply directs the airflow axially into the impeller eye or the inducer. Occasionally inlet guide vanes will be used to pre-swirl the air before the impeller to extend the operating range of the compressor. The impeller blades impart a swirling motion to the air, which leaves the impeller at high velocity. Work transfer takes place in the impeller and the flow is diffused through the impeller and turned towards the radial direction so that it leaves with a combination of radial and tangential velocity components. At the impeller exit, the flow still contains a substantial amount of kinetic energy, which has to be recovered through a diffusion process. A diffuser is employed, which converts the high velocity of the air leaving the impeller into static pressure by slowing it down carefully to an acceptable level. Finally the air is collected in the volute from the circumference of the diffuser and is delivered to single- or multi-exit ducts. A small amount of further diffusion may occur in the volute through a short conical duct at the exit of the volute.
Impeller

The impeller design is a compromise between aerodynamic requirements, mechanical strength considerations and manufacturing capabilities. Modern impeller designs are achieved through a combination of aerodynamic analysis (to optimize the efficiency and range) and structural analysis (for an acceptable service life). Figure 3.2 shows modern casted and fully machined impellers.

Beginning with the inducer, the hub-to-tip ratio is set on structural constraints and is frequently defined in the range of 0.3 to 0.4 for simple overhung impellers and in the range of 0.4 to 0.5 for doubly supported rotors (bearings in front of and behind the impeller) [2]. The inducer incidence angle is an important design variable described as the difference in the blade and incoming flow angles. The impeller blades and the approaching flow must be appropriately aligned to each other in order to prevent compressor surge and choke, depending on the operating condition. In general, subsonic impellers can operate with design point incidence angles in the range of approximately $+3^\circ$ to $+10^\circ$ while most of the low Mach number work has a design point incidence in the range of $+3^\circ$ to $+7^\circ$. Compressors operating in high subsonic regime (Mach numbers from 0.6 to 0.9) generally have incidence angles in range of $0^\circ$ to $+4^\circ$ at design point. For transonic compressors with Mach numbers from 0.9 to 1.3 at inlet, the design point incidence angles lie in the range of $-2^\circ$ to $+3^\circ$ [2].

The blades form the channels through which the moving air is compressed and diffused simultaneously. For high mass flow and compressor efficiency, sharp thin blades are desirable. However, considering the structural requirements, robust thick-at-the-root blades are needed [3]. The achievable thickness of the impeller blades is also dictated by manufacturing considerations. Splitter blades are commonly used to minimize the blockage of the inducer by increasing the throat area. Higher mass flow can be passed through an impeller with the use of splitters extending from half way through the inducer up to the impeller tip. They are normally designed by choosing a location for the splitter, where the area has in-
creased sufficiently to allow for the additional blade blockage \([2]\). As a general rule of thumb, impellers with inlet blade angles in excess of \(55^\circ\) to \(60^\circ\) benefit from the use of splitter since impellers with lesser inlet blade angles do not suffer from severe blade blockage.

Backsweep exerts its dominant effect on stage stability; the reduced stage loading at higher values of backsweep results in a steeper pressure ratio characteristics \([4]\). Such rising pressure ratio characteristic for backswept impellers is inherently stable in a sense that a turbocharger compressor containing it can swiftly respond to the varying engine demands by delivering the right amount of air to the manifold. Backsweep also gives better control of the internal flow and reduces the flow distortion transmitted from the impeller to the diffuser. The flow velocity leaving the impeller is significantly lower, due to which the diffuser downstream of the impeller has less flow deceleration to do and thereby, working more efficiently. The requirements of high efficiency together with a wide range of operation is thus met by the introduction of backsweep. However, backswept impellers also cause a reduction in the specific work input (or pressure head) to the flow, which has to be recovered by operating at relatively higher blade speeds. Consequently, backswept impellers operate at higher stress levels than impellers with no backsweep. Modern impeller designs can have backsweep angles as high as \(40^\circ\) to \(50^\circ\) \([5]\). The additional stress is managed through careful design of blade thickness and the back face profile of the impeller.

**Diffuser**

The kinetic energy of the flow leaving the impeller in a centrifugal compressor is approximately 30\% to 40\% of the total work input. Diffusers are used to convert this remaining kinetic energy into static pressure by one or both of the following two techniques:

1. An increase in flow passage area, which brings a reduction in the average velocity, and hence an increase in static pressure.

2. A change in mean flow path radius, which brings about a recovery in the tangential velocity of the flow according to the conservation of angular momentum principle, i.e., \(r \times C_\theta = \text{constant}\).

Although diffusers can have various configurations, two main classes of diffusers, vaned and vaneless type, are used commonly. Figure 3.3 shows the two diffuser types. Vaneless diffusers are commonly employed in turbocharger compressors based on their design simplicity (since they are made up of two parallel walls), low cost and more importantly, their broad operating range due to absence of a throat, which prevents choking of the diffuser. Moreover, having no vanes also ensures that there is no vane-driven vibratory coupling with the impeller blades, which for vaned diffusers can lead to excitation of the vane leading edges or rotor blades and an eventual fatigue failure. Its disadvantage however, is a lower
pressure recovery compared to a vaned diffuser of a similar diameter. Like other diffusers, vaneless diffuser requires an increase in area in the stream-wise direction to decelerate the flow, which is naturally provided by the increase in the diameter as the flow moves outwards. The frictional forces acting in the boundary layers also retard the flow close to the endwalls. If the diffusion is allowed to proceed sufficiently, the radial motion in the end wall flow can cease altogether causing the flow to move inwards towards the impeller, thereby setting up a recirculation in the diffuser. This will limit the achievable static pressure rise and may ultimately result in a compressor surge. Pinched vaneless diffusers are commonly employed to control such instability by radially regulating the passage height.

Vaned diffusers are used when high pressure ratio and stage efficiency are required. Vaned diffusers achieve a similar or better pressure recovery in a diameter similar to the vaneless diffuser. In addition, a vaned diffuser permits control of the diffuser exit flow angle, which may be important for a good coupling with a downstream volute or return channel, for instance preventing flow separations at the volute tongue. However, vaned diffusers exhibit a much lower flow range due to presence of a distinct throat in the vane passage. Vaned diffusers may use channel (or wedge) and airfoil (or cascade) passages. The channel diffuser is normally a simple straight-sided passage formed by means of discrete, wedge-shaped vanes. Cascade diffuser vanes on the other hand, are usually based on a standard airfoil shape such as NACA 65, but can vary considerably in solidity (chord/pitch ratio). In low solidity diffusers the vane count is small and there is no overlap between adjacent vanes, thus preventing the formation of a throat and eventual choking of the diffuser.

Diffusers operate in a complex flow field with adverse pressure gradients. Pressure recovery is effected by separation or stall, frequently including transitory stall or rotating stall, depending on the particular class of a diffuser. A diffuser designer is, therefore, concerned not only with achieving high levels of pressure recovery, but also with maintaining stable operating conditions.
Discharge Volute

A volute simply transfers the air from the diffuser to the pipe downstream such as the engine inlet manifold in case of turbochargers for automotive application. Despite the simplicity of purpose, volute is a complex component to design.

Fundamentally, it can be divided into three constituent parts; the scroll, tongue or cutwater, and the delivery pipe. The scroll is a spiral-shaped housing that collects the air from the diffuser. The divider separating the delivery pipe from the scroll section is known as the tongue (or cutwater). Volutes can be of the simple overhung or the symmetric type, where the simple overhung type is widely used for turbocharger compressors due to its compactness. An illustration of the two volute configurations is given in Fig. 3.4. The performance of the volute can have a significant impact on the overall compressor performance. For simple stages with an impeller and volute combination, the volute can represent the dominant portion of efficiency loss, particularly at off-design conditions or if the volute is not sized correctly. Depending on the relation between the diffuser exit area and the area at the exit of the volute, the flow in the volute may be diffusing, constant velocity, or accelerating [5]. The scroll cross-sectional areas increase linearly with the azimuthal angle in order to maintain a constant throughflow velocity at design point operation. At low mass flow operations, the flow further diffuses, whereas at high mass flow operations, the flow accelerates to satisfy continuity. Such off-design operations cause a variation of the flow incidence angle at the tongue, which leads to a considerable amount of flow separation and thereby, instability in the compressor. In case of a vaned diffuser, the tip of the tongue would normally be aligned with a vane to reduce the separations caused by the variation in flow incidence around the tongue at off-design operating conditions.
3.3 Impeller Performance Characteristics

Since the impeller is under evaluation for its performance variation due to manufacturing uncertainties, the fundamental phenomenon related to flow physics in such turbomachinery has to be detailed and discussed in order to formulate an understanding of their complex characteristics.

3.3.1 Velocity Triangles and Energy Transfer

The velocity triangles portray the vector relations of all the relevant velocity components, including the relative and absolute velocities and the blade speed, at any point in the turbomachine. In Fig. 3.5, the relative velocities, measured with respect to the rotating system, are denoted by \( W \), whereas the absolute velocities, which are taken with respect to a fixed system, are designated \( C \). The blade speed is represented by \( U \). In all the cases, the velocity triangles can be completed by using the vector relation:

\[
\text{Relative velocity} + \text{Blade speed} = \text{Absolute velocity}
\]

Each velocity can be resolved into components parallel to the principal axes of the machine. The common components are those in the meridional plane (plane containing the machine axis) and the tangential plane. Each component has a particular significance in the functioning of the machine. For instance, the meridional component of velocity \( C_m \) represents the mass flow through the stage, according to the law of conservation of mass, and the tangential component \( C_\theta \) determines the swirl and the work transfer of the stage. The rotating part of the compressor, i.e., the impeller is the only component in the compressor stage where energy transfer occurs and the stagnation enthalpy is increased.

The flow usually approaches the impeller through the inlet duct in the axial direction, unless pre-swirl has been applied to impose a non-zero tangential velocity.
Figure 3.5a shows the velocity triangle at the inducer. The incoming flow having a meridional velocity $C_{m1}$ finds the impeller rotating at blade speed $U_1$. A resultant relative velocity $W_1$ is formed having a flow angle $\beta_1$ inclined to the blade with angle $\beta_{1b}$. A particular incidence angle $i$ is formed between the incoming flow and the blades, which is a function of the operating mass flow rate.

The rotation of the impeller imparts angular momentum to the flow as shown in Fig 3.5b. The rate of change of angular momentum will equal the sum of moments of external forces, i.e., the torque $TQ$. Since angular momentum is a moment of linear momentum, the rate of change will be given by the product of mass flow rate $\dot{m}$, radius $r$ and the tangential component of velocity $C_\theta$. Thus the torque is given as:

$$TQ = \dot{m}(r_2C_{\theta 2} - r_1C_{\theta 1})$$

The energy transfer to the flow is given by the product of torque and angular velocity $\omega$ given by the Euler equation as:

$$\dot{W} = \omega TQ = \dot{m}(U_2C_{\theta 2} - U_1C_{\theta 1})$$

Euler equation forms the basis of energy transfer in rotating turbomachines. It is a combination of the conservation of energy and angular momentum equations affirming that a change in enthalpy is equivalent to a change in tangential velocity of the fluid. Equation (3.2) also shows that the use of positive or negative pre-swirl (when $C_{\theta 1} \neq 0$) will allow decreased or increased levels of work input.

### 3.3.2 Rothalpy

For turbomachines with negligible change in mean radius for instance the axial turbines and compressors, the change in enthalpy is entirely due to change in the tangential velocity. On the contrary, the centrifugal compressor achieves part of its pressure rise from the centrifugal and Coriolis forces due to rotation and the change of radii. This is in addition to the pressure rise achieved through flow turning as in an axial compressor, where energy transfer only takes place through a change of tangential momentum. Rothalpy $I$ can be defined here. It is a special form of the Euler turbomachinery equation and is applicable more easily in certain flow analyses. It is an invariant property obtained when the Euler equation is applied along a streamline. Without going into its derivation, rothalpy is given by the Euler equation represented in form of relative total enthalpy $h_0'$ as,

$$I = h_0' - \frac{1}{2}U^2$$

The rothalpy for any adiabatic flow in a turbomachine is constant regardless of the external work transfers or radius changes [6]. For an axial stage, in which no radius changes occur, the term in $U$ cancels with the relative difference in stagnation enthalpy and rothalpy becoming equal. However, this is not true for machines with significant radius change, and there only rothalpy is constant [7].
3.3. Slip Factor

Ideally, the fluid is expected to follow the impeller blading as it leaves the impeller. However, due to eventual blade unloading at the exit, there is a strong tendency in the flow to migrate from the pressure side towards the suction side. This development is commonly called deviation and for centrifugal compressors, it is traditionally termed as slip since the flow appears to have slipped against the direction of rotation. In Fig. 3.5, the slip has been represented as the difference in tangential velocities given as,

\[ \mu = C_{\theta 2\infty} - C_{\theta 2} \]  

where \( C_{\theta 2\infty} \) is the tangential component of absolute velocity, which would exist if the flow precisely followed the blades i.e., with no deviation or slip. Slip velocity \( C_{\text{slip}} \) is an alternative to the angle of deviation, which is commonly used to express the difference between blade and flow angles at the exducer for centrifugal compressor impellers. Based on the slip velocity, a slip factor can be defined as,

\[ \sigma = 1 - \left( \frac{C_{\text{slip}}}{U_2} \right) \]  

The slip velocity or the slip factor is not calculated; rather it must be specified by a value, or by using an supplementary equation based on some empirical data. The increase in the curvature of impeller blades with backswEEP has a tendency to align the flow in the direction of the channel shape. Impellers with high backswEEP, therefore, have lower slip.

3.3.4 Impeller Flow Physics

The flow through a compressor stage, apart from the inlet duct, is diffusing, i.e., the fluid travels from regions of low pressure to regions of high pressure. This is a difficult environment for the fluid as the boundary layers are prone to separate and the flow is extremely complex, with the separated wakes and main throughflow presenting an unsteady flow downstream of the impeller [6].

In the early days of centrifugal turbomachine development, researchers were aware of a “wake-like” flow region in centrifugal pumps [2]. Early theories of impeller passage flow suggested that the flow separated in the impeller, causing the development of large wake. However, this concept was frequently criticized by many who were off the view that wake flows would be obtained only for poor impellers with low efficiency. A number of high efficiency impellers exhibit regions of depleted boundary layers, often swept into a classical secondary flow. Thus, instead of using the early term “jet-wake”, it is more realistic to think of impellers, whether they have separated regions or not, as showing the inevitable strong secondary flow, hence illustrating the classical two-zone (primary and secondary) character. Correctly considered, the secondary flow comprises of skewed boundary layers, common secondary flows, and tip leakage, possibly with separation as
CHAPTER 3. 1D MEANLINE PERFORMANCE EVALUATION

Figure 3.6: Flow structure in a centrifugal compressor impeller [1]

illustrated in Fig. 3.6. Consequently, the flow field emerging inside a centrifugal compressor passage is very complex, three-dimensional, and turbulent all under the influence of curvature (due to curved and swept blades) and rotation [1].

There are strong viscous effects, significant regions of flow with separation and secondary flow, and also shock waves in case of high-speed transonic compressors. Further complications arise due to tip-clearance, whose influence is more dominant in small centrifugal compressors than in axial compressors or pumps. In view of the large pressure rise and presence of centrifugal and Coriolis force fields, the flow through the centrifugal compressor passages is affected adversely by large boundary layer growth, flow separation, and secondary flow. In view of these features, the centrifugal compressors usually have lower efficiency compared to their axial counterparts. The radial or span-wise transport of mass interacts with the shroud boundary layers and tip leakage flow to produce a very complex flow field. These effects are compounded in a small centrifugal compressor due to small blade height, narrow and curved passages, high rotational speed, and large tip clearance. The secondary flows within the boundary layer determine the position of the wake at outlet and probably influence separation. The wake can be moved by altering mass flow rate, rotational speed or by introducing an inlet distortion to the flow [8].

Considerable amount of work has been done to understand the complex fluid behavior inside centrifugal compressors, both analytically and experimentally. Perhaps the most renowned and established research has been performed by Eckardt in ref. [9,10], where detailed measurements of velocities, directions and fluctuation intensities have been performed with laser velocimeter in the internal flow field of centrifugal impellers, running at tip speeds up to 400m/s. Similarly Krain [11] also experimented to visualize the flow field development in a centrifugal compres-
Figure 3.7: Velocity measurements at various sections of an Eckardt compressor impeller [10]
sor stage consisting of a splitter bladed impeller coupled with vaned and vaneless diffusers.

Figure 3.7 presents a detailed visualization of flow in the impeller passage at different measurement planes in the impeller passage. The first velocity distribution shows a classical pattern of flow with a potential core and thin boundary layers along the wall. A slight inclination in the velocity profile can be seen as a result of some initial aerodynamic loading on the flow (work done on the flow by the blades). The velocities at the second plane are very similar except that the velocity magnitudes across the passage are now stronger. At the third location a new phenomenon appears along the shroud and near the suction side of the blade in shape of a considerable momentum deficit. This indicates the development of a separated flow. The velocity in this region is not zero and thus it cannot be termed as wake. However, it does contain a collection of low momentum fluid, which may be roughly wake-like in character, but also might be described as a strong secondary flow. The effect exists in the next location as well, such that the flow leaving the impeller has a significant momentum deficit at the shroud suction side. Since the meridional velocity component is responsible for convecting the fluid through the compressor stage, a time average of the flow from hub to shroud will reveal a substantial momentum deficit in the shroud region while the flow will be well energized near the hub. Thus the throughflow angle distribution from shroud to hub will be strongly affected and the flow coming out of the impeller will be almost tangential near the shroud and much less tangential (more radial) close to the hub.

Johnson [8] applied the secondary flow theory [12] to represent the passage flow vorticity in an Eckardt impeller analytically. He concluded that for a centrifugal impeller passage, there are three principal contributors to the generation of streamwise vorticity. The first is the inducer bend, where low momentum fluid from the boundary layers is convected from the shroud and hub walls to the suction surface due to centrifugal force. Next is the axial-to-radial bend, where strong centrifugal forces move low momentum fluid toward the shroud from the pressure and suction surfaces and from the unstable hub position. Finally the basic passage rotation increases in strength as the flow passes through the impeller to the radial direction. Here, Coriolis forces generate secondary flows, which move low momentum fluid from the hub and shroud walls and from the unstable pressure surface onto the suction surface. It can be established, therefore, that the main features of the centrifugal flow field are large secondary flows within the entire passage and, in many instances, separated flow.

3.4 Turbomachinery Design and Development

Before proceeding to performance modeling of the test case turbocharger compressor, it is important to understand the design process associated to turbomachinery. The distinction between design and analysis modes is necessary at this stage. In a
design process, a geometric configuration is created to meet certain performance specifications. On the other hand, in the analysis mode, one attempts to predict the performance of a configuration based on its known or proposed dimensions. Thus a good turbomachinery design process essentially utilizes both the design and analysis tools, which consist of the same set of modeling equations, but with a different arrangement.

Turbomachinery design process, as shown in Fig. 3.8, initiates with the mean-line optimization to obtain the basic 1D velocity triangles at each station in a turbomachine. A set of stations is chosen at the inlet or the discharge of each element (inlet guide vane, rotor, diffuser, volute, etc.). At every station, the appropriate thermodynamic and fluid dynamic conservation principles are applied to calculate the fundamental performance parameters, subject to estimated levels of diffusion and loss. After the meanline optimization is completed; usually involving many trial designs plus performance map calculation for off-design performance, the full blading geometry is defined and evaluated. Various geometric techniques are available to lay out the blade shape. After trial blade shapes are available, the next step involves the three-dimensional (3D) flow analysis. Quasi-3D inviscid codes and fully 3D viscous codes are available and are an essential part of the design process. At the same time or sequentially, 3D stress analysis can be conducted as well. At any time in the design process, defects may be found in the preliminary design, which requires the designer to drop back in the process, modify the preliminary design, and then proceed forward again. Following the flow and structural analyses, rotordynamic analysis for the stage is performed to ensure that all the rotating components of the system can operate together with acceptable dynamic
stability. Subsequently, good computer aided drafting is required to convert the
design information into final mechanical drawings for prototyping and eventual
laboratory testing.

3.4.1 Levels of 1D Design and Analysis

A large part of design and optimization work is carried out at the 1D level with
considerable amount of information arriving from high fidelity performance mod-
eling and laboratory testing. A 1D model is, therefore, a very useful tool for its
simplicity and the amount of information it can deliver in quick time. However,
performance modeling in 1D for a centrifugal compressor is complicated and heav-
ily relies upon different empirical models (developed over time using hundreds of
test cases and their test data). The empirical models support the calculation of
complex flow characteristics due to the presence of 3D secondary flow structures
in the passages influenced by rotation and curvature, thus invalidating the models
developed for stationary, non-curved two-dimensional flows.

The process of selection or specification of a design can be divided into three
levels of 1D modeling. The first level of design, level I, is the use of ‘similitude’
to scale an existing design to a new application. Whenever an appropriate stage
is available and can be scaled for a new application, then similitude should be
employed. Here the term “appropriate” includes requirements of flow, head, or
pressure ratio, efficiency and range. In addition, structural life, weight and man-
ufacturability are also essential considerations. If all of these needs can be met
with an existing stage that can be scaled to a larger or smaller size, then the use
of the similitude parameters is ideal. However it is rarely possible to carry out a
direct and complete scaling, at least if scaling down in size due to Reynolds num-
ber effects and hence, a small correction is necessary. It is common to find that
a constant scale factor will not suffice since blade thickness, as well as other as-
pects of the design (such as fillet radii and surface roughness) may become critical
manufacturing limits.

The next level of modeling is achieved by using carefully determined correla-
tions of component performance obtained from prior test experience. For instance,
impeller and diffuser efficiencies can be correlated as separated entities for sub-
sequent design purposes. Level II process is more generally known as single-zone
meanline modeling, which considers the flow in the impeller passage to be a single
flow stream, average flow process, implementing calculations of mass, momentum
and energy, and utilizes the empirical correlations to determine the performance
loss mechanisms and calculation of efficiency. The correlations have to be based
on proper dimensionless parameters thus assuring precise design work. Hence,
the effectiveness of this approach depends on the completeness of the informa-
tion available in form of correlations for various loss mechanisms and components
involved in the stage.

The third level of modeling, or more commonly known as two-zone meanline
modeling, the centrifugal compressor stage is evaluated in more detail. The pro-
cess begins at the inducer, proceeds to the exducer, continues with an effective mixed-out impeller exit station, and then to the stationary components comprising a diffuser and the discharge volute. Modeling in this case does not depend on component correlations, but relies on basic flow modeling instead. These are built on the most basic understanding of the fundamental flow physics intrinsic to centrifugal compressors. In the impeller, existence of a high energy isentropic core flow (primary zone) and a low momentum secondary core flow (secondary zone) is realized and the loss associated with the mixing of these two streams is estimated. The designer is thus attempting to establish a more accurate design based on the available information for basic viscous flow processes under the influence of adverse pressure gradients. The level III analysis is particularly important whenever a new design requires performance levels beyond those previously obtained by any product in a manufacturer’s line, or when an existing stage will be employed under radically new conditions that fall beyond the range of previously correlated performance [2]. Nonetheless, some level of empiricism is still involved in the two-zone modeling process.

3.4.2 Selection of an Appropriate Model

Selection of a suitable modeling method not only depends on the amount of information which is available to construct a model, but also relies on the amount of information, which is required out of the modeling process at the same time. Different industries have tended to focus on different levels based on their priorities. For instance it is possible that an industry will consider to bring forth a slightly different configuration in an unsure market based on existing design configurations that have been established and worked out over a period of time rather than the innovative design, which will inherently require several years of modification and perfection before it is a truly reliable commercial product. Level I and II modeling methods could provide them the platform to perform such a task and utilize these methods to their maximum economic benefit. Turbocharger market relies on a wide database of many tried and tested turbochargers for quick matching with the internal combustion engine requirements set by the customer. Alternatively, other industries pursue a more detailed level III modeling of the fundamental gas dynamic processes in every element of the compressor stage. Each of the elements may be modeled in terms of the basic flow phenomena, which occur in the elements including the primary and secondary flow phenomena, tip-clearance influence, disk friction effects and the mixing processes. Manufacturers pursuing this level of analysis may be involved in more costly designs, but well optimized and versatile designs are obtained, usually mass-produced in families.

A more fundamental review and critical comparison between level II and III modeling methods has been presented in ref. [13, 14]. Both single-zone and two-zone meanline models have been applied to predict performance of the Eckardt impellers. The results are compared against measured experimental data to determine the accuracy of the two modeling techniques. The rotor efficiency and
slip factor are correlated in the single-zone model while an additional commonly
used parameter, the blockage factor, is employed. The use of blockage factor is
fundamentally incorrect since the single-zone model requires either mass average
or an energy average state to support an exact thermodynamic description of the
rotor. One cannot separate off a blocked (boundary layer) region as distinct from
the core flow for this purpose. Comparison of the results obtained from the two
modeling methods against the experimental data revealed the two-zone method
to be more accurate. Consequently, the use of single-zone modeling can only be
accepted for approximate calculations of the detailed thermodynamic and fluid-
dynamic conditions leaving a rotor. However, the fundamental parameters, which
describe the impeller diffusion process and the secondary zone development in the
two-zone method, also depend on some amount of empiricism making it sufficiently
reliant upon a suitable database.

3.5 1D Meanline Model Development

Level II modeling [13] has been selected for the comprehensive performance analy-
asis of the test case turbocharger compressor considering the realism and accuracy
the method could provide. Figure 3.9 provides an illustration of the modeled com-
pressor stage. The analysis begins with the definition of ambient conditions, the
dimensions for different geometric parameters of compressor components and the
operating points. Inducer is analyzed first, which sets the impeller inlet state.
The calculations then proceed to the exducer, where the flow properties are de-
determined using the two-zone methodology. The essential impeller flow processes
are predicted including the primary-secondary flow structure and the resulting
mixed-out state, along with the external losses. This is in fact, a rather complex
computational process involving a system of equations, which are solved itera-
tively. To complete the calculations, it is necessary to specify the diffusion ratio.
For a new design, measured results from a very similar design may be used, or
the expected diffusion characteristics are estimated using correlations of the two-
elements-in-series (TEIS) model.

The vaneless diffuser is analyzed by means of detailed equations given in ref.
[15], where the flow is analyzed considering the wall friction effects and the area
change while neglecting the heat transfer effects. Fluid properties are determined
as a function of radius by solving the conservation equations as a succession of
states through the vaneless diffuser. In the end, final stage properties are calculated
at the volute exit using a simple modeling technique for overhung type volutes
based on geometric area ratio and estimation of losses.

3.5.1 Inducer Analysis

Function of an inducer is to increase the angular momentum of the incoming fluid
without increasing its radius of rotation. Inducer has the largest relative velocity in
the impeller and, if well designed and carefully fabricated, very good performance can be achieved. Present designs have the inducer blading bent in the direction of the relative approaching flow to control the incidence levels compared to older design practices, which had simple straight line elements not necessarily aligned to represent the relative flow at the inlet. Prior knowledge of the inlet ambient conditions, degree of pre-swirl and mass flow rate of the working fluid is necessary to carry out the inducer analysis.

To establish the inducer velocity triangle, the Mach number of the entering flow is determined by an iterative procedure. The following set of equations is solved beginning with an assumed Mach number $M_1$ at inducer inlet station 1:

\[
A_{in} = \pi (r_{1s}^2 - r_{1h}^2) \tag{3.6}
\]

\[
A_f = (1 - B)A_{in} \tag{3.7}
\]

\[
T_1 = \frac{T_{01}}{1 + \frac{\gamma - 1}{2} M_1^2} \tag{3.8}
\]

\[
P_1 = \frac{P_{01}}{\left(\frac{T_{01}}{T_1}\right)^{\frac{\gamma}{\gamma - 1}}} \tag{3.9}
\]
CHAPTER 3. 1D MEANLINE PERFORMANCE EVALUATION

\[ C_1 = \frac{mRT_1}{P_1 A_f} \]  
(3.10)

\[ M_1 = \frac{C_1}{\sqrt{\gamma RT_1}} \]  
(3.11)

Parameter \( B \) in Eq. (3.7) represents the inducer blockage factor due to presence of a boundary layer. Choice of a suitable blockage value depends on the type of inducer and design of the upstream duct. A blockage factor of 0.02 has been used for the test case impeller based on the recommended range of 0.02 to 0.04 provided in ref. [2] corresponding to ideal simple axial inlet types.

With \( M_1 \) calculated, the inlet velocity triangle can be constructed at the inducer tip and similarly, at the mid-span and hub locations using following the set of equations:

\[ C_{m1t} = C_1 A_k \]  
(3.12)

\[ U_{1t} = \frac{2\pi r_{1t} N}{60} \]  
(3.13)

\[ W_{1t} = \sqrt{C_{m1t}^2 + U_{1t}^2} \]  
(3.14)

\[ \beta_{1t} = tan^{-1}\left( \frac{U_{1t}}{C_{m1t}} \right) \]  
(3.15)

\[ i_{1t} = \beta_{ibt} - \beta_{1t} \]  
(3.16)

\[ M'_{1t} = \frac{W_{1t}}{\sqrt{\gamma RT_1}} \]  
(3.17)

\[ T'_{01t} = T_1 \left( 1 + \frac{\gamma - 1}{2} M'_{1t}^2 \right) \]  
(3.18)

\[ P'_{01t} = P_1 \left( \frac{T'_{01t}}{T_1} \right)^\frac{\gamma}{\gamma - 1} \]  
(3.19)

In Eq. (3.12), parameter \( A_k \) symbolizes the axial velocity ratio, which is basically a ratio between the tip and mid-span axial velocities at the inducer inlet. Value for \( A_k \) ranges from 1.02 to 1.06 for ideal simple axial inlets [2]. For the test case impeller, minimum value of 1.02 has been used.

### 3.5.2 Impeller Exducer/Tip-State Modeling

Calculation of the impeller tip-state or exducer flow properties can be divided into four distinct stages — diffusion estimation, primary zone calculation, secondary zone calculation and finally the mixed-out state calculation. The calculation procedure for the above mentioned stages is illustrated in detail in the following sections:


3.5. 1D MEANLINE MODEL DEVELOPMENT

Figure 3.10: Conceptual representation of the TEIS model [2]

Estimation of Impeller Diffusion

Lack of test data related to diffusion characteristics of an impeller necessitates the use of a pragmatic method to estimate the static pressure rise across the stage. The TEIS model provides a simple yet very useful method to estimate the amount of diffusion in an impeller passage. The passage is considered to be a rotating diffuser divided in two sections while recognizing the importance of conventional diffuser parameters such as area ratio and length to width ratio. As shown in Fig. 3.10, the first diffuser section or element \( a \) is considered a variable geometry diffuser depending on the mass flow rate while the second diffuser \( b \) is a fixed geometry diffuser from the inducer throat to the impeller exit. Both elements are considered to be operating with a specific effectiveness. Detailed derivation of the TEIS model is provided in ref. [16] and the governing equations in the model are given below:

Element \( a \):

\[
\eta_a = \frac{C_{pa}}{C_{pa,i}} \tag{3.20}
\]

\[
C_{pa,i} = 1 - \left( \frac{\cos \beta_{1t}}{\cos \beta_{1bt}} \right)^2 \tag{3.21}
\]

Element \( b \):

\[
\eta_b = \frac{C_{pb}}{C_{pb,i}} \tag{3.22}
\]

\[
A_{th} = A_{in} \sin(90^\circ - \beta_{1bt}) - Zb_{le}t_b \tag{3.23}
\]
CHAPTER 3. 1D MEANLINE PERFORMANCE EVALUATION

\[ A_e = 2\pi r_2 b_2 \cos \beta_{2b} - Zb_2 t_b \]  \hspace{1cm} (3.24)

\[ C_{pb,i} = 1 - \left( \frac{A_{th}}{A_e} \right)^2 \]  \hspace{1cm} (3.25)

Overall:
\[ DR^2 = \frac{1}{1 - \eta_a C_{pa,i}} \times \frac{1}{1 - \eta_b C_{pb,i}} \]  \hspace{1cm} (3.26)

\[ DR = \frac{W_{1t}}{W_{2p}} \]  \hspace{1cm} (3.27)

These definitions form the primitive TEIS model using constants, \( \eta_a \) and \( \eta_b \) representing the pressure recovery effectiveness of the two diffusing elements \( a \) and \( b \), respectively. The ideal state is calculated by assuming that there is no blockage or other viscous effects within the passage and the flow follows the blading (zero deviation) while effective blade blockage is considered in the exit flow state. Suitable ranges of values for the two effectiveness parameters have been provided in ref. [2]. Higher values of the coefficients apply to high performance, large scale impellers, whereas lower values apply to the smaller, low performance impellers.

With TEIS model, the two effectiveness parameters, \( \eta_a \) and \( \eta_b \) are used to generate the complete diffusion characteristics for an impeller. There are certain regimes, where the model overpredicts the diffusion characteristics, and thus fails to consider some basic phenomena, which are well known for diffusing or accelerating flow problems. The key defect in prediction comes just prior to the conditions where the diffusion characteristics begin to flatten out either as the result of stall or the development of significant secondary flows or some similar internal process. The primitive model shows a sharp discontinuity and does not have a gradual method to simulate the deterioration of the internal flow state. This defect is traceable to the constant value of the element \( a \) effectiveness \( \eta_a \) and to a lesser extent, element \( b \) effectiveness \( \eta_b \). For instance, each effectiveness should deteriorate as incidence angle increases due to rise in blade loading and streamline curvature in element \( a \) and as a consequence of the increased blockage from element \( a \) feeding into element \( b \). Effectiveness of the element \( a \) should depend on the incidence or turning, inducer tip-clearance, blade thickness, and inlet Mach number, along with the inlet Reynolds number and rotation number of some type. On the other hand, the effectiveness of element \( b \) can be described as a function of blockage, tip-clearance, some parameter representing turbulence and/or vorticity, passage Reynolds number, a rotation or velocity number, general geometric characteristics and passage loading. Equations (3.28) and (3.29) illustrate the dependence of the two effectiveness parameters on various factors related to the flow and the geometric characteristics of the impeller.

\[ \eta_a = f_a [Re_a, M'_{1t}, t_{clr,1}, RN_a, \Delta \theta_{turn}, t_b, ...] \]  \hspace{1cm} (3.28)

\[ \eta_b = f_b [B_a, M_{th}, Re_b, t_{clr,2}, RN_b, \chi \text{ or } \epsilon, \beta_{2b}, ...] \]  \hspace{1cm} (3.29)
This issue will come under further attention in chapter 4 once the impeller geometry will be altered at a constant operating point to carry out the performance sensitivity analysis. The TEIS model in its fundamental state performs reasonably and captures the diffusion characteristics of the centrifugal impeller at varying operating conditions, provided that suitable values for the effectiveness parameters are selected by comparing the model results against measured test data.

**Primary Zone Modeling**

The two-zone model is principally based on the classical jet-wake theory, where the impeller is considered to be containing an unseparated flow in the inducer, followed by a separated and jet-wake flow downstream from the separation point. Coriolis force keeps the two flow streams separated and does not allow any mixing in the impeller passage. However, after leaving the impeller, the jet and wake mix rapidly. The blades are assumed to be unloaded at the tip and the static pressure is considered uniform across the passage at the shroud tip. This assumption is actually not true on the basis of experimental data, but provides a sufficient representative for the sake of modeling [17].

The concept of wake flow was frequently criticized by many researchers who viewed a wake flow to be present only in poorly designed impellers with low efficiency. A number of high efficiency impellers do not show a separated or wake-like region. Instead, these impellers exhibit regions of depleted boundary layers, often swept into a classical secondary flow. Thus, instead of using the jet-wake terminology, it is more realistic to think of impellers, whether they have separated regions or not, as showing strong secondary flow, hence illustrating the classical two-zone character [2]. Fundamental assumptions associated with the two-zone model have been given in ref. [13] and are described below:

1. There is a (nearly) isentropic core zone and a non-isentropic secondary zone.

2. The diffusion of the core flow and development of the secondary zone is similar to other diffuser problems.

3. The tip static pressure may be assumed the same for each zone or corrected by a small factor.

4. Deviation of the secondary zone has a small effect on modeling; deviation of the primary zone is critical in setting overall slip.

5. An effective or equivalent mixed-out state can be computed at the impeller exit for thermodynamic state evaluation and for determining averaged flow properties.

Calculations of the tip-state begin with the calculation of primary zone, which is an iterative procedure initiating with an assumed value of secondary mass flux
fraction $\chi$ (where $\chi = \dot{m}_s/\dot{m}$). Following set of equations can be defined for this purpose:

$$T'_{02p} = T'_{01m} + \frac{\gamma - 1}{2\gamma R} (U_2^2 - U_{1m}^2)$$  \hspace{1cm} (3.30)

$$M'_{2p} = \frac{W_{1t}}{DR} \left[ \gamma R \left( T'_{02p} - \frac{\gamma - 1}{2\gamma R} W_{2p}^2 \right) \right]^{1/2}$$ \hspace{1cm} (3.31)

$$T_{2p} = \frac{T'_{02p}}{1 + [(\gamma - 1)/2]M_{2p}^2}$$ \hspace{1cm} (3.32)

Iterations start with the assumed value for $\chi$ as,

$$\dot{m}_p = \dot{m}(1 - \chi)$$ \hspace{1cm} (3.33)

$$X_p = \frac{\dot{m}_p \sqrt{R T'_{01m}}}{P'_{01m} M'_{2p}} \left( 1 + \frac{\gamma - 1}{2} M_{2p}^2 \right)^{\frac{\gamma + 1}{2(\gamma - 1)}} \left( 1 + \frac{\gamma - 1}{2\gamma R T'_{01m}} (U_2^2 - U_{1m}^2) \right)^{-\frac{\gamma + 1}{2(\gamma - 1)}}$$ \hspace{1cm} (3.34)

Here $X_p$ is defined as the primary flow area parameter (where $X_p = A_{2p} \cos \beta_{2p}$) derived from the non-dimensionalized continuity equation to facilitate the calculation of relative flow angle $\beta_{2p}$. A derivation of the fundamental equation for slip factor Eq. (3.4) combined with Whitfield’s slip factor correlation [18] is used. The calculation proceeds as,

$$\beta_{2p} = \frac{\beta_{2b}}{2} + \frac{1}{2} \sin^{-1} \left\{ \left[ 2 \frac{U_2}{W_{2p}} K \left( \frac{0.63}{2} \right) \frac{X_p}{r_{2b} Z} - \tan \beta_{2b} \right] \cos \beta_{2b} \right\}$$ \hspace{1cm} (3.35)

Use of the slip factor correlation is necessary as it can only be estimated empirically. Whitfield’s slip factor correlation is a modified version of the Stanitz slip factor correlation\(^1\). Advantage is that the correlation is able to accommodate the area of the actual primary flow rather than the complete geometric area of the blade passage. The correlation is given as,

$$\sigma = 1 - K \frac{0.63}{2} \left( \frac{A_{2p}}{r_{2b} Z} \right)$$ \hspace{1cm} (3.36)

Here $K$ represents an empirical parameter used in the correlation to allow for the non-uniform velocity profile in both the hub-to-shroud and blade-to-blade planes of the throughflow primary zone. Depending on the number of impeller blades, selection of an appropriate value for $K$ can be made using the trend provided in ref. [18]. For the test case turbocharger impeller, a value of 1.5 has been

\(^1\)The Stanitz correlation for slip factor of a radial flow compressor is given as $\sigma = 1 - 0.63\pi/Z$, which applies when the flow is assumed to be completely filling the impeller passage.
3.5. 1D MEANLINE MODEL DEVELOPMENT

Figure 3.11: Secondary zone mass flux fraction $\chi$ vs. area fraction $\epsilon$ [2]

used for this parameter. The calculation then proceeds to estimating the primary flow area $A_{2p}$ and finally the secondary mass flux fraction $\chi$ as,

$$A_{2p} = \frac{X_p}{\cos \beta_{2p}}$$

$$A_{geo} = 2\pi r b_2 - Z b_2 t_b / \cos \beta_{2b}$$

$$\epsilon = 1 - \frac{A_{2p}}{A_{geo}}$$

$$\chi = A \epsilon^2 - B \epsilon$$

Iterations terminate with the estimation of secondary flow area fraction $\epsilon$ in the process. Parameter $\chi$ has been normally kept constant by designers for the entire range of operation. This has been termed as a serious modeling defect in ref. [17] since $\chi$ is mainly a function of the compressor operating condition. An estimate of $\chi$ has been given in ref. [2] for different types of impellers based on the diameter and quality of design. The current method iteratively determines the value for $\chi$ using a second-order correlation instead, given by Eq. (3.40), which provides a useful empirical basis correlating the secondary mass flux fraction $\chi$ and area fraction $\epsilon$. This allows the simulation of secondary flow variation at different operating conditions. However, the correlation provided in Eq. (3.40) is dependent on a reasonable estimate of parameters $A$ and $B$.

Figure 3.11 shows two different sets of data presenting the relationship between $\chi$ and $\epsilon$. This data belongs to the three Eckardt impellers and nine comparatively
smaller Concepts ETI (CETI) impellers. A second-order trend can be observed between $\chi$ and $\epsilon$ for both impeller types. Smaller CETI impellers exhibit smaller $\chi$ at a particular $\epsilon$ compared to the much larger Eckardt impellers. Selection of a suitable empirical representation of $\chi$ and $\epsilon$ is clearly dependent on the type of impeller being modeled. This does not mean considering only the impeller size as a criterion, but overall design features have to be taken into account as well. The correlation representing CETI impellers has been considered initially for the test case turbocharger impeller based on similarity in sizes. However, the correlation will be subjected to further calibration for this particular impeller application using a comprehensive assessment with computational fluid dynamics (CFD) and quantification of secondary zone characteristics. Furthermore, it is important to mention here that the variation in $\chi$ is not only dependent on the operating condition, but any deviation in the impeller geometry will also contribute to variation of the secondary zone structure, not modeled here due to severe lack of database.

Finally, the primary zone velocity triangle and thermodynamic properties can be calculated using the following set of equations:

\begin{equation}
C_{m2p} = W_{2p} \cos \beta_{2p}
\end{equation}

\begin{equation}
C_{\theta2p} = U_2 - C_{m2p} \tan \beta_{2p}
\end{equation}

\begin{equation}
C_{2p} = \sqrt{C_{m2p}^2 + C_{\theta2p}^2}
\end{equation}

\begin{equation}
M_{2p} = \frac{C_{2p}}{\sqrt{\gamma RT_{2p}}}
\end{equation}

\begin{equation}
T_{02p} = T_{2p} \left(1 + \frac{\gamma - 1}{2} M_{2p}^2\right)
\end{equation}

\begin{equation}
P'_{02p} = \frac{P'_{01m}}{T'_{02p}^{\frac{\gamma - 1}{\gamma + 1}}}
\end{equation}

\begin{equation}
P_{2p} = \frac{P'_{02p}}{\left[1 + \{(\gamma - 1)/2\} M_{2p}^2\right]^\frac{\gamma}{\gamma - 1}}
\end{equation}

**Secondary Zone Modeling**

The original jet-wake theory considered the wake flow to be leaving the impeller congruent with the blading ($\beta_{2s} = \beta_{2b}$). However, the experimental investigations performed in ref. [19] on a centrifugal impeller showed that the classical jet-wake model needed improvement in its definition of the wake flow deviation. The two-zone methodology, on the other hand, does consider a deviation of the secondary zone, but also realizes its comparatively marginal influence on performance modeling.
3.5. 1D MEANLINE MODEL DEVELOPMENT

Calculation of the secondary zone properties is also an iterative process using the following set of equations:

\[
P_{2s} = P_{2p} \tag{3.48}
\]

\[
T'_{02s} = T'_{01m} + \frac{\gamma - 1}{2\gamma R}(U_{2}^2 - U_{1m}^2) \tag{3.49}
\]

\[
A_{2s} = A_{geo} - A_{2p} \tag{3.50}
\]

Iterations begin with an assumed value for \(\beta_{2s}\) and calculation of the secondary zone area parameter \(X_s\) as,

\[
X_s = A_{2s} \cos \beta_{2s} \tag{3.51}
\]

With the relative stagnation enthalpy established through Eq. (3.49), the only unknowns are the entropy-gain \(\Delta s/R\) and the relative Mach number \(M'_{2s}\). Since the static pressures in the primary and secondary zones are assumed equal, along with the primary zone considered isentropic, the entropy difference between the primary and secondary zones directly estimates the entropy increase in the secondary zone. The entropy gain function \(\sigma\) can be defined as,

\[
\sigma = e^{(-\Delta s/R)} = \left[ \frac{T_{2p}}{T'_{02s}} \left( 1 + \frac{\gamma - 1}{2} M_{2s}^2 \right) \right]^{\frac{\gamma - 1}{2}} \tag{3.52}
\]

The non-dimensionalized continuity equation adapted for non-isentropic processes by the incorporation of \(\sigma\) is given as,

\[
\dot{m}_s \sqrt{\frac{RT_{01m}}{P_{01m}}} = X_s M_{2s}' \left( 1 + \frac{\gamma - 1}{2} M_{2s}^2 \right)^{-\frac{\gamma(\gamma - 1)}{2(\gamma - 1)}} * \sigma \left[ 1 + \frac{\gamma - 1}{2\gamma R T'_{01m}} (U_{2}^2 - U_{1m}^2) \right]^{\frac{\gamma + 1}{2(\gamma - 1)}} \tag{3.53}
\]

Equations (3.52) and (3.53) can be solved simultaneously to calculate the secondary flow Mach number \(M_{2s}'\) using a resolved equation given as,

\[
M_{2s}' = \sqrt{-1 + \sqrt{1 + 2(\gamma - 1)K_s^2}} \tag{3.54}
\]

where \(K_s\) is given by,

\[
K_s = \frac{\dot{m}_s \sqrt{\frac{RT'_{01m}}{\gamma}}}{P_{01m} X_s} \left( \frac{T'_{02s}}{T_{2p}} \right)^{-\frac{\gamma - 1}{2}} * \left[ 1 + \frac{\gamma - 1}{2\gamma R T'_{01m}} (U_{2}^2 - U_{1m}^2) \right]^{\frac{\gamma + 1}{2(\gamma - 1)}} \tag{3.53}
\]

The calculation proceeds to the estimation of secondary flow angle \(\beta_{2s}\) with the iterations terminating once \(\beta_{2s}\) converges as,

\[
T_{2s} = \frac{T'_{02s}}{1 + [(\gamma - 1)/2]M_{2s}'^2} \tag{3.55}
\]
\[ W_{2s} = M'_{2s} \sqrt{\gamma RT_{2s}} \]  
(3.56)

\[ \rho_{2s} = \frac{P_{2s}}{RT_{2s}} \]  
(3.57)

\[ C_{\theta 2s} = f_{us} C_{\theta 2p} \]  
(3.58)

\[ C_{m2s} = \frac{\dot{m}_s}{\rho_{2s} A_{2s}} \]  
(3.59)

\[ \beta_{2s} = \tan^{-1} \left( \frac{U_2 - C_{\theta 2s}}{C_{m2s}} \right) \]  
(3.60)

Parameter \( f_{us} \) in Eq. 3.58 represents the tangential velocity factor used to estimate the deviation in the secondary flow. A value of 0.6 has been set for \( f_{us} \) in the 1D meanline model (section 4.6.3 of chapter 4). Finally, the calculation of secondary zone velocity triangle, along with other thermodynamic properties can be completed using the following set of equations:

\[ C_{2s} = \sqrt{C_{m2s}^2 + C_{\theta 2s}^2} \]  
(3.61)

\[ M_{2s} = \frac{C_{2s}}{\sqrt{\gamma RT_{2s}}} \]  
(3.62)

\[ P'_{02s} = P_{2s} \left( 1 + \frac{\gamma - 1}{2} M_{2s}^2 \right)^{\frac{\gamma}{\gamma - 1}} \]  
(3.63)

\[ P_{02s} = P_{2s} \left( 1 + \frac{\gamma - 1}{2} M_{2s}^2 \right)^{\frac{\gamma}{\gamma - 1}} \]  
(3.64)

\[ T_{02s} = T_{2s} \left( 1 + \frac{\gamma - 1}{2} M_{2s}^2 \right) \]  
(3.65)

The establishment of techniques to link the real irreversible flow and its ideal, theoretically manageable, equivalent is crucial to the success of any turbomachine design or analysis procedure [6]. Generally, these irreversible processes are not conformable to a mathematical treatment in the same depth or with the same confidence as their ideal equivalents. In almost all cases it is necessary to construct an ideal model, which is then modified by the inclusion of some form of loss coefficients. The overall loss is considered as the sum of what are believed to be its likely components, all of which are modeled separately as a function of the relevant dimensional and flow state parameters.

To provide more depth in understanding the loss mechanisms, a sub-iteration has been performed in the standard two-zone model to estimate the entropy production in the impeller passage using a set of loss correlations commonly used in
single-zone meanline modeling. The entropy-gain function is then calculated using these correlations, which estimate the losses based on change in the stagnation enthalpy $h_{02} - h_{02s}$ and converted to $\sigma$ as,

$$\sigma = e^{(-\Delta s/R)} = \left[1 - \frac{\gamma - 1}{\gamma R T_{02s}} \Delta h_{int}\right]^\frac{\gamma}{\gamma - 1}$$

Parameter $\Delta h_{int}$ represents the overall internal loss. The sub-iteration begins with the flow assumed to be isentropic ($\Delta s/R = 0$) and the secondary flow Mach number $M'_{2s}$ is estimated using Eq. (3.54). Velocity triangle is then calculated using $M'_{2s}$ and the assumed $\beta_{2s}$. The calculated velocities are plugged in the mean-line loss correlations, along with other parameters to predict the losses. Iterations continue till $\beta_{2s}$ and $\sigma$ are converged.

### Mixed-Out State Modeling

With velocity triangles and thermodynamic properties established for the primary and secondary zones, the mixed-out state can be calculated. In the original jet-wake model, it was assumed that the jet and wake mixed very rapidly after leaving the impeller. Experimental investigations have shown that the mixing process requires several passage heights past the exducer before it can be considered complete. Nonetheless, an effective mixed-out state has been considered in the two-zone model with the thermodynamic properties established on a mass-average basis. A static pressure rise and stagnation pressure drop are calculated at the impeller discharge due to mixing of the two zones, which in practice is only completed at some unknown point downstream of the impeller.

A control volume has been assumed at the impeller discharge, where dissipation of energy can be accurately calculated through the application of one-dimensional energy, momentum and continuity equations following an iterative
procedure. With the assumed imaginary duct or control volume encompassing the mixing process as illustrated in Fig. 3.12, the shear forces due to the side walls can be neglected and the tangential momentum equation can be used to calculate the mixed-out tangential component of velocity $C_{\theta 2m}$ as,

$$C_{\theta 2m} = (1 - \chi)C_{\theta 2p} + \chi C_{\theta 2s}$$ (3.67)

Iterations begin by assuming the total temperature at mixed-out state $T_{02m}$ and calculating the mixed-out radial component of velocity $C_{m2m}$ using the continuity equation as,

$$C_{m2m} = \frac{b - \sqrt{b^2 - 4ac}}{2a}$$ (3.68)

where

$$a = \frac{\dot{m}}{2\pi r^2 b} \gamma + \frac{1}{2\gamma}$$

$$b = \frac{P_{2p} A_{geo}}{2\pi r^2 b} + \frac{\dot{m}}{2\pi r^2 b} [(1 - \chi)C_{m2p} + \chi C_{m2s}]$$

$$c = \frac{\dot{m}}{2\pi r^2 b} \left( RT_{02m} - \frac{\gamma - 1}{2\gamma} C_{\theta 2m}^2 \right)$$

The radial momentum equation can be rearranged to give the mixed-out static pressure as,

$$P_{2m} = \frac{P_{2p} A_{geo}}{2\pi r^2 b} + \frac{\dot{m}}{2\pi r^2 b} [(1 - \chi)C_{m2p} + \chi C_{m2s} - C_{m2m}]$$ (3.69)

Calculation of the mixed-out velocity triangle can now be completed by determining the remaining properties using the following set of equations:

$$C_{2m} = \sqrt{(C_{m2m}^2 + C_{\theta 2m}^2)}$$ (3.70)

$$T_{2m} = T_{02m} - \frac{\gamma - 1}{2\gamma R} C_{2m}^2$$ (3.71)

$$\rho_{2m} = \frac{P_{2m}}{RT_{2m}}$$ (3.72)

$$T_{02m} = (1 - \chi)T_{02p} + \chi T_{02s} + \frac{\gamma - 1}{\gamma R \dot{m}} (\Delta h_{df} + \Delta h_{rc} + \Delta h_{lk})$$ (3.73)

$$P_{02m} = P_{2m} \left( \frac{T_{02m}}{T_{2m}} \right)^{\gamma-1}$$ (3.74)

The iterations terminate with the convergence of total mixed-out temperature $T_{02m}$ given by Eq. (3.73), which has been derived from the energy equation with the inclusion of external losses (detailed in section 3.5.3) leading to an increase in total enthalpy. Finally, the total mixed-out pressure $P_{02m}$ can be calculated thus completing the impeller performance modeling process.
3.5.3 **Empirical Loss Models**

Inclusion of the internal loss correlations in secondary zone calculation scheme has enabled the two-zone model to be used for assessing the impact of geometric variability on component losses individually and extract additional performance information. In addition to the internal loss mechanisms, which take into account the irreversibilities within the impeller passage, it is also important to consider the amount of work, which is actually required at the shaft. There are losses external to the primary gas path taken by the fluid in the machine resulting in an increase in the impeller discharge total enthalpy without any increase in pressure.

Careful selection of loss models is necessary as many different models have been reported in the literature for each loss mechanism. An optimum set of internal and external loss models is compiled in ref. [20] and has been used for the present 1D model.

**Internal Loss Models**

In general, a rotor or impeller passage loss may be considered to be a sum of a secondary flow loss, a skin friction loss and a tip-clearance loss. The internal loss models used in the present 1D model are described in detail below:

- **Incidence loss.** Only at one operating point will the fluid move smoothly into the passages of bladed components. At other operating points, an angle of incidence will exist between the incoming air and the blade. The operating point, where the incidence loss is minimum does not usually coincide with the zero incidence condition. The objective of an incidence model is to derive both the entropy-gain due to incidence and the incidence angle, for which the minimum incidence loss occurs. Incidence loss model developed by Conrad et al. [21] has been used for the 1D model and is given as,

  \[ \Delta h_{inc} = f_{inc} \frac{W_L^2}{2} \]  

  (3.75)

where parameter \( f_{inc} \) is considered to be between 0.5 and 0.7 while \( W_L \) is the relative velocity component normal to the optimum flow direction given as,

\[ W_L = W_1 \sin(|\beta_1 - \beta_{1,\text{opt}}|) \]  

(3.76)

The main difficulty lies in the specification of the optimum relative flow angle \( \beta_{1,\text{opt}} \), where the incidence loss will be minimum. For the present 1D model, \( \beta_{1,\text{opt}} \) has been determined using a relation given in ref. [6] derived from the continuity equation as,

\[ \tan \beta_{1,\text{opt}} = \frac{A_{in}}{A_f} \tan \beta_{1b} \]  

(3.77)
• **Blade loading loss.** Momentum loss associated to the boundary layer build up on the blade surfaces results in the blade loading or blade diffusion loss. In other words, deceleration of flow during diffusion in the impeller passage results in boundary layer growth, which increases the viscous dissipation and eventual transfer of momentum to internal energy of the flow as heat. Correlation derived by Coppage et al. [22] has been used in the 1D model, where the blade loading loss $\Delta h_{bld}$ is estimated as,

$$\Delta h_{bld} = 0.05D_f^2U_2^2$$  \hspace{1cm} (3.78)

Here, parameter $D_f$ denotes the diffusion factor representing the collective losses and expressed as,

$$D_f = 1 - \frac{W_{2s}}{W_{1t}} + \frac{0.75\Delta h_{Euler}/U_2^2}{(W_{1t}/W_{2s})[(Z/\pi)(1 - r_{1t}/r_2) + 2r_{1t}/r_2]}$$  \hspace{1cm} (3.79)

It is suggested in ref. [6] that 0.6 should be used as the constant in Eq. 3.79 instead of 0.75 for impellers with splitter blades. As the test case impeller consists of splitter blades, along with the full blades, the recommendation has been implemented in the 1D model.

• **Skin friction loss.** Losses due to viscous shear exerted by the surfaces on the fluid, taken to be equivalent to that experienced by a fully developed flow in a pipe of circular cross-section with a diameter and length equal to the average hydraulic diameter $D_{hyd}$ and the average length of the impeller flow passage $L_b$, respectively, come under the skin friction category. Correlation proposed by Jansen [23] has been used to estimate the skin friction loss as,

$$\Delta h_{sf} = 2c_f \frac{L_b}{D_{hyd}} \bar{W}^2$$  \hspace{1cm} (3.80)

where

$$\bar{W} = \frac{C_{1t} + C_{2s} + W_{1t} + 2W_{1h} + 3W_{2s}}{8}$$

$$c_f = \begin{cases} 
2.67 \frac{Re^{0.7}}{Re^{0.8}} & \text{Re} < 3 \times 10^5 \\
0.0622 \frac{Re^{0.8}}{Re^{0.4}} & \text{Re} \geq 3 \times 10^5 
\end{cases}$$  \hspace{1cm} (3.81)

$$Re = \frac{U_2r_2}{\nu_2}$$

$$D_{hyd} = \frac{2\pi \cos \beta_{1bt}(r_{1t}^2 - r_{1h}^2)}{2\pi \cos \beta_{1bt}(r_{1t} + r_{1h}) + 4Z(r_2 - r_{1t})} + \frac{2\pi r_2b_2 \cos \beta_{2b}}{2\pi r_2 \cos \beta_{2b} + Zb_2}$$
• **Clearance loss.** When a fluid particle has a translatory motion relative to a non-inertial rotating coordinate system, it experiences the Coriolis force. A pressure difference exists between the pressure side and suction side an impeller blade caused by the Coriolis acceleration. The shortest and least resistant path for the fluid to flow and neutralize this pressure differential is provided by the clearance gap between the rotating impeller and the stationary casing resulting in a clearance loss. Correlation developed by Jansen [23] has been used for the 1D model to estimate the clearance loss and is given as,

\[
\Delta h_{clr} = 0.6 \frac{t_{clr}}{b_2} C_{\theta 2s} \left\{ \frac{4\pi}{b_2 Z} \left[ \frac{r_2^2 - r_1^2}{(r_2 - r_1)(1 + \rho_{2s}/\rho_1)} \right] C_{\theta 2s} C_{m1m} \right\}^{1/2}
\]  

(3.82)

• **Mixing loss.** There are correlations available for estimating the mixing loss specifically for the single-zone model. However, in case of a two-zone model, the mixing loss can be readily determined by calculating the difference between total pressure at the mixed-out state and mass-averaged total pressure at the impeller trailing edge. The mixing loss is thus given as,

\[
\Delta h_{mix} = c_p T_{02m} \left[ \left( \frac{P_{2m}}{P_{2m}} \right)^{\gamma-1} - \left( \frac{P_{2m}}{P_{2,te}} \right)^{\gamma-1} \right]
\]

(3.83)

All the internal loss mechanisms described above are finally summed up to constitute the overall change in enthalpy \( \Delta h_{int} \) given in Eq. (3.66) to determine the entropy-gain.

**External Loss Models**

External loss models are usually classified as disc friction loss and recirculation loss, but can also include any heat transfer from or to an external source. For the unshrouded impellers, it is also important to recognize the influence of leakage effects. The following external losses have been incorporated in Eq. (3.73) to conclude the mixed-out state calculation:

• **Disk friction loss.** Rise in enthalpy due to work done by shear on fluid between impeller backface and the adjacent surface formed by the cartridge results in a disk friction loss \( \Delta h_{df} \). Disk friction loss normally includes the losses caused by the seals and bearings as well. Estimation of disk friction loss is commonly made by using the correlation developed by Daily and Nece [24] based on an experimental investigation of the power required to rotate discs in an enclosed space and given as,

\[
\Delta h_{df} = c_f \frac{\bar{\rho} U_2^3}{4m}
\]

(3.84)
where
\[ \bar{\rho} = \frac{\rho_1 + \rho_{2m}}{2} \]

The coefficient of friction \( c_f \) is estimated using Eq. (3.81), also used for modeling the skin friction loss.

- **Recirculation loss.** Recirculation happens because of flow reversal or backflow at the blade trailing edges and is a direct function of the discharge flow angle \( \alpha_{2m} \). As the flow angle becomes large, i.e., when it approaches the tangential direction at low mass flow operating conditions, recirculation loss is increased. Correlation developed by Oh et al. [20] has been used in the 1D model to estimate the recirculation loss and is given as,
\[ \Delta h_{rc} = 8 \times 10^{-5} \sinh(3.5\alpha_{2m}^3) D_f^2 U_2^2 \]  
(3.85)
As the equation is suggesting, recirculation loss is also dependent on the amount of diffusion in the impeller passage, which is represented by the impeller diffusion factor \( D_f \) and is determined by Eq. (3.79) provided in the previous section of the chapter. Compressor stages designed with high diffusion ratio are prone to more flow reversal and recirculation and, therefore, can have a higher recirculation loss.

- **Leakage loss.** Generally the loss due to tip-clearance gap is considered as an internal loss. However, Aungier [25] suggested that almost half of the leakage flow could re-enter into the blade passage and re-energized by the rotating impeller causing the leakage loss \( \Delta h_{lk} \). Leakage loss can be calculated using the following set of equations:
\[ \Delta h_{lk} = \frac{\dot{m}_{clr} U_{clr} U_2}{2\dot{m}} \]  
(3.86)
where
\[ U_{clr} = 0.816 \sqrt{(2\Delta P_{clr} / \rho_{2m})} \]
\[ \Delta P_{clr} = \frac{\dot{m}(r_2 C_{\theta 2m} - r_{1t} C_{\theta 1t})}{Z \bar{r} \bar{b} L_\theta} \]
\[ \bar{r} = \frac{r_{1t} + r_2}{2} \]
\[ \bar{b} = \frac{b_1 + b_2}{2} \]
\[ \dot{m}_{clr} = \rho_{2m} Z t_{clr} L_\theta U_{clr} \]
For unshrouded impellers, flow leakage occurs in the impeller and compressor cover or housing clearance gap. Parameter \( U_{clr} \) represents the velocity of the clearance gap leakage flow while parameter \( \Delta P_{clr} \) symbolizes the average pressure difference across the gap resulting in the leakage flow \( \dot{m}_{clr} \).
3.5.4 Vaneless Diffuser Modeling

For a diffuser, the designer is concerned with achieving high pressure recovery efficiently whilst also maintaining a broad operating range. Owing to the simplicity of vaneless diffusers, a one-dimensional analysis is often adopted. If the conservation equations are solved by integrating through the vaneless diffuser, a complete understanding of its performance can be achieved. In 1D modeling, this is done at successive stations through the vaneless diffuser. A set of equations defined by Stanitz in ref. [15] have been used in the 1D model, given in their basic form as,

Radial momentum equation:

$$ C_m \frac{dC_m}{dr} - \frac{C_{\theta}^2}{r} + c_f \frac{C^2 \cos \beta}{b \sin \phi} + \frac{1}{\rho} \frac{dp}{dr} = 0 $$  \hspace{1cm} (3.87)

Tangential momentum equation:

$$ C_m \frac{dC_{\theta}}{dr} + \frac{C_m C_{\theta}}{r} + c_f \frac{C^2 \sin \beta}{b \sin \phi} = 0 $$  \hspace{1cm} (3.88)

Continuity equation:

$$ \frac{1}{\rho} \frac{d\rho}{dr} + \frac{1}{C_m} \frac{dC_m}{dr} + \frac{1}{b} \frac{db}{dr} + \frac{1}{r} = 0 $$  \hspace{1cm} (3.89)

Energy equation:

$$ T_0 = T + \frac{\gamma - 1}{2\gamma R} C^2 $$  \hspace{1cm} (3.90)

Equation of state:

$$ \rho = \frac{p}{RT} $$  \hspace{1cm} (3.91)

Parameter $\phi$ represents the diffuser inclination, such that if $\phi = 90^\circ$, the diffuser is completely radial. It is important to specify an appropriate coefficient of friction $c_f$. Since the flow in the vaneless diffuser is neither fully developed nor simply an inlet flow, the values of skin friction coefficient must be adjusted to obtain reasonable values for a specific application. Values of $c_f$ can be used as an average for the entire diffuser or as a function of local geometry and flow condition [26]. Coefficient of friction $c_f = k(1.8 \times 10^5/Re)^{0.2}$ recommended by Japikse in ref. [2] has been used with parameter $k$ equal to 0.01 based on a review of various vaneless diffusers in industrial machines.

The predicted results from the one-dimensional analysis are reasonably accurate provided that the inlet flow to the vaneless diffuser is uniform. The mixing effects due to non-uniform flow conditions at the impeller exit are, therefore, not considered. Moreover, the influence of heat transfer has been neglected as well. Nonetheless, an advance model is available in ref. [26], where the diffuser inlet flow distortions due to two-zone flow have been considered.
3.5.5 Discharge Volute Modeling

A reasonable modeling method, originally developed by L.R. Young for performance prediction in simple overhung volutes, is given in ref. [2]. The technique is principally based on geometric area ratios with the following set of equations:

\[ AR = \frac{A_7}{A_5} = \left( \frac{\pi}{4} \right) \frac{D_7^2}{2\pi r_5 b_5} \]  
(3.92)

\[ \lambda = \frac{C_{\theta 5}}{C_{m5}} \]  
(3.93)

The basic loss mechanisms in the volute have to be established in order to estimate the pressure recovery. The losses can be estimated by a simple set of equations. First, it is assumed that the meridional component of kinetic energy entering the scroll is totally lost, such that the loss component can be written as,

\[ K_m = \frac{\Delta p_{0, \text{meridional component}}}{\frac{1}{2} \rho C_{\theta 5}^2} = \frac{1}{2} \rho \frac{C_{m5}^2}{C_{\theta 5}^2} \]

\[ \Rightarrow K_m = \frac{1}{1 + \lambda^2 F_1} \]  
(3.94)

Parameter \( F_1 \) depicts a correction factor, which allows some part of the meridional velocity to be used effectively in a downstream component for stabilizing the flow, such as in a conical diffuser at the volute exit. The parameter is normally used for high flow, low pressure ratio gas line compressors with values ranging from 0.6 to 1.0. Therefore, a value of 1.0 has been used for the test case turbocharger compressor scroll.

The tangential component of velocity can be modeled with two assumptions. If the core flow in the volute accelerates (\( C_{\theta 5} < C_7 \)), such as at high mass flow rate compressor operations, then it is valid to assume that no loss occurs. However, if the core flow decelerates in the volute (\( C_{\theta 5} > C_7 \)), such as at low mass flow rate compressor operations, then the volute flow is diffused and it is assumed that the pressure loss is equivalent to total pressure loss in a sudden area expansion. This can be modeled as follows,

\[ K_\theta = \frac{\Delta p_{0, \text{sudden expansion}}}{\frac{1}{2} \rho C_{\theta 5}^2} \]

\[ = \frac{1}{2} \rho C_{\theta 5}^2 \left( 1 - \frac{A_5}{A_7} \right)^2 \]

\[ \Rightarrow K_\theta = \frac{\left( \lambda - 1/AR \right)^2}{1 + \lambda^2} \]  
(3.95)
3.6. COMPRESSOR PERFORMANCE EVALUATION IN 1D

If the centre of radius of the volute $r_6$ is larger than the diffuser exit radius $r_5$, the volute may permit some degree of free vortex diffusion over $\Delta r = r_6 - r_5$. In this case, $C_{\theta 5}$ would be modified in Eq. (3.95) to give,

$$K_\theta = F_2 \left( \frac{r_5}{r_6} \right)^2 \frac{(\lambda - 1/AR)^2}{1 + \lambda^2}$$

(3.96)

Here, value for the parameter $F_2$ ranges from 0.5 to 1.0 and is used for volutes with $r_6 >> r_5$. Since the difference between $r_6$ and $r_5$ is small for the turbocharger compressor scroll, a value of 0.8 has been considered for $F_2$. The two loss mechanisms associated to the volute can now be combined to determine the overall pressure loss. Prediction of the pressure recovery coefficient on the other hand, depends on whether the flow condition corresponds to $\lambda AR > 1$ (low mass flow rate, i.e., diffusing) or $\lambda AR < 1$ (high mass flow rate, i.e., accelerating). Hence, the pressure recovery coefficient is estimated as,

$$C_p = \frac{2(\lambda - 1/AR)}{AR(1 + \lambda^2)} \quad \text{for } \lambda AR > 1 \quad \text{Flow diffuses}$$

(3.97)

$$C_p = \frac{\lambda^2 - 1/AR}{1 + \lambda^2} \quad \text{for } \lambda AR < 1 \quad \text{Flow accelerates}$$

(3.98)

Estimation of volute performance completes the 1D compressor modeling process. The 1D model is now ready for detailed performance calculations.

3.6 Compressor Performance Evaluation in 1D

The test case turbocharger compressor has been introduced in chapter 2. This section describes its performance evaluation using the 1D meanline model.

3.6.1 Compressor Geometry and Specifications

The test case turbocharger compressor comprises of an impeller wheel, vaneless diffuser and an overhung type volute as shown in Fig. 3.13, along with some of the geometric specifications provided in Table 3.1. The impeller is designed with backswept full and splitter blades, which are six in number each. The backsweep angle continually increases from hub to shroud by approximately 6° with an average backsweep angle of 35° at the trailing edge resulting in a twisted or leaned blade form. The main consequence of having a lean is a redistribution of the flow in the span-wise direction, which influences the impeller outlet velocity distribution and secondary flow structure [27].

Clearance gap between the blade tips and the compressor cover generally reduces during compressor operation mainly due to radial and axial displacement of the impeller, caused by centrifugal effects and shaft movements. For the test case compressor, an indication of operating tip-clearance was unavailable. True value
of the operating tip-clearance is difficult to determine and require comprehensive structural and rotordynamic analysis, along with appropriate experimentation. Hence, CFD has been used instead to estimate a suitable value of the operating tip-clearance for the test case compressor. The inducer and exducer tip-clearances have been assumed similar. The clearance gap has been varied gradually from no tip-clearance to the steady-state tip-clearance value in CFD. A comparison has been made against the available experimental data in shape of a compressor performance map. The analysis revealed an operating tip-clearance of 0.15mm (approximately 6% of tip height $b_2$) to be an appropriate estimate, giving a close agreement with the test compressor map. Nonetheless, this estimate is still doubt-

Table 3.1: Specifications of the turbocharger Compressor

<table>
<thead>
<tr>
<th>Component</th>
<th>Parameter</th>
<th>Dimension</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller</td>
<td>Inlet shroud radius, $r_{1s}$ [mm]</td>
<td>13.1</td>
</tr>
<tr>
<td></td>
<td>Tip radius, $r_2$ [mm]</td>
<td>18.5</td>
</tr>
<tr>
<td></td>
<td>Tip height, $b_2$ [mm]</td>
<td>2.6</td>
</tr>
<tr>
<td></td>
<td>Blade thickness, $t_b$ [mm]</td>
<td>0.5</td>
</tr>
<tr>
<td></td>
<td>Number of blades, $Z$ [-]</td>
<td>12</td>
</tr>
<tr>
<td></td>
<td>Backsweep angle, $\beta_{2b}$ [deg]</td>
<td>35</td>
</tr>
<tr>
<td>Vaneless diffuser</td>
<td>Outlet radius, $r_5$ [mm]</td>
<td>33</td>
</tr>
<tr>
<td></td>
<td>Outlet height, $b_5$ [mm]</td>
<td>1.7</td>
</tr>
<tr>
<td>Volute</td>
<td>Outlet diameter, $D_7$ [mm]</td>
<td>27.5</td>
</tr>
</tbody>
</table>
3.6. COMPRESSOR PERFORMANCE EVALUATION IN 1D

The turbocharger compressor uses a vaneless type diffuser with a front pinch. The pinch results in a more uniform flow field at the impeller and diffuser exit stations with an improvement in the impeller efficiency. The diffuser flow is finally collected in the overhung volute, thereby completing the compressor stage.

3.6.2 Stage Performance Mapping and Validation

The 1D model has originally been built to assess the impact of manufacturing uncertainties on centrifugal compressor performance so as to include the manufacturing aspects in the design as well. An immediate test of the 1D model reliability can be made by simple 1D performance mapping and comparison with the test data available from the turbocharger manufacturer. Figure 3.14 shows the compressor performance map comprising of stage pressure ratio and isentropic efficiency plots obtained from 1D analysis and test data for 220,000rpm rotational speed. The results show a reasonable agreement for mass flow rates close to best-efficiency operating point. The model overpredicts the compressor performance at near-stall and choke operating points, showing the limitations of the 1D modeling process. Further validation at other turbocharger speeds could not be made mainly due to lack of reliable test data and to some extent, the geometric information as well.

While all the empirical variables affect the 1D model predictions, the diffusion ratio has the most significant influence on stage performance calculation. A sensitivity analysis for diffusion ratio estimation by the TEIS module revealed that the selection of effectiveness parameters, (3.28) and (3.29), should be made carefully and through a validation against test data. Element a effectiveness parameter $\eta_a$
has a considerable impact on the slope of impeller pressure ratio and isentropic efficiency speed lines while element b effectiveness parameter $\eta_b$ influences their overall location on the ordinate. The values suggested for the effectiveness parameters $\eta_a$ and $\eta_b$ in ref. [2] range from 0.4 to 0.6 and -0.3 to +0.3, respectively, for small or poorly designed impeller cases. However, these are preliminary suggestions and users can establish better estimates based on their own experience and database. Values of 0.6 and 0.3 have been found suitable as $\eta_a$ ad $\eta_b$, respectively, for the test case impeller confirmed by the agreement observed in Fig. 3.14. Other modeling assumptions, such as the unloaded tip condition and representation of the impeller passage flow as two distinct fluid zones, have a marginal influence on 1D model accuracy. As mentioned in section 3.5.2, the correlation representing the CETI impellers in Fig. 3.11 has been considered initially for Eq. (3.40) to model the secondary flow characteristics of the test case turbocharger compressor. Although the CETI correlation performs sufficiently, a further improvement in the correlation has been made using the database developed via comprehensive CFD analysis of the turbocharger compressor. As a result, a new correlation has been devised and used in the 1D model in place of the CETI correlation. The correlation is given as,

$$\chi = 0.25\epsilon^2 + 0.15\epsilon$$  \hspace{1cm} (3.99)

The 1D performance data presented in Fig. 3.14, and in later sections of the chapter, has been modeled using the new correlation in Eq. 4.2. Information related to CFD modeling and quantification of the two-zone flow has been detailed in chapter 4.

### 3.6.3 Study of the Compressor Loss Mechanisms

Various loss mechanisms contributing to entropy production in the impeller have been described in fair detail in the chapter. These include the internal and external loss mechanisms. To study the compressor losses in detail, the entropy-gain plots have been obtained in 1D for three different compressor speed lines. A comprehensive evaluation of the results has been presented in this section to develop an understanding about the loss mechanisms.

#### Internal Losses

Figure 3.15 compiles the various internal losses in the impeller passage. These comprise of the incidence, blade loading, skin friction, clearance and the mixing losses expressed in terms of entropy-gain $\Delta s/R$. It is important to mention here that the standard two-zone model does not compute the internal losses individually unless a provision is made separately. Instead, the TEIS model and the $\chi - \epsilon$ correlation predict the internal losses collectively. However, with the addition of loss correlations in the secondary zone calculation scheme, the behavior of internal loss mechanisms at various operating conditions and with geometric deviations can be studied as well.
3.6. COMPRESSOR PERFORMANCE EVALUATION IN 1D

Figure 3.15: 1D internal loss plots at different rotational speeds

The plot for incidence loss given in Fig. 3.15 represents an average of incidence losses at inducer hub, mid-span and tip locations. For each speed line, a point of minimum incidence loss can be observed from the plot indicating an optimum operating condition, where inducer blading and the relative flow make an optimum incidence angle. Overall, the incidence losses do not have a significant effect on the compressor performance apart from low mass flow rate operations, where a steep rise in entropy-gain is obtained. The blade loading loss is a consequence of negative velocity gradients in the boundary layer and the associated separation of flow in the impeller [29]. Plot shows a reduction in the blade loading loss with increase in mass flow rates for the three speed lines as a result of simultaneous decrease in diffusion ratio and work transfer to the passage flow. Overall, the blade loading loss has a considerable effect on impeller performance. Plot for skin
friction loss shows a sharp increase in the entropy-gain with increase in mass flow rates and rotational speeds owing to higher flow velocities in the impeller passage and more wall-attached flow due to reduction in the secondary zone. Overall, the skin friction loss is also one of the main contributors to the entropy gain in the impeller. From the plot, it is interesting to see that the clearance loss achieves a maximum near the best-efficiency operating point on the three speed lines. With further increase in the mass flow rate, the clearance loss decreases. This is contrary to what is presented in ref. [30] where the clearance loss increases with increase in the mass flow rate. A major factor for such a difference can be traced to the blade backsweep angle and the resulting difference in the meridional and tangential velocities (or the swirl) at the impeller exit. The turbocharger impeller in ref. [30] is fully radial, thereby having $0^\circ$ backsweep angle compared to the $35^\circ$ backsweep of the test case turbocharger impeller. For a fully radial impeller, only the meridional velocity varies at different mass flow rates for a particular rotational speed, while the tangential velocity remains constant at the impeller exit. For a backswept impeller under similar operating conditions, the tangential velocity at the impeller exit also shows a large variation along with the meridional velocity. As a consequence, the pressure head characteristics are much steeper for the backswept impeller compared to the fully radial impeller [2]. The increase in the clearance loss in ref. [30] can, therefore, be attributed to the increase in the meridional velocity from stall to choke (and the subsequent increase in the clearance gap flow) at a constant tangential velocity. On the other hand, for a backswept impeller, the sharp decrease in the tangential velocity (and subsequent decrease in the clearance gap flow due to lesser blade loading) with increase in the mass flow rate ultimately results in a decrease in the clearance loss. Compared to other loss mechanisms, clearance loss has a moderate contribution to the overall
3.6. COMPRESSOR PERFORMANCE EVALUATION IN 1D

![Graphs showing diffuser and scroll losses with mass flow rate and entropy gain]

Figure 3.17: 1D stationary component loss plots at different rotational speeds

entropy-gain in the impeller secondary zone. Mixing loss is produced due to mixing of blade wakes with the free stream flow [31]. Two-zone modeling considers a rapid mixing between the low-momentum secondary and the isentropic primary flows, which produces a certain loss in the passage. Proper mixing calculations are incorporated at an effective mixed-out state in the 1D model, where the mass-averaged flow properties have been determined. From the plot, it is evident that the mixing loss reduces considerably as the operating point moves from stall to choke. Such reduction is expected with a reduction in the secondary zone as compressor operates at high mass flow rates. Overall, mixing loss is a major contributor to entropy-gain in the impeller.

External Losses

The external losses due to leakage, disk friction and recirculation are associated to the flow in the clearance gaps [31]. Figure 3.16 compiles the external loss mechanisms for the test case turbocharger compressor.

At low mass flow rates, impeller work is increased due to recirculation of flow back in the impeller passage giving rise to the recirculation loss. The 1D analysis predicts a large recirculation loss at lower mass flow rates for the three speed lines. With increase in the mass flow rate, the recirculation loss reduces sharply until it becomes minimal. At low mass flow rate operating conditions, the flow also leaks into the gap between the impeller disk and the cartridge or bearing housing. Disk friction loss is thus produced due to friction caused in the leakage flow between the impeller disk and the housing. The plot for disk friction loss shows a decrease in the entropy-gain as the mass flow rate increases. The flow in the clearance gap between the impeller disk and the housing would substantially reduce thus resulting in a reduction in disk friction loss. Some amount of leakage
Figure 3.18: Compressor loss grading at the best-efficiency operating point

flow re-enters into the blade passage from the clearance gap between inducer tip and shroud if no seal has been installed. This leakage flow is re-energized by the impeller and the power consumed to re-energize the flow represents a leakage loss. As leakage reduces with an increase in mass flow rate, the leakage loss is also reduced considerably. Overall, leakage loss has a significant contribution to the external loss production for the particular test case turbocharger compressor.

**Stationary Component Losses**

A diffuser is used for pressure recovery while the scroll guides the flow out of the compressor to the components downstream. In addition to the impeller, the stationary components also have contribution to entropy production in the compressor stage. Figure 3.17 contains the plots for entropy-gain in the vaneless diffuser and the scroll of the test case turbocharger compressor. The flow leaving the impeller has a high velocity; in fact this is the highest relative velocity anywhere in the machine and entropy generation on the walls (due to viscous friction) of the vaneless space will be extremely high [32]. The vaneless diffuser loss falls with increasing mass flow rate due to the increasing flow angle at the entry to the diffuser, thereby resulting in shortening of the flow path and a consequent reduction in the predicted friction loss [30]. On the other hand, the scroll exhibits a sharp increase in the entropy gain and, therefore, an increase in the scroll loss as the mass flow rate is increased from stall to choke.
3.7. ADDITIONAL 1D MEANLINE MODELING

Compressor Loss Grading

Figure 3.18 illustrates the contribution of various loss mechanisms to the deficit in isentropic efficiency at the compressor best-efficiency operating point (mass flow rate of 60g/s at 220,000rpm rotational speed). It is highly likely that the loss grading is different for other operating points such as stall and choke. Nonetheless, since the focus is on best-efficiency performance mainly, other operating points have been ignored. At this operating point, the 1D model calculates the stage pressure ratio and isentropic efficiency of 2.38 and 73.6%, respectively. Results indicate that the impeller is a major contributor to drop in isentropic efficiency compared to the stationary components. Approximately 17.5% points of the overall 26.4% points isentropic efficiency drop is associated to the internal and external impeller losses. The vaneless diffuser and scroll contribute only 3.8% and 5.3% points to the isentropic efficiency drop, respectively. Therefore, it can be concluded that the impeller is the prime loss source in a centrifugal compressor with blade loading, skin friction and mixing loss mechanisms contributing significantly to the overall deterioration in compressor isentropic efficiency.

3.7 Additional 1D Meanline Modeling

The 1D meanline model presented in the thesis is an extension of a standard two-zone modeling method for predicting centrifugal compressor performance. The model combines the empirical loss correlations used in the single-zone modeling method, thereby providing additional information related to compressor loss mechanisms. An additional study has been carried out by a Master of Science student, Sin-Yun Yang from Delft University of Technology (TU Delft). In the study, the standard single-zone and two-zone meanline models, along with the extended two-zone model, have been applied on the test case turbocharger compressor, and two well documented Eckardt impellers. The detailed analysis gave a further insight to 1D meanline modeling techniques used for centrifugal compressors and revealed the advantages and disadvantages of the extended 1D model proposed in the thesis. More information on the study can be found in ref. [33,34].

3.8 Conclusions and Recommendations

The following conclusions can be drawn based on the 1D performance analysis of the test case turbocharger compressor:

- 1D meanline modeling of centrifugal compressors is a difficult task due to the presence of a complex 3D flow field. A considerable amount of empiricism is used in shape of data driven empirical models to estimate the compressor performance.
• Centrifugal compressors have an inherent secondary flow zone in the impeller passage regardless of how efficient the impeller is. The secondary flow mainly comprises of the low-momentum boundary layers and the leakage flow from the clearance gap between the impeller and the compressor cover.

• 1D performance modeling based on the two-zone methodology is logical considering the fact that there are indeed two distinct zones of flow in an impeller passage, which later undergo rapid mixing downstream of the impeller. Compared to the single-zone modeling scheme, the standard two-zone model not only represents the actual impeller flow physics, but also uses less empiricism. Nonetheless, to understand the compressor loss mechanisms in detail, a provision has been made in the standard two-zone model by decomposing the entropy production in the secondary zone into its sources using single-zone empirical loss models.

• Compressor performance is majorly influenced by the diffusion characteristics of the impeller. The TEIS module for estimating this diffusion in the impeller passage predicts the compressor performance reasonably for different operating conditions and rotational speeds. However it would be interesting to see how the module behaves once the geometric deviations are incorporated at a particular operating condition. The use of constant effectiveness parameters in the TEIS model is not applicable once the impeller geometry is varied.

• Compressor loss grading suggests that the impeller is contributing the most to loss in compressor isentropic efficiency. Improvement in the impeller design for higher performance, along with achieving robustness against manufacturing uncertainties will, therefore, be beneficial for the test case turbocharger. The 1D model can be helpful in achieving these goals, but only after further calibration and validation using high fidelity data.

References


REFERENCES


REFERENCES


Performance Evaluation in 3D CFD

4.1 Preface

The application of three-dimensional (3D) Computational fluid dynamics (CFD) modeling is important to gain an insight into the test case centrifugal compressor flow characteristics at a higher level than one-dimensional (1D) meanline modeling. Consequently, the acquired knowledge related to flow characteristics in the impeller passage at different operating conditions will be advantageous for improving the 1D meanline model. The compressor has been simulated in CFD following a set of procedures and a comprehensive evaluation of the flow structure and behavior has been made. The results have been used to calibrate the 1D meanline model in order to improve its accuracy of prediction. The present chapter describes the methodology and outcomes of the analysis in detail.

4.2 Introduction to CFD

CFD is a comprehensive, but compared to a 1D meanline model, a computationally expensive procedure for aerodynamic design and analysis of turbomachinery. However, present day design process is incomplete without it. CFD numerically solves the Navier-Stokes equations (comprising of governing equations for continuity, momentum and energy) for systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulations.

Most turbomachinery flows are turbulent or transitional between laminar and turbulent flows. Turbulence is characterized by irregular or random fluctuations in the flow, originating as instability as it undergoes laminar to turbulent transition.
Turbulent flows are three-dimensional with velocity fluctuations existing in all directions. In addition, turbulence exhibits diffusive and dissipative properties. Diffusion gives rise to rapid mixing and increased rates of momentum, heat and mass transfer, whereas dissipation increases the internal energy of the fluid through viscous shearing and vortex stretching processes, resulting in velocity fluctuations at the expense of the kinetic energy and entropy production in the flow.

Principally, the Navier-Stokes equations are applicable to both laminar and turbulent flows. In practice, however, the computational effort required to track even the largest turbulent eddies (rotating vortex structures in a turbulent flow) is far beyond present capabilities. Instead, the Reynolds-averaged Navier-Stokes (RANS) equations are commonly used. In Reynolds-averaged approaches to turbulence, all of the unsteadiness is averaged out, i.e., all unsteadiness is regarded as part of the turbulence [1]. Additional turbulent stress terms, known as the Reynolds stresses, are consequently introduced in the governing equations, which are approximated by the turbulence models.

On the whole, turbulence influences the aerodynamic and thermodynamic characteristics of turbomachines. Therefore, its inclusion in turbomachinery analysis is compulsory. The upcoming sections of the chapter elaborate on the CFD modeling of the test case centrifugal compressor, having complex and largely three-dimensional (3D) passage flow with large turbulent flow regions in the passage.

### 4.3 CFD Preprocessing and Solution setup

CFD modeling requires a number of steps to prepare a case for comprehensive 3D simulation and analysis. This section presents the preprocessing of the test case centrifugal compressor including its geometric parameterization, grid processing and solver setup.

#### 4.3.1 Geometry Definition

The first step in preprocessing is to obtain high-quality geometric models of the components undergoing analysis. For the turbocharger compressor, the available geometric data in shape of part drawings and physical measurements have been transformed into a 3D geometric model using the commercial software ANSYS BladeGen. The tool allows rapid definition of blade rows and provides the essential parameterization for advance design and optimization problems. The inlet duct, impeller and the vaneless diffuser have been modeled as a single fluid domain with interfaces defined between the stationary and rotating components. A main advantage of this procedure is the conformity of the interface mesh, i.e., the discretization of the fluid domain continues from one component to the other. Consequently, the computational effort required to solve the fluid properties on conformal meshes can be considerably reduced.
Figure 4.1 shows the single passage fluid domain. The flow enters the inlet duct and leaves from the diffuser. The inlet duct has been extended upstream of the impeller to accommodate any flow reversals from the impeller during low mass flow rate simulations. The impeller has been modeled as five equally spaced meridional sections lofted in the stream-wise direction by defining successive coordinates obtained from the part drawings. Similarly, the blades have been parameterized by defining successive stream-wise wrap angles along the hub and the shroud. The wrap angle can be defined as the angular variation of the blade curvature from leading to trailing edges, when projected on a two-dimensional (2D) plane. At the trailing edge, the hub has a larger wrap angle compared to the shroud which causes the blades to lean by approximately $60^\circ$ in the direction of rotation, when measured span-wise from hub to shroud. Blade lean effectively redistributes the flow velocities at the impeller exit by increasing the overall static pressure gradient in the hub region [2]. This prevents the separation and recirculation of flow at the impeller exit following an increase in the flow velocity at the shroud induced by conservation of mass. Furthermore, difference in wrap angles at hub and shroud sections also reduces the backsweep angle by approximately $6^\circ$ span-wise from hub to shroud with the mid-span backsweep angle approximately equal to $35^\circ$.

The blade thickness distribution has been kept constant in both stream-wise and span-wise directions. In reality, the blades are tapered and fillets are added to strengthen against blade root stresses. For the CFD analysis, however, the blade thickness has been assumed constant using the value at mid-span. The volute has been not been modeled in CFD mainly due to unavailability of a suitable geometric
model. Nonetheless, excluding the volute from the fluid domain also reduces the computational expense.

### 4.3.2 Grid Processing

The governing equations are discretized using an appropriate computational mesh including most of the geometrical features of the centrifugal compressor. The fluid domain shown in Fig. 4.1 is exported to ANSYS TurboGrid, which is a quick and easy to use grid processor for turbomachinery applications. A 3D structured grid has been created using the H-Grid and O-Grid topologies as illustrated in the Fig. 4.2. The O-Grid topology provides a good mesh around the blades, while rest of the passage consists of the H-Grid. The mesh at the inlet-impeller and impeller-diffuser interfaces is conformal, i.e., the grid is shared between the two connecting surfaces at the interface, which is an advantage of having the complete domain constructed in BladeGen. The grid downstream of the impeller blades into the diffuser is also well resolved thus facilitating a good modeling of blade wakes.

To resolve the viscous sublayer in the turbulent boundary layer, it is essential to maintain the non-dimensional wall distance $y^+$ close to 1 by having a fine near-wall mesh. For the compressor fluid domain, the near-wall mesh has been made fine by providing sufficient number of nodes in the boundary layer region. As a result, an overall $y^+$ approximately equal to 1 has been achieved at the wall boundaries except the trailing edge of the blades, where a fine near-wall mesh could not be constructed.

A numerical method is said to be convergent in terms of mesh resolution if the solution of the discretized equations tends to the exact solution of the dif-
ferential equation as the grid spacing tends to zero [1]. As the grid provides the specific locations for discretization of governing equations, the solution converges to a grid-independent condition, where any further grid refinement will not have a significant influence on the solution. For the test case centrifugal compressor, a grid independence study has been performed accordingly using three different grid sizes of 0.25 million, 0.475 million and 0.70 million hexahedral elements. Impeller and diffuser outlet properties including pressure ratio and efficiency have been compared for the three grid sizes. Maximum difference in pressure ratio and isentropic efficiency has been found to be approximately 0.18% and 0.15% respectively for the three grid sizes. Based on this marginal variation in properties with increasing grid size, a grid of 0.4 to 0.5 million elements is selected to be sufficient for further modeling.

The tip-clearance gap of 0.15mm (approximately 6% of tip height $b_2$) is introduced in TurboGrid by trimming the blade shroud profile. The number of hexahedral elements in the clearance gap has been set to 15 to capture the clearance gap flow and associated properties.

### 4.3.3 Boundary Conditions

Figure 4.1 also illustrates the different boundary conditions applied to the compressor fluid domain. Since the compressor is rotationally symmetric, the fluid domain comprising of the inlet duct, impeller and the diffuser can be subdivided and simulated as a single passage domain. Periodic boundary conditions are defined at the symmetric surfaces. The results obtained from the single passage simulations will be periodically repetitive (specially near the best-efficiency or design point operation) and significant computational time can be saved compared to simulating the full compressor. The impeller shroud has been defined as a counter-rotating wall which models the influence of relative motion between the rotating impeller and a stationary shroud. For the wetted surfaces, no roughness has been defined and the compressor walls have been assumed to be hydraulically smooth.

For the compressor, total pressure and temperature of 101.325kPa and 288.15K have been defined at the inlet respectively, which correspond to the International Standard Atmosphere (ISA) ambient conditions. Using the ISA conditions prevents a correction of the non-dimensional parameters for change in atmospheric conditions and assists in comparing the simulated compressor map with the available experimental data. Turbulent intensity is also required to be defined at the inlet which is a ratio of the root-mean-square of turbulent velocity fluctuations and mean velocity. In the absence of related experimental data, a sensitivity analysis with different turbulent intensities (between 1% and 10%) has been performed. The analysis revealed a marginal change in the compressor performance and hence a medium turbulent intensity of 5% has been permanently maintained at the inlet boundary for all compressor simulations.

The outlet boundary condition has been defined to simulate the operating con-
ditions at specific speed lines by varying the mass flow rates or static pressures. The choice between mass flow rate and static pressure at the compressor outlet depends on the operating condition. Close to choke, where slightest variations in mass flow can result in a considerable change in performance, static pressure has been defined instead of mass flow rate to set the operating point. Remaining operating points have been simulated by controlling the mass flow rate at compressor outlet.

The interfaces between the inlet duct, impeller and the vaneless diffuser are defined as frozen rotor. In this approach, the rotating and stationary components have a fixed relative position, while the frame of reference and pitch (in case of rotational misalignment of components) undergo a transformation. The governing equations and fluxes are transformed accordingly across the interfaces. Frozen rotor models produce a steady-state solution for problems with multiple frames of reference and require much lower computational effort compared to the transient approach. Furthermore, since the impeller is the only bladed component and the diffuser does not contain any stator vanes, there are no transient coupling effects except the inherent unsteadiness of the turbulent flow. The selected steady-state approach is, therefore, suitable for the analysis.

4.3.4 Turbulence Model Selection

Fundamentally, turbulence models approximate the Reynolds stress tensor in the RANS equations. However, since they are engineering approximations, it is important to select the most suitable model representing the fluid characteristics for a specific application. Out of many different models available, the two-equation turbulence models, $k-\epsilon$ and $k-\omega$ are widely used. The $k-\epsilon$ turbulence model has been employed over a wide range of cases with a varying degree of success. However, it is inaccurate for flows with adverse pressure gradients and also extremely difficult to integrate through the viscous sublayer. The $k-\omega$ model on the other hand, has several advantages over the $k-\epsilon$ model. The model is very accurate for 2D boundary layers with variable pressure gradients and performs particularly well in simulating the separated flows. The only weakness of the $k-\omega$ model appears to be in its sensitivity to freestream boundary conditions for free shear flows, where the $k-\epsilon$ model does not.

Considering the pros and cons of the two widely used two-equation turbulence models, the $k-\omega$ Shear Stress Transport (SST) turbulence model [3] has been used for the test case compressor simulations. It is a combination of the $k-\omega$ model (in the inner boundary layer) and the $k-\epsilon$ model (in the outer region and outside of the boundary layer). A blending function is used to switch from one model to the other depending on the type of flow (free stream or near-wall). One of the main advantages of using the $k-\omega$ formulation is the near-wall treatment for low-Reynolds number computations. The $k-\omega$ based SST model gives more accurate predictions, when there are flow separations under adverse pressure gradients compared to the standard two-equation turbulence models.
4.4 Performance Evaluation and Validation

The test case compressor has been simulated steady-state at different operating points ranging from stall to choke at 220,000rpm rotational speed. The results have been compared with the available test data for validation. It is important to mention that the test data has been obtained for the complete turbocharger compressor with fluid properties measured at compressor inlet and volute outlet stations. The fluid domain for CFD, however, comprises of the impeller and vaneless diffuser only. In order to include the volute performance for comparison with the test data, the volute module from the 1D meanline model has been used. Consequently, the flow properties at volute outlet have been calculated using the diffuser exit conditions determined by the CFD analysis.

Figure 4.3 contains the speed lines representing the CFD model and test data in form of pressure ratio and isentropic efficiency plots. A good overall agreement can be observed between the two data sets. Some discrepancy can also be seen for the CFD simulations made at low mass flow rate operating points. Results show that the CFD simulations are predicting a downward kink in the pressure ratio plot as the compressor operates at low mass flow rates. Such reduction in pressure ratio shows an onset of a rotating stall in the impeller, which is instability caused by cells of stalled flow rotating around the annulus, at a fraction of the rotor speed. Although rotating stall is an unsteady (time-dependent) phenomenon, the steady-state simulations are indicating an unstable compressor operation at low mass flow rates. Rotating stall can subsequently lead to surge (by growth of the stalled area till it occupies the total annular circumference), or the system may arrive at a new stable operating point on the stalled characteristic, with severely reduced performance and efficiency [4]. The test data, on the other hand, does not show...
such instability. Possible reasons for the difference in CFD and test data speed lines are the numerical errors, steady-state simulations with the turbulence model, and differences in the 3D geometric model compared to the actual impeller. Furthermore, the wetted compressor surfaces have been assumed to be hydraulically smooth, i.e., no wall roughness has been modeled in CFD. The positive influence of roughness to compressor stability at an expense of performance is hence, not predicted by the simulations. Nonetheless, the best-efficiency operating point is of main interest in this study and has been modeled well with CFD, as indicated by the pressure ratio and isentropic efficiency plots.

4.5 Two-Zone Flow Investigation

The investigation begins with visualization of flow behavior in the impeller passage. Figure 4.4 illustrates the secondary flow development in form of relative velocity streamlines at stall, best-efficiency and choke operating points for 220,000rpm rotational speed. The instability in form of vorticity in streamlines can be seen increasing successively as the compressor is throttled to operate from high to low mass flow rates. The distinct secondary flow is found even at the choke operating condition (now comprising mainly of over-tip leakage flow) with more mass flow in the vortex structure, but contained in a smaller passage area compared to the secondary flow for operation at stall.

The full blades, being exposed to the incoming flow, are mainly responsible for the flow separation at the inducer. The separated flow subsequently interacts and influences the splitter blades. The secondary flow is formed due to mixing of separated flow from the leading edges with over-tip leakage or clearance gap flow migrating from the pressure to suction side of the blades. The splitter blades also contribute to the secondary flow under the influence of the full blades. The remaining core flow, also known as the primary flow, passes through the impeller passage and is found mostly at the mid-span and hub span-wise locations.

The overall magnitude of secondary flow can be observed to be a function of the operating mass flow rate. A large and vigorous vorticity is present in the impeller passage at stall operating point ($\dot{m} = 40\text{g/s}$) along with a reversed flow in the inlet. The secondary flow dominates most of the shroud region, containing the weak and low-momentum flow at stall operating point. The flow finally leaves the impeller at high absolute angle into the vaneless diffuser, where it is slowed down for static pressure recovery. An additional diffusion also takes place in the volute (not modeled in CFD) at stall operating point. The throughflow velocity in the volute reduces to maintain the conservation of mass as the volute cross-sectional areas at different azimuth angles (originally designed for best-efficiency operating point) are large for a small mass flow rate. A large flow separation takes place into the diffuser due to high incidence angle formed between the incoming from the diffuser and the tongue of the volute at stall operating point. Flow separation at the tongue is typically responsible for triggering compressor surge.
Figure 4.4: Relative velocity streamlines representing the secondary flow vortex structure at different operating points for 220,000rpm speed line
At the best-efficiency operating point ($\dot{m} = 60g/s$), the secondary flow is observed to have moved downstream in the impeller passage with a considerable reduction in its magnitude compared to the stall operating point. The flow reversal is removed and the secondary flow comprises mostly of the tip-leakage flow as the impeller operates at an optimum incidence with the incoming flow. The flow leaves the impeller at a smaller absolute angle compared to stall operating point and undergoes diffusion in the vaneless diffuser. No further diffusion is expected to take place in the volute at this operating point as the volute cross-sectional area is defined accordingly to maintain a constant throughflow velocity in the volute. This results in an optimum incidence angle between the diffuser flow and tongue with minimum flow separations.

At the choke operating point ($\dot{m} = 82g/s$), the secondary flow is pushed further downstream, where it consists of the tip-leakage flow mainly, similar to the best-efficiency operation. The splitter blades still exhibit a large secondary flow vortex. Nonetheless, the overall size of the secondary flow vortex is considerably reduced compared to stall and best-efficiency operating points. The flow exits the impeller at low absolute angle and is diffused in the vaneless diffuser. In the volute, the flow will accelerate at the expense of static pressure since throughflow velocity increases in order to maintain the conservation of mass. Flow would separate at the tongue due to large incidence angle between the approaching flow from the diffuser and the tongue. Unlike the tongue separation at stall operating point, the separated flow at choke would be found in the volute outlet, thereby causing a considerable loss in compressor performance.

From the current representation of the flow in the impeller passage, the secondary zone indeed appears as a reservoir for low-energy flow coming from various sources. The secondary flow mass flux and the passage area engulfed by it, determine the entropy-gain during compression and eventually, the impeller efficiency. A proper scrutiny of the secondary flow and accurate representation of the secondary flow properties in shape of an empirical model is hence important for fundamental compressor performance evaluation. Such an empirical model can be used to improve the 1D model and its prediction capability. Quantification of the secondary flow properties at different operating conditions along a speed line is, therefore, required.

### 4.5.1 Two-Zone Flow Quantification

Although the secondary flow begins in the inducer region of the impeller, the flow structure formed at the trailing edge is critical and determines the end compressor performance. As the flow leaves the impeller blades, a rapid mixing takes place between the low momentum secondary flow (covering mostly the region from mid-span to shroud) and the almost isentropic primary flow. During the description of the 1D meanline model in the previous chapter, secondary flow mass flux fraction $\chi$ and area fraction $\epsilon$ were introduced. A correlation between $\chi$ and $\epsilon$ was established through a simple quadratic equation given in ref. [5]. The correlation represents the
variation in the secondary flow properties as a function of the operating condition only. However, there are other factors involved in the formation and variation of secondary flow in the impeller passage including the geometrical features and Reynolds number. Quantifying the influence of all such variables on secondary flow structure requires a comprehensive database collected from measurements performed over a considerable number of compressor test cases. One such analysis has been given in ref. [6], where empirical models have been developed to predict $\chi$ and $\epsilon$ using high-quality test data, applicable to a broad range of turbomachines. However, due to lack of such data, the present quantification of the secondary flow properties is restricted to the data obtained from the CFD simulations and estimation of only the quadratic empirical correlation between the mass flux and area fractions.

In ref. [7], the secondary flow has been defined and quantified by selecting one of the flow properties such as the meridional velocity, isentropic efficiency or the absolute flow angle. A particular isoline of this variable then forms the threshold and represents the border between the primary and secondary flow zones. According to ref. [8], a wake exists where the local value of the meridional velocity component is less or equal to 90% of the global mean value. Numerical analysis on a 2D impeller presented in ref. [9] considers 98% of averaged efficiency and 90% of meridional velocity as the threshold definition between the two zones and the area with a local value below the threshold is assigned as part of the secondary zone. A similar approach has been presented in ref. [10] for quantification of the secondary flow in the impeller passage with varying backsweps and exducer heights using CFD and experimentation. Here, the secondary mass flux fraction $\chi$ and area fraction $\epsilon$ are calculated according to Eq. (4.1), where the locally calculated efficiency $\eta_i$ is compared to the mass averaged efficiency at the impeller exit plane with a condition provided as,

$$\text{if } [\eta_i \leq (1 - k_s) \bar{\eta}] \text{ then } \dot{m} = \dot{m}_s$$

where $k_s$ is a constant chosen arbitrarily equal to 0.05 to estimate the threshold representing the transition between jet and wake flow. It is important to mention that the compressors used in these references are much larger and geometrically different compared to the test case turbocharger compressor under evaluation in this study. Therefore, a new quantification of the two-zone flow is needed.

A slightly different approach has been employed for the test case compressor. The mass flow distribution at the trailing edge of impeller blades has been used to determine the secondary mass flux fraction $\chi$, while the secondary flow area fraction $\epsilon$ has been estimated using the relative velocity distribution at the same location. This selection of flow properties to identify and quantify the two-zone flow is in accordance with the velocity triangle formulation, where the passage mass flow is represented by the meridional velocity component (similar to other references provided above), while the flow in a rotating frame, such as the impeller, is represented by the relative velocity component. The trailing edge has been
selected as a measurement location since it determines the end of work transfer region, where the primary and secondary flow distribution will determine the final impeller performance. Suitable thresholds have to be selected, which would form the isolines between the primary and secondary zones eventually. As the flow leaves the blades, a rapid mixing takes place between the primary and secondary flows resulting in a large mixing loss.

The plots in Fig. 4.5 represent the distribution of mass flux (mass flow per unit area) at impeller trailing edge. The plots, obtained from CFD, contain the blade-to-blade variation of the mass flux at three span-wise locations i.e. hub, mid-span and shroud for stall, best-efficiency and choke operating points. The plots indicate a significant variation in the mass flux at these locations for all the operating conditions. For the stall operating point ($\dot{m} = 40$g/s), the trailing edge is mostly covered by the low-momentum flow, as seen by considerably low mass fluxes (approximately 150kg/m$^2$/s to 200kg/m$^2$/s) at shroud and parts of mid-span and hub. As mass flow rate is increased to the best-efficiency operating point ($\dot{m} = 60$g/s), the secondary flow presence at mid-span and hub is reduced compared to stall operating point, while the shroud is still fairly covered with the low-momentum flow. Finally, at the choke operating point ($\dot{m} = 82$g/s), the low-momentum flow is significantly reduced and is mainly confined to the shroud. Considering the mass flux distribution for the three operating conditions at impeller trailing edge, a threshold of 150kg/m$^2$/s is the most reasonable selection below which the passage flow can be considered a part of the secondary zone flow.

In order to calculate the mass flow distribution in the primary and secondary zones, approximate passage area covered by the two zones at trailing edge has to be determined. For this purpose, additional data has been obtained from the numerical simulations in shape of contour plots for mass flux at the three particular operating points under consideration. Figure 4.6a shows the contour plots and evidently, with a more refined illustration of mass flux at the trailing edge. The primary and secondary zones can be distinguished by a relatively sharp change in mass flux between the two flow streams. The assumed threshold of 150kg/m$^2$/s forms the isoline, separating the two zones. By averaging the mass flux below 150kg/m$^2$/s threshold and estimating the corresponding passage area, the secondary zone mass flow fraction $\chi$ has been determined for the three operating points. Similar procedure has been applied for the remaining simulated operating points as well for the 220,000rpm speed line.

To estimate the secondary flow area fraction $\epsilon$, contours of relative velocity at the trailing edge, as illustrated in Fig. 4.6b, have also been obtained from CFD. Once more, a threshold has to be defined which would form an isoline between the primary and secondary flow streams for selecting the passage area engulfed by the secondary zone. In order to identify a reasonable threshold, the secondary flow vortex structure presented in Fig. 4.4 has been utilized. The passage area covered by secondary zone vortex at the trailing edge has been marked on the contour plots and measured for all the simulated operating points. Using the approximations, the secondary flow area fraction $\epsilon$ is finally calculated. The secondary zone data
Figure 4.5: Mass flow distribution at impeller trailing edge for stall, best-efficiency and choke operating conditions at 220,000rpm rotational speed
collected from the comprehensive CFD analysis has been plotted and a quadratic relationship, presented by Eq. (4.2), has been obtained, which correlates the secondary flow mass flux fraction $\chi$ and area fraction $\epsilon$ as,

$$\chi = 0.25\epsilon^2 + 0.15\epsilon$$  \hspace{1cm} (4.2)

Figure 4.6 shows the plots representing the new correlation for the test case turbocharger compressor compared to the correlations from other data sets presented in [5]. A difference can be seen between the three plotted data sets. Despite being almost half the diameter of the smallest reference Concepts ETI (CETI) impeller, the test case impeller is performing comparatively better in terms of secondary flow development and progression according to the correlation obtained from CFD. For compressor operations near stall (towards right of the plot) for which a large secondary zone is located in the impeller passage, the secondary mass flux fraction
4.6 Improvement of the 1D Meanline Model

The two-zone flow investigation has given an insight to the compressor flow structure and its behavior at different operating conditions for a particular rotational speed. To consider the 1D meanline model for uncertainty quantification (UQ) and subsequent robust design optimization of the test case impeller, it is necessary to carry out a comprehensive evaluation of its accuracy in predicting the compressor performance, when subjected to geometric deviations. A high fidelity CFD data is required as reference in this case, for comparison with the 1D model outcomes. The following sections of the chapter illustrate the evaluation of the 1D model, including its calibration using the CFD data, and its eventual validation.

4.6.1 Design Space Definition

Seven main impeller geometric parameters have been selected and assigned a deviation range as given in Table 4.1. Note that the deviations in the inlet tip radius $r_{1t}$ and exducer height $b_2$ have been modeled as the inducer tip-clearance $t_{clr,1}$
and exducer tip-clearance $t_{clr,2}$. Manufacturing deviations introduced during impeller trimming will ultimately vary the tip-clearance gaps between the impeller and the shroud cover formed by the volute. Consequently, such an arrangement will ensure that only the geometric deviations in the impeller are modeled, with constant shroud cover geometry. The geometric parameters and their deviation ranges given in Table 4.1, have been applied to model the compressor performance variation due to deviations in the impeller geometry using the 1D meanline model and CFD.

### 4.6.2 Impeller Parameterization and CFD Evaluation

The fluid domain given in Fig. 4.1, which consists of the inlet duct, impeller and the vaneless diffuser, has been used to implement the geometric deviations in the impeller. The preprocessing has been automated in Matlab, i.e., the BladeGen script file is accessed and altered in Matlab for each impeller geometry. The meshes are processed in TurboGrid, where tip-clearances are also added. The automation subsequently delivers the files ready for execution in Ansys CFX.

The parameterization of the fluid domain is shown in Fig. 4.8. The size of the inlet duct is controlled by the inlet hub radius $r_{1h}$ and inlet shroud radius $r_{1s}$. The impeller hub and shroud profiles have been converted to third-order Bézier curves, which provide an adequate resolution and control of the impeller shape. The inlet shroud radius $r_{1s}$ and the shroud profile are maintained constant, while the inlet tip radius $r_{1t}$ only varies when the inducer tip-clearance $t_{clr,1}$ is added during grid processing in TurboGrid. Furthermore, since the turbocharger comprises of a pinched vaneless diffuser, the exducer height $b_2$ has been combined with the tip radius $r_2$, such that any deviation in $r_2$ will increase or decrease $b_2$ accordingly, following the pinch angle of the vaneless diffuser. This maintains a consistent vaneless diffuser geometry and replicates the trimming process, where $b_2$ increases with a reduction in $r_2$, and vice versa. After adjusting $b_2$, the exducer tip-clearance $t_{clr,2}$ is finally added.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Deviation</th>
<th>Lower</th>
<th>Nominal</th>
<th>Upper</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inducer tip-clearance, $t_{clr,1}$ [mm]</td>
<td>±0.1</td>
<td>0.05</td>
<td>0.15</td>
<td>0.25</td>
</tr>
<tr>
<td>Exducer tip-clearance, $t_{clr,2}$ [mm]</td>
<td>±0.1</td>
<td>0.05</td>
<td>0.15</td>
<td>0.25</td>
</tr>
<tr>
<td>Inlet hub radius, $r_{1h}$ [mm]</td>
<td>±0.2</td>
<td>4.6</td>
<td>4.8</td>
<td>5.0</td>
</tr>
<tr>
<td>Tip radius, $r_2$ [mm]</td>
<td>±0.5</td>
<td>18.0</td>
<td>18.5</td>
<td>19.0</td>
</tr>
<tr>
<td>Blade thickness, $t_b$ [mm]</td>
<td>±0.1</td>
<td>0.4</td>
<td>0.5</td>
<td>0.6</td>
</tr>
<tr>
<td>Inlet blade angle at tip*, $\beta_{1bt}$ [deg]</td>
<td>$±2^\circ$</td>
<td>$63^\circ$</td>
<td>$65^\circ$</td>
<td>$67^\circ$</td>
</tr>
<tr>
<td>Backsweep angle, $\beta_{2b}$ [deg]</td>
<td>$±2^\circ$</td>
<td>33°</td>
<td>35°</td>
<td>37°</td>
</tr>
</tbody>
</table>

* similar deviation is applied to inlet blade angles at mid-span and hub
4.6. IMPROVEMENT OF THE 1D MEANLINE MODEL

Figure 4.8: Impeller parameterization

The blade shape is controlled by the blade angles at inducer and exducer locations. These include the backsweep angle $\beta_{2b}$ and the inlet blade angles $\beta_{1b}$. Since $\beta_{1b}$ varies span-wise from hub to shroud, the $\pm 2^\circ$ deviation is incorporated collectively, i.e., the inlet blade angle deviates by the same magnitude at the hub, mid-span and tip locations. The remaining blade shape is kept constant. The variation in geometry and Bézier control points has been restricted to a single dimension, as shown in Fig. 4.8 by their respective direction vectors.

The mesh size, boundary conditions and the turbulence model given in section 4.3 are used for the studies presented hereafter. Furthermore, only the best-efficiency operating point (60g/s at 220,000rpm rotational speed) has been considered for the present evaluation. The CFD results have been plotted in Fig. 4.9 and discussed in section 4.6.4 during validation of the 1D meanline model.

4.6.3 Calibration of the 1D Meanline Model

Before evaluating the 1D meanline model, an attempt has been made to calibrate its empirical modules using the CFD data obtained from the parametric evaluation presented in section 4.6.2. Three important aspects, which significantly influence the 1D compressor performance calculations have to be highlighted here:

- **Two-elements-in-series model.** Limitations in the two-elements-in-series (TEIS) model have been described in chapter 3. The TEIS model assigns constant values to the two effectiveness parameters $\eta_a$ and $\eta_b$, which provide
a reasonable basis to empirically predict the impeller diffusion characteristics at different operating points and rotational speeds. However, the effectiveness parameters are also dependent on the impeller geometry, for e.g., incidence or turning angle, tip-clearance, blade thickness, backsweep angle, etc. Dedicated parametric models are, therefore, required to predict the variation in the effectiveness parameters including geometric changes. For instance, extended models for predicting the effectiveness parameters $\eta_a$ and $\eta_b$ have been presented in ref. [6]. These models have been developed from a large experimental database belonging to hundreds of compressor stages. In ref. [5], it is recommended that researchers use their indigenous data to develop such extended models. Such models are not available for the test case turbocharger compressor due to lack of appropriate database. Therefore, an acceptable prediction of impeller diffusion characteristics subjected to geometric deviations is not possible, when using the standard TEIS model.

- **Secondary zone mass flux fraction $\chi$.** The two-zone modeling can be termed as case-specific in terms of computation of the mass flow distribution between the primary and secondary zones. This distribution is generally represented by the secondary mass flux fraction $\chi$ and the secondary flow area fraction $\epsilon$ in a 1D meanline model. Early researchers continued to use a constant value for $\chi$, which has been considered a serious modeling defect in ref. [11]. Estimate of $\chi$ was based on compressor size and quality of design. In reality, the secondary zone is not constant; rather it is considerably influenced by the compressor mass flow rate and geometry. The 1D meanline model, therefore, may not deliver the high level of accuracy required to predict the compressor performance variation as a result of geometric deviations.

- **Secondary zone relative flow angle $\beta_{2s}$.** The standard two-zone model assumes the secondary flow as perfectly guided by the impeller blades with no deviation (i.e., $\beta_{2s} = \beta_{2b}$). Experiments performed in ref. [8] on a 350mm diameter centrifugal compressor impeller having a considerably high backsweep angle of $70^\circ$ revealed that this assumption is not correct. Furthermore, a tangential velocity factor $f_{us}$, ranging between 0.92-0.94, has been recommended to calculate the secondary flow tangential velocity $C_{\theta_{2s}}$ and relative flow angle $\beta_{2s}$. The recommended range is not suitable for the current test case compressor since it represents a larger and well designed impeller. A new estimate of $f_{us}$ is needed instead.

According to ref. [6], the effectiveness parameters $\eta_a$ and $\eta_b$ have a dominating impact on the shape and position of the overall head characteristics in a centrifugal compressor stage compared to the secondary zone mass flux fraction $\chi$. Furthermore, the secondary zone is a function of the amount of diffusion in the impeller passage. The flow separations and over-tip leakage are augmented by an increase in the impeller diffusion, and vice versa. A deviation in the impeller geometry will directly influence the passage flow area and eventually, the diffusion of
4.6. IMPROVEMENT OF THE 1D MEANLINE MODEL

flow. Therefore, it is logical to improve the estimation of diffusion ratio in the 1D meanline model to include the influence of impeller geometry as well. Diffusion ratio data has been obtained from the CFD evaluation of geometric deviations given in section 4.6.2. An assessment of the data revealed that the diffusion ratio \( DR \) (where \( DR = \frac{W_1}{W_2} \)) is mainly affected by the inlet blade angle \( \beta_1 \), inducer tip-clearance \( t_{clr,1} \), exducer tip-clearance \( t_{clr,2} \) and the backweep angle \( \beta_2 \). Therefore, relevant diffusion ratio data has been extracted from the CFD for these particular geometric parameters, and used to construct a second-order response surface model (a metamodel) using regression analysis. The response surface model will be explained in detail in chapter 5.

To model the secondary zone mass flux fraction \( \chi \) and area fraction \( \epsilon \), the correlation for CETI rotors, given in ref. [5], was initially employed in the 1D meanline model. Later, a modification has been made to the correlation using Eq. (4.2) obtained from CFD to improve the 1D performance calculations. However, this modification only account for the variation in secondary zone at different operating conditions, and the influence of impeller geometry is lacking.

The secondary flow tangential velocity parameter \( f_{us} \) has also been estimated from the CFD data and has been implemented in the 1D meanline model to calculate the secondary zone relative flow angle \( \beta_{2s} \) with a better accuracy. A range of 0.5-0.6 has been selected for \( f_{us} \) based on the CFD data at best-efficiency operating point. More enhanced modeling of \( f_{us} \) (such as a metamodel similar to one applied for passage diffusion estimation) has not been attempted since it has a marginal effect on the predictions made by the 1D meanline model.

4.6.4 Validation of the 1D Meanline Model

The calibrated 1D meanline model has been used to evaluate the variation in impeller performance due to geometric deviations. The impeller pressure ratio and isentropic efficiency have been considered as the output performance parameters in the analysis. The hub radius \( r_{1h} \) has been excluded from the analysis due to its negligible influence on impeller performance. Figure 4.9 presents the outcomes of the analysis including a comparison with the CFD data as a relative difference. The results are discussed in detail below:

- **Deviation in blade thickness \( t_b \).** Blade thickness \( t_b \) is an important parameter to consider as its deviation will have a direct impact on the passage area, thus affecting the flow capacity and diffusion characteristics of the impeller. Figure 4.9a shows that the thicker blades will result in considerable performance deterioration. Overall, the agreement between the 1D meanline model and CFD data is satisfactory. Some inconsistency is, however, seen in the 1D pressure ratio plot beyond the nominal \( t_b \) of 0.5mm.

- **Deviation in tip radius \( r_2 \).** Deviation in the tip radius \( r_2 \) has a significant influence on the delivery pressure and efficiency of the impeller, as confirmed
CHAPTER 4. PERFORMANCE EVALUATION IN 3D CFD

Figure 4.9: Performance evaluation of impeller geometric deviations using the 1D meanline model and comparison with the CFD data.
by Fig. 4.9b. Impeller pressure ratio improves at the cost of efficiency as \( r_2 \) is increased. Note that the tip height \( b_2 \) is also changing with \( r_2 \), following the diffuser pinch angle in order to maintain a constant shroud cover geometry. Overall, the 1D meanline model and CFD results show a good agreement.

- **Deviation in tip-clearances \( t_{clr,1} \) and \( t_{clr,2} \).** Tip-clearance is largely a function of the differential expansion between rotor and housing, thrust bearing clearance and tolerance build-up\(^1\) [12]. Deviation in the inlet tip radius \( r_{1t} \) and tip height \( b_2 \) due to manufacturing uncertainty, will eventually cause the inducer tip-clearance \( t_{clr,1} \) and exducer tip-clearance \( t_{clr,2} \) to vary accordingly for a constant shroud cover geometry. Figures 4.9c and 4.9d show that both pressure ratio and isentropic efficiency are considerably decreased as \( t_{clr,1} \) and \( t_{clr,2} \) increase. A notable disagreement has been found between the 1D meanline model and CFD results.

- **Deviation in inlet angle \( \beta_{1b} \).** Figure 4.9e illustrates the performance variation due to deviations in overall impeller inlet angle \( \beta_{1b} \). The plots predict an increase in the pressure ratio and isentropic efficiency as the incidence angle is increased. Beyond \(+2^\circ\) the performance deteriorates, indicating a stall. A considerable disagreement is seen between the 1D meanline model and CFD data.

- **Deviation in backsweep angle \( \beta_{2b} \).** Any deviation in the backsweep angle \( \beta_{2b} \) during manufacturing can influence the performance and structural integrity of the compressor impeller. Generally, each \( 10^\circ \) of backsweep is worth one to two points of stage efficiency [5]. Figure 4.9f shows that with an increase in the \( \beta_{2b} \), a significant reduction in pressure ratio occurs, while isentropic efficiency is improved. A good agreement is achieved between 1D meanline model and CFD data.

Evidently, the comparison between the 1D model outcomes and similar data obtained from high fidelity CFD modeling has revealed the shortcomings in the former. Although 1D model has been successively improved using various forms of CFD data, the predictions for geometric parameters including inducer tip-clearance \( t_{clr,1} \), exducer tip-clearance \( t_{clr,2} \) and the inlet angle \( \beta_{1b} \) are not satisfactory, thereby making the 1D model unsuitable for the intended UQ and robust design optimization.

### 4.7 Conclusions and Recommendations

The following conclusions can be drawn from the CFD analysis, 1D meanline model improvement and its validation for the test case turbocharger compressor:

---

\(^1\)Refers to lack of fit in assembly due to stack up of individual part tolerances.
The presence of an inherent secondary zone, consisting of separated boundary layer and over-tip leakage flow, has been confirmed through the CFD analysis. The passage area is distinctively divided into primary and secondary flow zones.

The secondary flow vortex initiates at the leading edge, on the suction side of the impeller blades. The secondary flow vortex is augmented by the over-tip leakage flow, which progresses to the exducer eventually. This low-momentum flow mostly resides near the shroud in the impeller passage. The intensity increases as the compressor operates at low mass flow rates due to increased leading edge flow separations and flow reversal.

Quantifying or measuring the secondary flow properties in terms of secondary mass flux fraction $\chi$ and secondary flow area fraction $\epsilon$ is a complicated task. The Meridional velocity component has been used to estimate $\chi$, while the relative velocity component has been used to determine $\epsilon$. The correlation given by Eq. (4.2), obtained from the CFD analysis, has been implemented in the 1D meanline model. The analysis revealed a slightly better secondary flow characteristics in the test case impeller at low mass flow rate operations compared to other reference impellers. Nonetheless, an experimental validation is always important to confirm the accuracy of the results obtained from CFD.

The 1D meanline model has been found unsuitable for UQ and robust design optimization due to lack of accuracy in predicting the centrifugal compressor performance. Further calibration of the 1D meanline model is not feasible; rather a metamodel derived from high fidelity CFD modeling could be a useful replacement, and is recommended.

References


5

Manufacturing Uncertainty Quantification

5.1 Preface

A considerably large number of samples is required to be evaluated in order to perform the uncertainty quantification (UQ) using the Monte Carlo simulation (MCS). Therefore, to carry out swift calculations, a metamodel has been constructed and trained through a comprehensive design of experiments (DoE) conducted in computational fluid dynamics (CFD). This chapter illustrates the preparation of the metamodel and its application to perform a detailed UQ for the centrifugal compressor impeller.

5.2 Metamodel Preparation

Quantification of manufacturing uncertainties via MCS requires an evaluation of a large number of samples to determine the variation in the output quantity of interest, due to uncertainties in the input variables. The Monte Carlo approach is stochastic, however, the calculations involved are deterministic and require proper modeling. Performing MCS in computationally intensive models, for instance CFD, would be impractical. Low fidelity models, including lower order models and metamodels are, therefore, preferred. Such models may lack the accuracy of the high fidelity models, but they will be able to perform a large number of calculations, with comparatively lesser computational effort. A lower order 1D meanline model has been evaluated in chapter 4. Although computationally inexpensive, the 1D model did not provide satisfactory results. A metamodel has been prepared instead to perform the UQ and subsequent robust design optimization of the test case impeller.
5.2.1 Metamodeling Scheme

A number of steps have been followed to construct a metamodel, as illustrated in Fig. 5.1. The procedure begins with the definition of a design space, followed by the application of a suitable DoE method. A number of DoE samples are defined in the design space. The samples are then preprocessed (section 4.6.2 of chapter 4) and evaluated in CFD. Finally, the metamodel is constructed over the DoE responses and necessary diagnosis is performed to check the metamodel fit and convergence. If needed, additional DoE samples are prepared and evaluated in CFD to satisfy the metamodel convergence.

5.2.2 Introduction to Design of Experiments

A metamodel is an approximation of a physical process. The most influential input parameters are sampled in a predefined design space and subsequently used to construct the metamodel. The technique of defining and investigating the possible conditions in an experiment or trial calculations involving multiple design variables is known as DoE [1]. The objective of DoE in such a context is to generate a data that can be used to fit a metamodel, which reliably predicts the true trends of the input-output relationship [2].

DoE methods can be divided into two main types – the classical DoE methods and the space-filling methods. Both methods are fundamentally different from each other. An understanding of their methodology and application is, therefore, necessary before making a selection. The following sections briefly describe the different DoE methods:
Classical DoE Methods

Classical DoE methods include the full- and fractional-factorial designs, central composite design and the Box-Behnken design. A common feature of these techniques is that the sample points are placed at the extreme ends of the design space. Furthermore, the designs used for fitting first-order or linear metamodels are called the first-order designs, while the designs used for fitting second-degree or quadratic metamodels are termed as the second-order designs.

Among the first-order designs, the two-level $2^k$ factorial (where $k$ is the number of design variables) are commonly used. In a $2^k$ factorial design, each design variable is measured at two levels, which can be normalized between values $-1$ and $+1$ corresponding to the so-called low and high levels respectively. This design consists of all possible combinations of $k$ factors at two levels such that the design matrix will eventually consist of all $+1$, all $-1$, and the combinations of $+1$ and $-1$ levels. In this case, the number of experimental runs is equal to $2^k$ provided that no single design point is replicated more than once (replication increases confidence in the design). It can be noted that the number of experiments required for the $2^k$ factorial method increase directly with the number of design variables. Therefore, the method is computationally infeasible for large number of design variables. The Plackett-Burman design can be used instead, which allows two levels for each of the $k$ design variables, similar to the $2^k$ design, but require $k+1$ number of experimental runs.

The first-order designs fail to study the interior layout of a design space as they tend to focus on the outer extremes. If the midpoints are also included, the design is converted to a three-level $3^k$ factorial, which can be used to fit a second-order metamodel. The most frequently used second-order designs are the $3^k$ factorial, central composite and the Box-Behnken designs.

The $3^k$ factorial design consists of all the combinations of the $k$ design variables having three levels each. If the levels are equally spaced, the design variables will have normalized values of $-1$, $0$ and $+1$ at three levels. The number of experiments for such a design is equal to $3^k$, which can be very large as the number of design variables increases. Hence, $3^k$ factorial designs are expensive to produce. As an alternative, the central composite design (CCD) is frequently used as a second-order design method, resulting in $2^k + 2k + n_0$ number of experiments, where $n_0$ is the number of centre points. The Box-Behnken designs differ from CCD in the positioning of extreme or corner design points. The Box-Behnken avoids all the corner points, which could be beneficial for experiments where the extreme points are difficult to evaluate.

Space-Filling DoE Methods

The space-filling DoE methods tend to extract the maximum amount of information regarding the underlying input-output relationship by defining sample points, which fill the design space in an optimum sense. Consequently, the metamodel is
fitted over a sample space, which also contains the internal details of the design space provided that a sufficient number of samples is used.

Monte Carlo is perhaps the simplest of space-filling DoE methods, wherein the basic idea is to use a random number generator to sample the design space. Being purely random, Monte Carlo method may not be completely space-filling, unless this shortcoming is compensated by a large number of samples. However, Monte Carlo method would be infeasible to employ for DoE, when evaluated with high-fidelity models, such as CFD, FEA, etc. Several pseudo Monte Carlo methods have been invented to address this deficiency. Some of the widely used pseudo Monte Carlo methods include stratified Monte Carlo, Latin hypercube sampling (LHS) and orthogonal array sampling.

LHS is a well-known and widely used space-filling DoE method. The underlying concept of LHS sampling method is to divide the design space into a number of bins of equal probability and generate pseudo random samples, such that for each design variable, no two design points lie in the same bin. Accordingly, when a 1D projection of the design is taken, there will be one and only one sample in each bin. Orthogonal Arrays is another useful method to manage and optimize the sample spread in a design space. It may be seen as a generalization of LHS sampling, with the 1D projections also uniformly spaced. Orthogonal arrays based LHS give a substantial improvement over standard LHS [3].

5.2.3 DoE for the Test Case Impeller

For the test case impeller, the seven geometric parameters considered for evaluation, along with their predefined deviation range, are given in Table 4.1 of chapter 4. The space-filling LHS method has been used to carry out the DoE. A total of 100 LHS designs were generated initially in Matlab, which were later increased to 200 for metamodel convergence. All the DoE samples have been parameterized following the preprocessing and solver setup described in section 4.6.2 of chapter 4. The DoE samples have been modeled in CFD to obtain the responses in form of impeller pressure ratio and isentropic efficiency. Figure 5.2 presents the response scatter from the CFD evaluation of the DoE samples. As evident, a significant variation in impeller performance is obtained. The impeller pressure ratio varies approximately from 2.4 to 3.1, with a mean value equal to 2.68. Similarly, the impeller isentropic efficiency shows a variation of approximately 74% to 85%, with a mean value of 80.98%. The data has been collected and used for metamodel construction for predicting impeller performance.

5.2.4 Introduction to Metamodels

In simple terms, metamodel is a mathematical approximation obtained as a function constructed on a database in space, which contains a few measured function values (such as the DoE responses). Fundamentally, a metamodel acts as a curve fit to the available data so that the results may be predicted without the use of the
primary source of that data, i.e., the computationally intensive simulation tools. Metamodels are constructed using the data drawn from high-fidelity models, and provide fast approximations of the function values at new design points, thereby making sensitivity and optimization studies feasible. However, preparation of a high quality and advance metamodel may also require a large computational time and effort.

Many metamodels are available, which include the polynomial response surface, artificial neural networks, radial basis function, kriging, etc. Two widely used metamodels, polynomial response surface and kriging, have been evaluated as possible candidates for the present study, and are detailed in the following sections of the chapter:

Response Surface Model

Response surface models [4] feature a method of fitting a polynomial function to discrete responses obtained from high fidelity calculations, performed over the DoE samples in a specified design space. It consists of a group of mathematical and statistical techniques employed to develop an adequate functional relationship between the output response $y$ and the input design variable $x$. The basic functional form of a response surface model is given as,

$$y(x) = f(x) + \epsilon$$  \hspace{1cm} (5.1)

where $y(x)$ is the unknown function of interest, $f(x)$ is the polynomial approximation of $x$, and $\epsilon$ is a random error that is assumed to be normally distributed with mean and variance equal to zero and $\sigma^2$, respectively. Based on the function, a first-order linear response surface, given by Eq. (5.2), and a second-order
quadratic response surface, given by Eq. (5.3), with \( n \) design variables can be expressed in their general functional forms as,

\[
\hat{y} = \beta_0 + \sum_{i=1}^{n} \beta_i x_i
\]  

(5.2)

\[
\hat{y} = \beta_0 + \sum_{i=1}^{n} \beta_i x_i + \sum_{i<j} \beta_{ij} x_i x_j + \sum_{i=1}^{n} \beta_{ii} x_i^2 + \epsilon
\]  

(5.3)

Parameters \( \beta \) and \( x \) represent the regression coefficients and the set of design variables, respectively. The method of least squares is typically used to estimate the regression coefficients by minimizing the sum of the squares of the random error. The least squares estimator of the regression coefficients is given as,

\[
\beta = (X'X)^{-1} X'y
\]  

(5.4)

where \( X \) is a design matrix of the sampled data points, \( X' \) is its transpose, and \( y \) is a column vector that contains the values of the measured response for each sample point.

The quadratic form of the response surface is widely used due to its simplicity and ability to cope with the data containing noise. However, because of its underlying methodology, a quadratic response surface will fail to predict any multiple peaks and troughs in the design space, which is disadvantageous in case of highly non-linear functions.

**Kriging Model**

Kriging [5] also belongs to the family of least squares algorithms, such as the response surface model. However, it exactly reproduces the observed data by interpolating between the data points, thus making the new predictions more accurate. The kriging method in its basic formulation, estimates the value of a function at some un-sampled location as the sum of two components — the global approximation (usually a response surface model) and a variation component representing the localized departures. However, unlike a response surface model, kriging model requires a considerable amount of training and development time with added complexity, while slightly more accurate predictions are obtained [6]. The basic functional form of kriging model is given by,

\[
y(x) = f(x) + Z(x)
\]  

(5.5)

where \( y(x) \) is the unknown function of interest, \( f(x) \) is the known global function, and \( Z(x) \) is the realization of a stochastic process with mean zero, variance \( \sigma^2 \), and non-zero covariance. While \( f(x) \) globally approximates in the design space, \( Z(x) \) creates localized deviations so that the kriging model interpolates through the \( n_s \)
number of sampled data points. Such interpolation requires the quantification of covariance, which is a measure of the relative change amongst the random sample points. The covariance matrix of \( Z(x) \) is given as,

\[
\text{Cov}[Z(x^i), Z(x^j)] = \sigma^2 R([R(x^i, x^j)] \quad (5.6)
\]

where \( R \) represents the correlation matrix and \( R(x^i, x^j) \) is a correlation function between any two of the \( k \) sampled data points \( x^i \) and \( x^j \). \( R \) is a \((n_s \times n_s)\) symmetric matrix with ones along the diagonal. A number of correlation functions are available [5]. The choice of a suitable correlation function depends on the underlying phenomenon, for e.g., the function to be optimized or the physical process being modeled. If the underlying phenomenon is continuously differentiable or non-linear, it is highly likely that the function will show a parabolic behavior near the origin. A Gaussian, cubic or spline function can be used for such cases. For the processes with a linear functional behavior near the origin, other correlation functions; linear, exponential or spherical types, would perform adequately. One of the most used functions is a Gaussian correlation function given as,

\[
R(x^i, x^j) = e^{-\sum_{k=1}^{n} \theta_k |x^i_k-x^j_k|^2} \quad (5.7)
\]

where \( n \) is the number of design variables while \( \theta_k \) are the unknown correlation parameters used to fit the model. The correlation parameters \( \theta_k \) (also known as hyperparameters) control the degree of non-linearity in the model. Smaller values for \( \theta_k \) indicate that the output is a smooth function, while larger values produce a highly non-linear behavior. Estimation of the correlation parameters \( \theta_k \) for every design variable is a complex task and requires maximization of the likelihood function given as,

\[
\max_{\theta_k > 0} \Phi(\theta_k) = - \left[ n \ln(\hat{\sigma}^2) + \ln |R| \right] / 2 \quad (5.8)
\]

Both \( \hat{\sigma}^2 \) and \( |R| \) in Eq. 5.8 are functions of the correlation parameters \( \theta_k \) given by the following equations as,

\[
\hat{\beta} = (f^T R^{-1} f)^{-1} f^T R^{-1} y \quad (5.9)
\]

\[
\hat{\sigma}^2 = \left[ (y - f \hat{\beta})^T R^{-1} (y - f \hat{\beta}) \right] / n_s \quad (5.10)
\]

While any value for \( \theta_k \) creates an interpolative kriging model, the best kriging model is found by solving the \( k \)-dimensional unconstrained linear optimization problem given in Eq. (5.8). Finally the newly predicted estimates, \( \hat{y}(x) \), of the response \( y(x) \) at untried values of \( x \) are given by,

\[
\hat{y} = \hat{\beta} + r^T(x) R^{-1} (y - f \hat{\beta}) \quad (5.11)
\]
where \( y \) is the column vector of length \( n_s \) that contains the measured responses, and \( f \) is a column vector of length \( n_s \) that is filled with ones when \( f(x) \) is considered a constant value. In Eq. (5.11), \( r^T(x) \) is the correlation vector of length \( n_s \) between an untried \( x \) and the sampled data points \( \{x^1, ..., x^{n_s}\} \) given as,

\[
 r^T(x) = [R(x, x^1), R(x, x^2), ..., R(x, x^{n_s})] \tag{5.12}
\]

Although the inclusion of covariance and the estimation of hyperparameters \( \theta_k \) make kriging a high fidelity metamodel, the complete modeling process can be time consuming. Determining the maximum likelihood estimates of the \( \theta_k \) parameters is a multi-dimensional optimization problem, which can require a significant computational effort. A large number of uniformly distributed DoE samples are also needed to construct a good kriging model without severe functional deviations in its approximations.

### 5.2.5 Metamodel for the Test Case Impeller

The two metamodels – quadratic response surface and kriging, have been constructed over the DoE responses collected from the comprehensive CFD analysis. The impeller pressure ratio and isentropic efficiency have been considered as the output parameters of interest. For each output parameter, a dedicated metamodel has been constructed. The deviation ranges given in Table 4.1 of chapter 4 have been simulated using the metamodels to obtain the corresponding impeller performance variation. Outputs from the metamodels have been compared with the CFD data (section 4.6.2 of chapter 4) for validation; an approach similar to the validation of the 1D meanline model. Selection of a suitable metamodel for UQ and robust design optimization for the test case impeller is based on this evaluation.

#### Quadratic Response Surface Model Construction and Evaluation

The DoE responses have been fitted with a second-order or quadratic response surface in Matlab. The modeling process begins with the calculation of the design matrix \( X \). The design matrix \( X \) is constructed in order to simplify the calculations and contains the input variables normalized as \(-1 \) to \(+1\) limits. The regression coefficients \( \beta \) are determined by the least squares estimator given by Eq. (5.4). Finally, the functional forms of the response surfaces are composed by employing the calculated regression coefficients \( \beta \) and design variables \( x \) in Eq. (5.3).

Before the response surface models are considered usable, it is necessary to assess their goodness of fit. The coefficient of determination \( R^2 \) provides a relevant measure, which is a ratio of sum of squares due to regression \( SS_R \) and the total sum of squares \( SS_T \) given as,

\[
 R^2 = \frac{SS_R}{SS_T} \quad \text{or} \quad 1 - \frac{SS_E}{SS_T} \tag{5.13}
\]
where $SS_T$ is a combination of the sum of squares due to regression $SS_R$ and the sum of squares due to error $SS_E$. These parameters are calculated as,

$$SS_R = \sum_{i=1}^{n}(\hat{y}_i - \overline{y})^2$$  \hspace{1cm} (5.14)

$$SS_E = \sum_{i=1}^{n}e_i^2 = \sum_{i=1}^{n}(y_i - \hat{y}_i)^2$$ \hspace{1cm} (5.15)

The terms $y_i$, $\overline{y}$ and $\hat{y}_i$ represent the measured responses, their mean and the metamodel responses, respectively. The term $e_i$ depicts the residual, which is a difference between the actual observation $y_i$ and the corresponding model response $\hat{y}_i$. Finally the total sum of squares $SS_T = SS_R + SS_E$ is determined as,

$$SS_T = \sum_{i=1}^{n}(\hat{y}_i - \overline{y})^2 + \sum_{i=1}^{n}e_i^2 = \sum_{i=1}^{n}(y_i - \overline{y})^2$$  \hspace{1cm} (5.16)

The coefficient of determination $R^2$ is, therefore, a measure of the amount of reduction in the variability of the actual or observed response $y$ obtained by using the regressor variables $x_1, x_2, ..., x_k$ in a response surface model. A maximum value equal to 1.0 for $R^2$ would occur when all the variation in the observed response values $y$ are emulated by the response surface model. However, a large value of $R^2$ does not necessarily imply that the regression model is actually good. Adding a new variable to the model will always increase $R^2$, whether the additional variable is statistically significant or not. Therefore, it is possible for models having large values of $R^2$ to predict poorly for new observations. As a consequence, an adjusted $R^2_{adj}$ statistical test is also used to provide a better measure of model fit, and is given as,

$$R^2_{adj} = 1 - \frac{SS_E/(n - p)}{SS_T/(n - 1)} = 1 - \left(\frac{n - 1}{n - p}\right)(1 - R^2)$$ \hspace{1cm} (5.17)

where $p = k + 1$ is the total number of regressors or regression coefficients $\beta$ in the response surface model while $n$ is the sample size.

The two coefficients of determination, $R^2$ and $R^2_{adj}$, for the quadratic response surfaces constructed over DoE samples ranging from 60 to 200, are compiled in Table 5.1. High values for the coefficients of determination at different sample sizes are achieved. The response surface model for impeller pressure ratio constructed over 200 LHS designs resulted in $R^2$ and $R^2_{adj}$ equal to 0.989 and 0.975, respectively. This indicates that the response surface model can emulate 98.9% and 97.5% of the variability in $y$, respectively, in predicting new observations. For the response surface representing impeller isentropic efficiency, $R^2$ and $R^2_{adj}$ are calculated to be 0.977 and 0.945, respectively, which also indicates that the response surface model can emulate 97.7% and 94.5% of the variability in $y$, respectively, to predict new observations. Typical values for the two coefficients range between 0.9 and 1.0, when the observed response values are accurately emulated by a response
Table 5.1: Response surface model diagnostics

<table>
<thead>
<tr>
<th>Measure</th>
<th>Pressure Ratio Π</th>
<th>Efficiency η</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R^2_{60}$</td>
<td>0.998</td>
<td>0.994</td>
</tr>
<tr>
<td>$R^2_{adj,60}$</td>
<td>0.990</td>
<td>0.971</td>
</tr>
<tr>
<td>$R^2_{100}$</td>
<td>0.995</td>
<td>0.983</td>
</tr>
<tr>
<td>$R^2_{adj,100}$</td>
<td>0.985</td>
<td>0.948</td>
</tr>
<tr>
<td>$R^2_{150}$</td>
<td>0.992</td>
<td>0.978</td>
</tr>
<tr>
<td>$R^2_{adj,150}$</td>
<td>0.979</td>
<td>0.943</td>
</tr>
<tr>
<td>$R^2_{200}$</td>
<td>0.989</td>
<td>0.977</td>
</tr>
<tr>
<td>$R^2_{adj,200}$</td>
<td>0.973</td>
<td>0.945</td>
</tr>
</tbody>
</table>

surface model. Therefore, high values of the coefficients of determination indicate a satisfactory fit of the two response surface models.

Although the response surface models for impeller pressure ratio and isentropic efficiency show high-quality fit at various sample sizes, it is not necessary that the models have evolved to a converged state, thereby predicting the true variation of impeller performance subjected to geometric deviations. Figure 5.3 shows a comparison between the outcomes from the two response surface models constructed for different sample sizes, and compared with the corresponding CFD data. Interestingly, as the number of DoE samples increases from 60 to a maximum of 200, the predictions made by response surface models gradually improve in accuracy in comparison to the CFD data. The most prominent improvement can be observed in the plots representing the impeller blade thickness $t_b$, incidence angle $\Delta \beta_{1b}$ and the backsweep angle $\beta_{2b}$, while the remaining parameters also advance closer to the benchmark CFD data. As convergence is achieved, the response surface models have successfully emulated the CFD data, thus replacing the CFD modeling for predicting impeller performance at new observations.

Overall, the two quadratic response surface models for impeller pressure ratio and isentropic efficiency have revealed high levels of fit and accuracy of prediction. The models improve as the number of DoE samples is step-wise increased. The convergence of response surface models emphasizes the importance of employing a sufficient sample size in building them for good accuracy of prediction.

**Kriging Model Construction and Evaluation**

The kriging models for impeller pressure ratio and isentropic efficiency have also been constructed over the DoE responses in Matlab. Fitting a kriging model requires an estimation of the hyperparameters $\theta_k$, which can be obtained by maximizing the likelihood function given by Eq. (5.8). A suitable optimization algo-
Figure 5.3: Performance evaluation of impeller geometric deviations using the response surface model and comparison with the CFD data
Algorithm is required to carry out the maximization. Pattern search and simulated annealing algorithms have been used for maximization of likelihood function in ref. [6, 7]. Consequently, pattern search has been selected to estimate the hyper-parameters $\theta_k$ in the present study. Pattern search methods are characterized by a series of exploratory moves that consider the behavior of the objective function at a pattern of points, all of which lie on a rational lattice [8]. The step size or search space is gradually reduced (by one half) after a convergence is achieved so that the iterates remain on the rational lattice.

In order to check the kriging model fit, the leave-one-out cross-validation [3, 9] method is regularly used. However, the procedure is computationally intensive as a large number of metamodels, corresponding to the applied number of DoE samples, have to be constructed for validation.

Similar to the response surface model convergence check, the convergence of the kriging models for impeller pressure ratio and isentropic efficiency has been evaluated by constructing the models independently for 100 and 200 DoE responses, and predicting the impeller performance variation due to selective deviations introduced in its geometry. The results have been compared with the benchmark CFD data. Figure 5.4 illustrates the evaluation of the kriging model convergence. From the plots, it can be observed that the kriging models tend to interpolate through the responses in order to predict the impeller performance more precisely compared to the quadratic response surface models. A satisfactory convergence, however, is not achieved even after 200 DoE samples have been included in the model database. Some improvement in the predictions is seen for tip radius $r_2$, inducer tip-clearance $t_{clr,1}$, and exducer tip-clearance $t_{clr,2}$ as the DoE samples are increased. For the remaining geometric parameters, the kriging models predict inaccurately.

Evidently, although the kriging model attempts to be more accurate than the quadratic response surface model by interpolating through the DoE samples, it fails to deliver satisfactory results. The most probable cause for this deficiency is a lack of sufficient number of DoE samples, especially for the inputs where the model predictions deviate significantly compared to the CFD data.

### 5.2.6 Metamodel Selection

The impeller performance does not exhibit a highly non-linear behavior as evident from the analyses presented above. The plots given in Fig. 5.3 for the predictions made by the response surface model reveal a very good match with the reference CFD data. Conversely, as seen in Fig. 5.4, the kriging model did not show a satisfactory agreement with the reference CFD data and requires further training by adding more DoE samples in its database. The response surface model is, therefore, a better option to consider in terms of accuracy of prediction and metamodel convergence. Consequently, it has been selected as a metamodel for the test case impeller and employed for UQ and robust design optimization.
5.2. METAMODEL PREPARATION

Figure 5.4: Performance evaluation of impeller geometric deviations using the kriging model and comparison with the CFD data
5.2.7 Parameter-Wise Sensitivity Ranking

After validation and selection of the response surface model as the metamodel for predicting impeller performance, the data plotted in Fig. 5.3 has been used for the sensitivity analysis. The most influential or sensitive geometric parameters, which contribute significantly to the variation in impeller performance when deviated from their nominal value, have to be identified. The objective is to rank the geometric parameters according to their sensitivity and select the most sensitive parameters for UQ and robust design optimization, in order to save unnecessary computational expense. Here sensitivity corresponds to the ratio of output (pressure ratio and isentropic efficiency in this case) to the specified deviation ranges of the input geometric parameters.

Figure 5.5 summarizes the impeller pressure ratio and isentropic efficiency sensitivities in percentages. Evidently, the inducer tip-clearance $t_{\text{clr},1}$ and the exducer tip-clearance $t_{\text{clr},2}$ are contributing the most to impeller performance variation, while blade thickness $t_b$ and tip radius $r_2$ are also sensitive geometric parameters. In comparison, the remaining geometric parameters have a marginal influence on impeller performance. Therefore, based on the sensitivity ranking, the inducer tip-clearance $t_{\text{clr},1}$, exducer tip-clearance $t_{\text{clr},2}$, along with the blade thickness $t_b$ and the tip radius $r_2$ have been selected as the most sensitive geometric parameters for the impeller.

It is important to mention that for the test case impeller, the magnitude of uncertainties introduced in the geometric parameters during manufacturing is unknown. It is quite possible that the parameters most sensitive may not be the most influenced by manufacturing uncertainties, while the least sensitive parameters may incur large manufacturing uncertainty. A large uncertainty carried by the geometric parameters with low sensitivities, once propagated, may not influence the output performance significantly, and vice versa. On the other hand, a highly sensitive geometric parameter with large manufacturing uncertainty will be the one most critical, ultimately causing a large variation in output performance.

5.3 Uncertainty Quantification

The response surface models for impeller pressure ratio and isentropic efficiency have been employed to propagate the manufacturing uncertainties using MCS. The resulting performance variation, compiled as the output probability distributions, has been statistically evaluated. The following sections of the chapter describe the different stages of manufacturing uncertainty quantification in detail.

5.3.1 Data Assimilation

The stochastic manufacturing uncertainties can be characterized by a statistical distribution with a specific probability density function (PDF) through data assimilation. The analysis is typically focused on the specific inputs required by the
5.3. **UNCERTAINTY QUANTIFICATION**

(a) Pressure ratio

(b) Isentropic efficiency

Figure 5.5: Sensitivity ranking of impeller performance due to geometric deviations predicted by the response surface models

A mathematical framework that will be applied in the MCS. The objective of conducting data assimilation is to characterize the uncertainties in the input parameters, based on the available information, which can be experimental observations, theoretical reasoning, expert opinion, etc. Knowledge of the input PDFs is important for precise UQ. However, process capability or geometric variability data from the turbocharger manufacturer is unavailable for the test case impeller. Absence of relevant process capability justifies the use of a uniform probability distribution for allocating the random Monte Carlo samples to propagate uncertainty. Uniform probability distribution is the simplest type in which all the quantities of a random input variable occur with equal probability. As a consequence, the computed results can be considered as conservative estimates of the actual predictions.
The deviation ranges given in Table 4.1 of chapter 4, for the different impeller geometric parameters were defined for constructing and evaluating the 1D mean-line model and the metamodels over a wide design space. New upper and lower limits to variation, i.e., the tolerance margins have to be defined for the selected geometric parameters to perform the UQ. Actual manufacturing tolerances for the impeller are unknown. Therefore, the tolerances, given in Table 5.2, have been assumed based on expert opinion.

5.3.2 Uncertainty Propagation

Uncertainty propagation problems essentially involve the calculation of statistical moments of the output, along with the complete probability distribution. Two main approaches — intrusive and non-intrusive, are applied to propagate uncertainty through computational models. Intrusive approaches require modifications of the existing deterministic codes [10]. Some common intrusive approaches include the perturbation expansion, Neumann series expansion and stochastic Galerkin methods. Non-intrusive approaches, on the other hand, use the deterministic models as a black-box for uncertainty propagation [11]. Popular non-intrusive approaches include the Monte Carlo, stochastic collocation and non-intrusive polynomial chaos methods. The non-intrusive metamodel based Monte Carlo approach has been preferred over the complex intrusive techniques for uncertainty propagation in the test case impeller.

Metamodel Based Monte Carlo Approach

Monte Carlo methods are still the benchmark [12]. Monte Carlo approach applies stochastic sampling to solve complex deterministic and probabilistic problems involving numerical integration and differentiation, optimization and uncertainty propagation. With inputs defined in their respective PDFs, the Monte Carlo method can be applied to compute the complete statistics of the response quantities of interest with an arbitrary level of accuracy provided sufficient number of samples is used [2]. A random number generator is employed to select the sample points, for instance \( x^{(1)}, x^{(2)}, ..., x^{(n)} \), in the design space where the response \( y(x) \) is evaluated using a computational model. Subsequently, the integral is approxi-

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Deviation</th>
<th>Lower</th>
<th>Nominal</th>
<th>Upper</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inducer tip clearance, ( t_{clr,1} ) [mm]</td>
<td>±0.05</td>
<td>0.1</td>
<td>0.15</td>
<td>0.2</td>
</tr>
<tr>
<td>Exducer tip clearance, ( t_{clr,2} ) [mm]</td>
<td>±0.05</td>
<td>0.1</td>
<td>0.15</td>
<td>0.2</td>
</tr>
<tr>
<td>Tip radius, ( r_2 ) [mm]</td>
<td>±0.1</td>
<td>18.4</td>
<td>18.5</td>
<td>18.6</td>
</tr>
<tr>
<td>Blade thickness, ( t_b ) [mm]</td>
<td>±0.05</td>
<td>0.45</td>
<td>0.5</td>
<td>0.55</td>
</tr>
</tbody>
</table>
5.3. **UNCERTAINTY QUANTIFICATION**

imated by an average of the realizations of $y$ given by the relation as,

$$\langle y(x) \rangle \approx \tilde{y} = \frac{1}{n} \sum_{i=1}^{n} y(x^{(n)})$$  \hspace{1cm} (5.18)

where $\tilde{y}$ is referred to as the Monte Carlo estimate. The variance of $\tilde{y}$ can be calculated as,

$$\text{Var}(\tilde{y}) = \frac{1}{n} \sum_{i=1}^{n} (y(x^{(n)}) - \tilde{y})^2 = \frac{\sigma_y^2}{n}$$ \hspace{1cm} (5.19)

Here $\sigma^2$ is a sample estimate of the variance of $y(x)$ and can be used to evaluate the accuracy of Monte Carlo estimate. Hence, the standard error of $\tilde{y}$ is given by $\sigma/\sqrt{n}$, which shows that the convergence rate of the Monte Carlo estimate is equivalent to $1/\sqrt{n}$. Equation (5.19) also shows that the standard error of $\tilde{y}$ is independent of the dimensions of the design space.

Although the methodology behind MCS is conceptually simple, the requirement of a large number of sample evaluations makes a direct application of MCS impossible with computationally intensive tools as CFD. Therefore, in the present context, a metamodel based MCS has been used to estimate the probabilistic measures of the deterministic functions having random arguments, i.e., the mean and standard deviation of impeller pressure ratio and isentropic efficiency subjected to random geometric variation.

**Impeller Performance Variation under Uncertainty**

A total of hundred thousand samples have been used for the metamodel based MCS for uncertainty propagation. The deterministic results have been used to calculate the mean $\mu$ and standard deviation $\sigma$ of impeller pressure ratio and isentropic efficiency. Moreover, the PDFs of impeller performance have been obtained, as shown in Fig. 5.6. According to the PDFs, impeller pressure ratio and isentropic efficiency exhibit normal or Gaussian distributions. The mean and standard deviation of impeller pressure ratio are 2.69 and 0.055, respectively, while the mean and standard deviation of impeller isentropic efficiency are 81.11% and 0.81, respectively. The resulting variation in pressure ratio ranges approximately from 2.53 to 2.85, while isentropic efficiency ranges from 78.69% to 85.53%, equally covering $\pm 3\sigma$ normal distributions.

Note that a significant number of impellers manufactured under uncertainty are located far away from the mean value. If arbitrary maximum and minimum bounds for impeller performance are specified, many of these impellers will be unacceptable performance-wise and eventually rejected. Improvement in the manufacturing process capability in order to achieve stricter tolerance margins may result in a reduction in output performance variation, but at a higher manufacturing cost. The attempted manufacturing uncertainty quantification, therefore, highlights the application of an optimization procedure to design the impeller with robustness against manufacturing uncertainties.
A large variation in impeller performance has been observed. The MCS data has been used to perform the analysis of variance (ANOVA) to identify the most uncertain impeller geometric parameters. ANOVA is a method for decomposing the variance in a measured outcome into source variance that can be explained, such as by a regression model, and variance that cannot be explained, such as by a random error. Linear regression is generally employed, which constructs a first-order response surface model through the scatter plots made between the output responses (experimental or computational) and the input design variables. The goodness of fit reveals the magnitude of uncertainty that each design variable introduces in the output. A similarity is, therefore, observed between regression analysis and ANOVA, where regression is forming the basis to carry out ANOVA. Finally, the statistical tests presented in section 5.2.5 for response surface model diagnostics have been used to reveal the uncertain input parameters.

Figure 5.7 shows the scatter plots of impeller performance variation. The impeller pressure ratio and isentropic efficiency, predicted by their respective response surface models, have been plotted against the sensitive impeller geometric parameters. The spread of impeller performance in the plots can already provide a preliminary estimate of the level of uncertainty each input parameter is introducing to the output performance uncertainty. Consider Fig. 5.7a, which shows the impeller performance scatter for blade thickness $t_b$. The slope of the scatter plots for impeller pressure ratio and isentropic efficiency is similar to the sensitivity gradient predicted during the sensitivity analysis (section 5.2.5). For a fixed value of $t_b$, the scatter observed in impeller performance is being caused by the uncertainty
5.3. UNCERTAINTY QUANTIFICATION

The decomposition of impeller performance uncertainty has been performed by applying ANOVA on the MCS data in Matlab. The sum of squares due to regression $SS_R$, partitioned for each of the impeller geometric parameters, and the sum of squares due to error $SS_E$ have been used to evaluate and rank the sources of variance. Uncertainty ranking of impeller geometric parameters is shown in Fig. 5.8. The inducer tip-clearance $t_{clr,1}$ and the exducer tip-clearance $t_{clr,2}$ are contributing the most to impeller performance uncertainty. The tip radius $r_2$ has more influence on uncertainty in pressure ratio, while having a marginal contribution to uncertainty in impeller isentropic efficiency. Conversely, uncertainty in impeller pressure ratio has a marginal contribution from $t_b$, while it has a consid-
Figure 5.8: Uncertainty ranking of the impeller geometric parameters decomposed by ANOVA

ANOVA has suggested that \( t_{clr,1} \) and \( t_{clr,2} \) are the most critical parameters due to their considerable influence on impeller performance. Since impeller castings are trimmed to desired dimensions by machining, strict tolerances have to be defined and regulated in order to avoid unwanted variability in tip-clearances and the resulting variability in impeller performance. Impeller blade thickness \( t_b \), although not important for uncertainty in impeller pressure ratio, has a large influence on uncertainty in impeller isentropic efficiency. On the other hand, tip radius \( r_2 \) is more critical for impeller pressure ratio uncertainty than the uncertainty in impeller isentropic efficiency.
5.4 Conclusions and Recommendations

The following conclusions can be drawn from the manufacturing uncertainty quantification of the test case impeller,

- The metamodel is a fine alternative to 1D meanline model. However, attention must be given to initial data processing for DoE as accuracy and convergence of metamodels is dependent on it.

- Response surface model is a useful metamodel for its simplicity and rate of convergence. The kriging model, on the other hand, is more accurate due to its interpolative methodology. Nonetheless, it requires comprehensive training, which is computationally very intensive.

- Data assimilation is important for determining the PDFs of input geometric parameters in order to model the influence of manufacturing uncertainties on performance with precision. Such data is not available for the test case impeller. Instead, a uniform probability distribution has provided a reasonable start.

- The metamodel based MCS approach provides a useful means for UQ. For the selected tolerance margins, the output performance variation is considerably large for the test case impeller.

- From sensitivity analysis and ANOVA, the inducer tip-clearance $t_{clr,1}$ and the exducer tip-clearance $t_{clr,2}$ have been identified as the most sensitive and most uncertain impeller geometric parameters.

References


6

Robust Design Optimization

6.1 Preface

The objective of carrying out a robust design optimization is to obtain an impeller design, for which performance is relatively insensitive to variability in geometry, without reducing the sources of variation, i.e., the manufacturing uncertainties. This chapter presents a robust design optimization of the test case impeller using an optimization algorithm combined with the metamodel based Monte Carlo simulation (MCS). Finally, the optimized robust impeller options have been evaluated against the baseline design and their advantages and disadvantages have been discussed.

6.2 Introduction to Optimization

In mathematical terms, optimization usually involves maximizing or minimizing the objective function by searching for optimal design variables within a specified search space with certain constraints. The solution of an optimization problem is a set of allowed values of the input variables for which the objective function reaches an optimal value.

In its most general form, optimization may be defined as the search for a set of inputs $x$ that optimize the outputs of an objective function $f(x)$, subjected to inequality constraints $g(x) \leq 0$ and equality constraints $h(x) = 0$. The functions $f(x)$, $g(x)$, and $h(x)$ may be represented by simple expressions, complex computer simulations or even large-scale experimental results.

Optimization problems can be subdivided into single- and multi-objective types, depending on the number of objectives to be optimized. If optimization of one objective is required, the problem is termed as a single-objective optimization. On the other hand, for optimizing a number of objectives simultaneously, the problem is considered a multi-objective optimization.
A single-objective optimization problem involving a set of input design variables can be mathematically formulated as,

\[
\begin{align*}
\text{minimize:} & \quad f(x) \\
\text{subject to:} & \quad g_j(x) \leq 0, \quad j = 1 \ldots m \quad (6.1) \\
& \quad h_k(x) = 0, \quad k = 1 \ldots l \quad (6.2) \\
& \quad x_{i,\min}(x) \leq x_i \leq x_{i,\max}(x), \quad i = 1 \ldots n \quad (6.3)
\end{align*}
\]

where \( x \) is a vector consisting of the design variables given as,

\[
x = \begin{bmatrix} x_1 \\ x_2 \\ \vdots \\ x_n \end{bmatrix}
\]  

(6.5)

It is important to make a distinction between objectives and constraints. Constraints, unlike the objectives, are only limits to certain properties of a system, which have to be satisfied during optimization of the objective function.

In some applications, more than one property of a system is required to be optimized. This type of optimization is called a multi-objective optimization and is formulated as,

\[
\begin{align*}
\text{minimize:} & \quad f_i(x), \quad i = 1 \ldots l \\
\text{subject to:} & \quad g_j(x) \leq 0, \quad j = 1 \ldots m \\
& \quad h_k(x) = 0, \quad k = 1 \ldots n \\
& \quad x_{p,\min}(x) \leq x_i \leq x_{p,\max}(x), \quad p = 1 \ldots q
\end{align*}
\]  

(6.6)

A multi-objective optimization problem is always conflicting as minimization of one objective results in maximization of the other. Since no single optimal solution exists, a so called Pareto front of non-dominated solutions is constructed. The Pareto front comprises of designs for which one objective cannot be improved without worsening the other. An example of the Pareto front for two objectives to be minimized, \( f_1 \) and \( f_2 \), is given in Fig. 6.1. Design \( D_1 \) is dominated by design \( D_2 \) since both objectives are improved with respect to \( D_2 \). However, design \( D_3 \) is not dominated by design \( D_2 \) and vice versa since an improvement of \( f_1 \) by going from design \( D_2 \) to design \( D_3 \) will lead to a deterioration of \( f_2 \). If no other design is found improving both objectives, the newest design will belong to the Pareto front, similar to \( D_2 \) and \( D_3 \).

Multi-objective optimization problem can also be solved by combining the objectives in one function and ranking their importance by employing appropriate weights. A weighted objective function is given as,

\[
F(x) = \sum_{i=1}^{l} w_if_i(x)
\]  

(6.10)
6.2. INTRODUCTION TO OPTIMIZATION

Figure 6.1: An example of the Pareto front in case of a multi-objective optimization. Design $D_1$ is dominated by designs $D_2$ and $D_3$, while $D_2$ and $D_3$ are non-dominating for each other; hence they are the Pareto optimal solutions

where $w_i$ are the weights given to the $i$-th objective function. In order to find multiple solutions, the problem is solved multiple times with different weight combinations. However, assignment of suitable weights is difficult and can influence the overall outcome of the optimization problem.

6.2.1 Optimization Methods and Selection

The objective function $f(x)$ categorizes the relationship between the inputs and the outputs in shape of models. The relationship can be linear, non-linear or even discontinuous. Moreover the function can be steady, time dependent or even stochastic in nature. The computational expense of an optimization problem is thus directly linked to the nature of the problem and the type of model involved. For instance, lower-fidelity models will benefit an optimization problem for their relatively lower computational time compared to high-fidelity models for which a single solution can take many hours to complete. On the other hand, the optimization method also incurs computational cost in addition to the type of problem being considered. Hence, a swift optimization method is also required to minimize the overall computational cost of a problem.

A variety of optimization methods exist to solve the optimization problems, which can be subdivided into two main classes — local and global optimization methods. In simple terms, local optimization aims at finding a local optimum in the neighborhood of an initial guess, whereas global optimization aims at finding the global optimum in the complete design space. Figure 6.2 sketches the difference
between the two optimization methods through a function \( f(x) \) with one variable \( x \) having a local and global minimum (where the derivative or slope of the function is zero) at \( x = x_1 \) and \( x = x_2 \) respectively.

Based on the search techniques used to determine the function optima (locally or globally) the optimization methods are further categorized into deterministic and stochastic types. Deterministic methods are used for local optimization problems and require some gradient information of the objective function, which can result in considerable computational costs. Calculation of gradients is carried out using various methods, for e.g., finite difference method, automatic differentiation (AD) and adjoint methods. Apart from requiring a continuous objective function, deterministic methods also have a tendency to be trapped in a local minimum. Steepest descent method, Newton’s method, conjugate gradient method and pattern search (pattern search does not require gradient information) are few of the examples of local deterministic optimization methods.

On the other hand, the stochastic search methods (also known as zero-order methods) are employed to solve global optimization problems with non-linear and discontinuous objective functions. Only the function values are used to search for the minimum (without the need to calculate any gradients or derivatives) by making a random or systematic sweep of the entire design space using an appropriate search algorithm. For multi-objective optimization problems requiring the calculation of a Pareto front, the stochastic optimization methods are advantageous since a number of function evaluations have to be made initially in order to locate the global minimum. However, such large number of evaluations even for a small number of design variables can result in high computational costs. Some of the stochastic optimization methods include random search, simulated annealing, genetic algorithms, tabu search, particle swarm optimization, fuzzy optimization and neural network based methods.

Efficiency and robustness are the main factors dictating the choice of an optimization algorithm for a particular application. The efficiency of an algorithm is measured in terms of the total number of evaluations required to trace the optimal
6.3. STOCHASTIC SEARCH METHODS

Design or a design of a specified performance level. Reducing the total number of evaluations has a large impact on the time required to find an optimized design. On the other hand, robustness of an optimization algorithm refers to its ability to converge at the same conclusion regardless of the starting conditions. This property is important for adding confidence in the results of an optimization algorithm.

For turbomachinery applications involving multidisciplinary optimization problems, robustness of an optimizer is of main importance as the complete design space is searched to locate the optimum design globally. Optimization efficiency has drastically increased by employing a metamodel assisted optimization method. Metamodel based optimization has become a frequent method to carry out a comprehensive design search in relatively short times. However, preparation of an accurate metamodel is a prerequisite and can be a computationally intensive task. For final validation, the optimized designs are simulated in a high-fidelity model and their solutions are added in the design of experiments (DoE) database for metamodel training.

For their efficiency and robustness, along with the need to simultaneously evaluate a number of designs for carrying out a multi-objective optimization and obtain Pareto optimal solutions, stochastic search methods are well-suited for the current application. Robust design optimization is necessarily a multi-objective problem since it aims to maximize the mean and minimize the variation of performance in a system. Therefore, stochastic search methods have been selected for robust design optimization of the test case impeller. The stochastic search methods are illustrated in detail in the following sections of the chapter.

6.3 Stochastic Search Methods

Zero-order or stochastic search methods require only the function evaluations to locate the optimum by making a systematic sweep of the design space. Random search are the simplest approaches, in which a large number of design inputs are evaluated in the objective function. The input with the minimum (or maximum depending on the optimization problem) function value and satisfying all the constraints will be the optimum. Although simple, this method is not efficient and, therefore, not feasible for complex optimization problems. Other stochastic search methods accelerate the optimization process by replacing the systematic sweep by a more intelligent selection of new designs using the information obtained during previous calculations. Commonly used stochastic search methods include simulated annealing and genetic algorithms.

6.3.1 Genetic Algorithms

Genetic algorithm (GA) is a subclass of the evolutionary algorithms that are population-based metaheuristic optimization methods inspired from biologi-
Figure 6.3: Illustration of the tournament selection method with $s = 2$ [9]

cal mechanisms such as mutation, crossover, natural selection, and survival of the fittest in order to refine a set of solution candidates iteratively. The GA optimization begins by generating a random population consisting of $n$ number of individuals (design points), each represented by a binary string similar to the representation of chromosomes belonging to living creatures. Generally, the number of individuals in a population is two to four times the number of design variables involved in an optimization problem. The population is evaluated by the objective function and the fitness level (output performance) of each individual is obtained. Good strings from the population are selected as parents for reproduction based on a selection procedure. The selection of parents is critical as mating between two individuals will influence the characteristics of the offspring. For instance, parents with an elitist or superior fitness levels will result in offspring with relatively superior characteristics. The selection stage ensures that highly fit individuals live and reproduce, and less fit individuals die. Various operations involved in the optimization process by GA are summarized in Fig. 6.4.

Mainly two selection methods are used – tournament and roulette wheel selection. The tournament selection [6,7], illustrated in Fig. 6.3, is a simple and effective selection method. In tournament selection, $s$ individuals are selected randomly from the population and the best among them is chosen as the first parent. The second parent is also selected in a similar way from the remaining population. Parameter $s$ represents the tournament size and can take values between 1 and $n$ (total population size). In ref. [8], a value of $s = 2$ is proposed as the tournament size in order to select the individuals with the highest fitness as parents. A large value of $s$ gives more chances to the best individuals to create the offspring and favor a rapid, but possibly a premature convergence to an optimum. On the contrary, a lower $s$ will ensure more diversity in the population by allowing individuals with less fitness to be selected as parents as well. The diversity of a population often allows the exploration of new domains of the search space, which is not possible if the binary strings of the individuals are too alike [2].
An alternative selection scheme, roulette wheel, selects an individual with a probability proportional to its fitness. More area is reserved on the roulette wheel for an individual with higher fitness level and, therefore, it has a better chance to be selected for parenting. However, the selection of parenting individuals takes more time to converge for the roulette wheel. The resulting population will also be less diverse due to more elitist selection.

During reproduction, crossover takes place by the recombination of genes in a living organism. Similarly in the GA, the binary strings representing the selected parents are recombined at a random position by crossover to generate new individuals. In most crossover operations, two individuals are picked at random from the mating pool generated by the selection process and some portion of their binary strings is exchanged between them. Single-point crossover operation is a commonly used process in which a crossover site is selected at random along the string length, and the binary digits lying on the right side of the crossover site are swapped between the two strings with a crossover probability $p_c$. A crossover probability $p_c$ preserves some of the good strings of the mating pool as part of the population in the next generation. Higher values of $p_c$ result in an increased mixing of parts of a binary string, however, with a disadvantage of dislocating good
parts of it as well. Generally, a $p_c$ of 70% to 90% is employed [2], which allows some of the strings to be retained in their original form in the new generation.

After completion of the crossover operation, mutation is applied on the new generation of individuals. The purpose of mutation is to avoid the algorithm from being trapped in a local minimum by changing a bit in the binary sequence from its original state (from 0 to 1 and vice versa) after crossover with a certain mutation probability $p_m$. This prevents the chromosomes of the individuals in a population from becoming too similar to each other and thus slowing or even stopping the evolution. Typical values for $p_m$ are around 0.1% to 0.8% [2].

Finally, each pair of parents is allowed to have two children for maintaining a constant population size. A new generation of individuals is formed eventually once the parents have been replaced by their offspring in a particular population. The GA cycle continues till new generations with improved values of fitness are obtained, and the process converges to the optimum fitness value of the objective function.

GA algorithm is inherently suited for multi-objective problems as they have the ability to find multiple Pareto-optimal solutions in one simulation run. A multi-objective optimization problem can be directly solved by applying the multi-objective genetic algorithm (MOGA) with an aim to obtain a population distributed evenly on a Pareto front. A Pareto ranking selects and removes the non-dominated individuals from the entire population of a particular generation. The Pareto front is constantly updated for each generation till convergence is achieved at a true Pareto front.
6.4 Robust Optimization of the Impeller

Many design optimization studies [10–14] have been performed on different types of turbomachinery using a variety of methods and algorithms. However, most of these studies are deterministic, where the design has been optimized for a particular geometry and the resultant performance. It is likely that deterministic designs, when subjected to manufacturing uncertainties, may result in a large variation in performance. The present study aims at carrying out a robust design optimization for the test case impeller using MOGA coupled to the metamodel based Monte Carlo simulator. The following sections of the chapter describe the optimization setup and results in detail.

6.4.1 Optimization Setup

Figure 6.5 illustrates the methodology used to perform a robust optimization of the turbocharger compressor impeller. While searching for designs with reduced variability in output performance, it is likely that a design with more robustness would not deliver the required level of performance. A robust design optimization, therefore, includes maximization of the mean performance with minimization of its variation. A multi-objective approach has been used to obtain a compromise between performance and robustness of the impeller design. Two objective functions have been defined in form of mean and standard deviation of the selected impeller performance parameters, i.e., pressure ratio and isentropic efficiency. The mean values of the two performance parameters have been normalized and combined in a single performance mean function (PMF) given as,

$$ PMF_{\text{min}} = \left\{ \left( \frac{\mu_\Pi}{\mu_\Pi,\text{nominal}} \right) + \left( \frac{\mu_\eta}{\mu_\eta,\text{nominal}} \right) \right\}^{-1} $$ (6.11)

Note that the minimization of the performance mean function in Eq. (6.11) would eventually maximize the mean performance of the impeller. Similarly the performance variation function (PVF) combines the normalized standard deviations of the impeller pressure ratio and isentropic efficiency given as,

$$ PVF_{\text{min}} = \left( \frac{\sigma_\Pi}{\sigma_\Pi,\text{nominal}} \right) + \left( \frac{\sigma_\eta}{\sigma_\eta,\text{nominal}} \right) $$ (6.12)

The metamodel based Monte Carlo simulations have been coupled with MOGA available in Matlab. Two MOGA optimizations have been performed with different search spaces. Initially a shorter search space, designated as Case I and defined by the tolerance limits given in Table 5.2 of chapter 5, has been considered. The first MOGA optimization hence locates a robust design within the specified tolerance limits for the selected geometric parameters; inducer tip-clearance $t_{\text{clr},1}$, exducer tip-clearance $t_{\text{clr},2}$, exit radius $r_2$ and blade thickness $t_b$. The robust impeller design obtained from the Case I optimization is, therefore, not radically
different from the baseline configuration. The second MOGA optimization, designated as Case II, expands the search to an extended design space defined by the deviation ranges given in Table 4.1 of chapter 4 for the selected geometric parameters, thus allowing a larger change in the impeller design compared to the baseline configuration and Case I impeller designs.

For each design in a population, the metamodel based Monte Carlo simulation is performed within the tolerance limits specified in Table 5.2 of chapter 5 to determine the mean and standard deviation of the output performance. A hundred thousand Monte Carlo samples were used for uncertainty quantification (UQ) of the baseline impeller. However, convergence of mean and standard deviation is reached in approximately two to three thousand samples. Therefore, the robust optimization using MOGA is carried out using only three thousand Monte Carlo evaluations in order to save additional computational effort. Pareto optimal solutions are obtained from the multi-objective optimization using a population size of 100 with 800 generations, where generations represent the number of iterations. The tournament selection method has been used with a selection size of 2 in order to choose the elite individuals as parents having the highest fitness level. An intermediate value of 0.8 for the crossover probability $p_c$ has been applied, which determines the amount of blending of chromosomes in the parents to produce the next generation of solution candidates or offspring. If no crossover takes place, the offspring will be exactly the duplicates of the two parents. After crossover, the mutation operator is applied in the binary strings, which preserves the diversity of the offspring. The adaptive mutation function available for MOGA in Matlab
6.4. ROBUST OPTIMIZATION OF THE IMPELLER

Figure 6.7: Schematic comparison between the trims of the baseline impeller and the optimized robust impeller designs

has been employed to ensure that the least significant bits are more likely to be mutated in high-fitness chromosomes, thus to improve their accuracy [15].

6.4.2 Optimization Results

Pareto-optimal solutions obtained from MOGA optimization are presented in Fig. 6.6. Case I and Case II represent the short and extended search spaces respectively. It is obvious that both performance and robustness criterions cannot be enhanced simultaneously and a tradeoff is inevitable between the two objectives. Nonetheless, a comparison with the baseline can help in identifying the final robust impeller design delivering more or less a similar performance but more importantly, a potential design to manufacture in the presence of uncertainties.

Three different robust impeller designs; impeller A from Case I and impeller B and C from Case II, have been selected for further investigation. These impellers are expected to be more robust compared to the baseline design (also shown in Fig. 6.6 in terms of PMF and PVF). The metamodel based Monte Carlo simulation has been performed for each impeller design to determine the mean and standard deviation of impeller pressure ratio and isentropic efficiency. Table 6.1 summarizes the results along with the geometric data for each impeller design, while Fig. 6.7 presents a schematic illustration of the robust impellers compared to the baseline configuration. A general assessment of the data suggests that the increase in robustness for all the optimized impeller designs is caused by a reduction in inducer tip-clearance \( t_{clr,1} \), increase in exducer tip-clearance \( t_{clr,2} \) and a reduction in tip radius \( r_2 \) in comparison to the baseline impeller design. Furthermore, the
Table 6.1: Geometric and performance comparison between the baseline and robust impeller designs obtained from metamodel based MOGA optimization

<table>
<thead>
<tr>
<th>Measure</th>
<th>Baseline</th>
<th>A</th>
<th>B</th>
<th>C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inducer tip-clearance, ( t_{clr,1} ) [mm]</td>
<td>0.15</td>
<td>0.11</td>
<td>0.06</td>
<td>0.12</td>
</tr>
<tr>
<td>Exducer tip-clearance, ( t_{clr,2} ) [mm]</td>
<td>0.15</td>
<td>0.17</td>
<td>0.19</td>
<td>0.25</td>
</tr>
<tr>
<td>Tip radius, ( r_2 ) [mm]</td>
<td>18.50</td>
<td>18.41</td>
<td>18.03</td>
<td>18.02</td>
</tr>
<tr>
<td>Blade thickness, ( t_b ) [mm]</td>
<td>0.50</td>
<td>0.48</td>
<td>0.42</td>
<td>0.53</td>
</tr>
<tr>
<td>Nominal pressure ratio, ( \Pi ) [-]</td>
<td>2.68</td>
<td>2.66</td>
<td>2.50</td>
<td>2.46</td>
</tr>
<tr>
<td>Mean pressure ratio, ( \mu_\Pi ) [-]</td>
<td>2.69</td>
<td>2.67</td>
<td>2.60</td>
<td>2.47</td>
</tr>
<tr>
<td>Pressure ratio variation, ( \sigma_\Pi ) [-]</td>
<td>0.055</td>
<td>0.051</td>
<td>0.041</td>
<td>0.031</td>
</tr>
<tr>
<td>Nominal isentropic efficiency, ( \eta ) [%]</td>
<td>81.12</td>
<td>81.76</td>
<td>83.12</td>
<td>80.70</td>
</tr>
<tr>
<td>Mean isentropic efficiency, ( \mu_\eta ) [%]</td>
<td>81.11</td>
<td>81.79</td>
<td>83.14</td>
<td>80.66</td>
</tr>
<tr>
<td>Isentropic efficiency variation, ( \sigma_\eta ) [-]</td>
<td>0.81</td>
<td>0.73</td>
<td>0.57</td>
<td>0.61</td>
</tr>
</tbody>
</table>

variability in impeller isentropic efficiency is considerably more in magnitude than the variability in impeller pressure ratio.

The reduction in performance variability for impeller A and impeller B in comparison to the baseline is shown in Fig. 6.8a and Fig. 6.8b respectively by their corresponding probability distributions. The results have been normalized to have a common mean value in order to see the change in performance variability only. It is apparent that the size of search space has a significant influence on the location of the optimum design. If the level of robustness achieved by impeller A is considered satisfactory, the impeller design is ideal for having a similarity with the baseline impeller design in terms of geometry and performance. More robustness is achieved with impeller B, which is beneficial if the shift in mean performance is acceptable to the designer. However, the design may be structurally infeasible for a fail-safe compressor operation as values for inducer tip-clearance \( t_{clr,1} \) and blade thickness \( t_b \) are considerably lower compared to the baseline design. All in all, impeller A provides some level of robustness, which (along with its similarity in geometry and performance with the baseline) can be advantageous for high-quality manufacturing.

The mean performance delivered by impeller A is almost similar to the baseline impeller since Case I optimization was performed in a smaller search space (within the tolerance margins). The mean pressure ratio drops by 0.02 points, while mean isentropic efficiency is increased by 0.68% points. The standard deviations for both pressure ratio and isentropic efficiency have reduced by 7.8% and 10.1% respectively compared to the baseline design, thereby showing a relatively lower variability in performance. Geometrically; the inducer tip-clearance \( t_{clr,1} \), exit radius \( r_2 \) and blade thickness \( t_b \) are reduced, whereas the exducer tip-clearance \( t_{clr,2} \) is increased.

The mean performance delivered by impeller B is almost similar to the baseline impeller since Case I optimization was performed in a smaller search space (within the tolerance margins). The mean pressure ratio drops by 0.02 points, while mean isentropic efficiency is increased by 0.68% points. The standard deviations for both pressure ratio and isentropic efficiency have reduced by 7.8% and 10.1% respectively compared to the baseline design, thereby showing a relatively lower variability in performance. Geometrically; the inducer tip-clearance \( t_{clr,1} \), exit radius \( r_2 \) and blade thickness \( t_b \) are reduced, whereas the exducer tip-clearance \( t_{clr,2} \) is increased.
6.4. ROBUST OPTIMIZATION OF THE IMPELLER

Figure 6.8: Probability distributions of variation in performance of robust impeller designs and comparison with the baseline (denoted by ‘Bsl’ in the plots)

(a) Impeller A (white histogram - baseline, grey histogram - robust)

(b) Impeller B (white histogram - baseline, grey histogram - robust)

(c) Impeller C (white histogram - baseline, grey histogram - robust)
is increased. For impeller B in Case II optimization, relatively higher robustness than impeller A is achieved. The pressure ratio and isentropic efficiency variability is reduced by 34.1% and 40.3% respectively compared to the baseline design. However, the shift in mean performance is considerably larger than impeller A as mean pressure ratio reduces by 0.09 points, while mean isentropic efficiency increases by 2.03% points from the baseline. Impeller B shows a large change in geometry compared to the baseline design as the inducer tip-clearance $t_{\text{clr},1}$, exit radius $r_2$ and blade thickness $t_b$ are reduced, while the exducer tip-clearance $t_{\text{clr},2}$ increases considerably.

For a higher level of robustness without exceeding the structural limitations as perceived in the impeller B design, impeller C has also been considered as an alternative from Case II optimization. The variability in pressure ratio and isentropic efficiency for impeller C is reduced by 77% and 31.2% respectively from the baseline as shown in Fig. 6.8c. The performance drop is however, larger amongst all the robust impeller designs as the mean pressure ratio is reduced by 0.22 points and the isentropic efficiency is reduced by 0.45% points in comparison to the baseline impeller. The inducer tip-clearance $t_{\text{clr},1}$ is somewhat reduced, while a significant increase in the exducer tip-clearance $t_{\text{clr},2}$ is obtained compared to the baseline. The exit radius $r_2$ also shows a large reduction. However, the blade thickness $t_b$ is comparable to the baseline, thus demonstrating a structurally feasible design.

The nominal performance for all the optimized impellers including the baseline impeller is also given in Table 6.1. The difference between the nominal performance (given by the impeller pressure ratio and isentropic efficiency) and the mean performance (given by the mean impeller pressure ratio and mean isentropic efficiency) is marginal. This indicates the absence of a systematic error mainly due to lack of bias in the assumed uniform probability distribution for the input geometric parameters. For further confirmation, the optimized impellers have been parameterized and simulated in CFD using the pre-processing and solution setup described in chapter 4. Table 6.2 summarizes the results. Nominal results for impeller pressure ratio and isentropic efficiency computed by the metamodels show a

<table>
<thead>
<tr>
<th>Measure</th>
<th>Impeller A</th>
<th>Impeller B</th>
<th>Impeller C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure ratio, $\Pi$ [-]</td>
<td>2.63</td>
<td>2.58</td>
<td>2.51</td>
</tr>
<tr>
<td>Percentage difference*, $\Delta \Pi$ [%]</td>
<td>-1.13</td>
<td>-0.38</td>
<td>+2.03</td>
</tr>
<tr>
<td>Isentropic efficiency, $\eta$ [%]</td>
<td>81.68</td>
<td>83.47</td>
<td>79.86</td>
</tr>
<tr>
<td>Percentage difference*, $\Delta \eta$ [%]</td>
<td>-0.10</td>
<td>+0.42</td>
<td>-1.04</td>
</tr>
</tbody>
</table>

* percentage difference from the nominal metamodel results in Table 6.1
good agreement with the CFD results for all the optimized impeller designs. This validates the accuracy of performance predictions made by the metamodel for new impeller designs.

6.5 Conclusions and Recommendation

The following conclusions can be drawn from the robust design optimization of the turbocharger compressor impeller:

- Lower performance variation is obtained by reducing the inducer tip-clearance $t_{clr,1}$, tip radius $r_2$ and blade thickness $t_b$, while increasing the exducer tip-clearance $t_{clr,2}$. Exducer tip-clearance $t_{clr,2}$ is the most sensitive and uncertain geometric parameter and has a large influence in the optimization process.

- Structural constraints are not considered (but they are known) and, therefore, not included in the current optimization. These constraints can be critical factors for overall design of the impellers.

- Robust design optimization for the test case impeller has been carried out at a single best-efficiency operating point for a fixed rotational speed. For a turbocharger application, the compressor operating range, represented by surge and choke margins, are critical performance parameters. It is important to consider the off-design operation of the turbocharger as well during the robust design optimization process.

- All the optimized impeller designs deliver different levels of robustness. Selection of the most suitable design is an open question depending on the compromise a designer can make between the mean performance and the variability in performance that can affect the quality of manufacturing eventually.

References


REFERENCES


7

Experimental Validation

7.1 Preface

An opportunity was provided by Mitsubishi Turbocharger and Engine Europe BV (MTEE) to validate the computational uncertainty quantification (UQ) through comprehensive experimentation at their turbocharger testing facility. This chapter presents the experimental validation\(^1\) carried out at MTEE in detail. A sample of impellers has been manufactured according to a full-factorial design of experiment (DoE) method. The impellers have been sequentially installed on the turbocharger and tested on the MTEE turbocharger test bench. The results have been collected and used to construct a response surface model, which serves as an experimental metamodel to conduct the Monte Carlo simulation (MCS).

7.2 Methodology and Setup

The procedure initiates with the selection of a suitable set of geometric parameters and the DoE method for the impeller. After manufacturing according to the DoE assigned dimensions, the impellers have been tested on a turbocharger test bench. Different steps involved in setting up the experimental DoE are explained in the following sections of the chapter in detail.

7.2.1 DoE for the Test Case Impeller

Compared to a computational DoE, an experimental DoE is expensive to build and execute. Hence, it is favorable to conduct a minimum number of experiments

\(^1\)The experimental work at MTEE is part of an internship assignment carried out by a Master of Science student, Tarik Hartuc from Delft University of Technology (TU Delft). The contents of this chapter have been permitted to publish in the public domain by MTEE.
to save valuable time and costs. However, the number of experiments increases with the number of input design parameters involved in a DoE. Consequently, three impeller geometric parameters have been considered. These include the most sensitive and uncertain geometric parameters, i.e., the inducer tip-clearance $t_{\text{clr,1}}$, exducer tip-clearance $t_{\text{clr,2}}$ and the tip radius $r_2$. Manufacturing of these parameters is straightforward, since a turning operation in a lathe machine is mainly required on the impeller castings to obtain the required trims. The blade thickness $t_b$, being the least sensitive amongst the geometric parameters and also infeasible to manufacture, since it is a casting property (unless the impellers are machined completely from a forged billet), has been excluded from the study. The remaining geometric parameters are unchanged and maintained at their nominal values.

The $3^k$ full-factorial design [1], consisting of all the combinations of $k$ number of design variables having three levels each, has been used for the experimental DoE. With three geometric parameters considered for DoE, this corresponds to $3^3 = 27$ unique design combinations, where a third level facilitates the evaluation of a quadratic input-output relationship. Table 7.1 shows the selected deviation range for the three geometric parameters. The lower, nominal and upper values of the selected geometric parameters correspond to $-1$, $0$ and $+1$ levels, respectively. The repeatability or stability of the experiments has to be determined to quantify the measurement uncertainties. For a $3^k$ factorial design, this is accomplished by using the center points (also known as control runs), which are dedicated samples tested multiple times at regular intervals during experimentation. For the experimental DoE, a total of five center point measurements have been carried out using a single impeller having the nominal geometry. All in all, a total of 32 tests have been made for the experimental DoE including the five center point measurements.

The steady-state tip-clearances$^2$ at the inducer and the exducer sections are expected to reduce up to 6% of exducer height $b_2$ (section 3.6.1 of chapter 3) during turbocharger operation at high rotational speeds, thereby complying with the operating tip-clearance values used in the computational DoE. Therefore, the selected deviation has been introduced in the steady-state tip-clearances for the impeller and machined accordingly.

---

$^2$The information on the nominal steady-state tip-clearance values and the respective lower and upper deviations cannot be disclosed due to confidentiality.
7.2. METHODOLOGY AND SETUP

(a) Raw castings
(b) Backface machining
(c) Final machining
(d) Measurements

Figure 7.1: Manufacturing of the DoE impellers for experimentation

7.2.2 Manufacturing of the DoE Impellers

The manufacturing process of the DoE impellers consists of three main stages. First the impellers have been acquired as raw castings as shown in Fig. 7.1a. The impeller casting process has been elaborated in chapter 2. In the second and third stages, the castings undergo turning operations in the computer numerical control (CNC) lathe machine. The impellers are clamped one at a time by the boss (protrusion in front of the impeller casting) and their backface is machined, followed by an initial machining of the exit diameter and machining of the shaft hole (or bore) of the impeller. Figure 7.1b shows the impellers after the second stage machining operation. The third stage machining operation forms the shroud profile of the impeller blades. The impellers are clamped by their disks during this operation. The boss is machined off at the end of this operation and the finished impellers, as shown in Fig. 7.1c, are obtained. Due to clamp size limitation, the exit diameter could not be trimmed on the CNC lathe machine. Therefore, the machining has been carried out separately on a conventional lathe machine. All the DoE impellers underwent a thorough inspection, as shown in Fig. 7.1d, through different measurements in order to assure precise manufacturing, and also to record any manufacturing deviation.

Prior to machining, the selected impeller geometric parameters have to be converted to a set of machine-readable parameters for the CNC lathe machine.
Figure 7.2 shows the typical machining parameters for the impeller, which include the inlet diameter $M$, exit diameter $Q$, parameter $G$ and the trim radius $R$. Trim radius $R$ is also known as the $R$-shape parameter, since it determines the curvature of the shroud profile during machining. The parameter $G$ sets the datum or the starting point for the CNC lathe machine to trim the $R$-shape. Therefore, it is a combination of various other geometric parameters including the disk thickness $t_d$, trim radius $R$ and the tip height $b_2$ of the impeller. The tip height $b_2$ is connected to the exducer tip-clearance $t_{clr,2}$ so that any manufacturing deviation in $b_2$ will ultimately cause a variation in $t_{clr,2}$ for a constant compressor cover shroud. The inlet diameter $M$ forms the inlet tip radius $r_{1t}$ (where $M = 2 \times r_{1t}$), as illustrated in Fig. 7.2. Therefore, any manufacturing deviation in $M$ will affect the inducer tip-clearance $t_{clr,1}$ since $r_{1t}$ and $t_{clr,1}$ are coupled for a constant compressor cover shroud. Conversion of the impeller tip radius $r_2$ forms the exit diameter $Q$ (where $Q = 2 \times r_2$), as shown in Fig. 7.2. A deviation in $Q$ during manufacturing will result in a variation in $r_2$. In addition, a deviation in $Q$ also affects $b_2$ accordingly, changing along the impeller shroud profile inclined at 12.5°.

7.2.3 Experimental Setup and Testing

The impellers have been tested on the MTEE test bench, which is an advanced experimental setup for measuring turbocharger performance. Figure 7.3 shows the test bench and its different components, including a turbocharger. A natural gas
burner is used to heat the compressed air, which is then supplied to the turbine through the turbine inlet pipe. At low power requirement, the compressed air can also be heated in the electric heater installed in parallel to the burner. The gas is expanded in the turbine and exhausted out of the test cell through the turbine outlet pipe leading to the chimney.

A throttle valve located downstream of the compressor is used to regulate the mass flow during performance map measurement. Pre-heated oil at 100°C is pumped in the bearing housing of the cartridge. The difference in oil temperature measured at the inlet and outlet stations of the bearing housing provides a conservative estimate of the mechanical loss in the turbocharger. Necessary insulation is used on the compressor inlet and outlet piping, along with the sensors to minimize heat transfer from the hot turbine side.

Flow properties are measured at inlet and outlet stations of the turbocharger compressor, turbine and bearing housing with total temperature thermocouple probes and static pressure sensors. Multiple sensors are used for each measurement and average values are calculated. The data is then converted to static temperatures and total pressures using measurements from the thermal mass flow meters installed upstream of the compressor and the turbine.

A single cartridge, which forms the middle section of the turbocharger and houses the shaft and the bearings, has been used for all the tests. The 32 DoE impellers are installed one by one for each test on the same cartridge. During production, the cartridge assembly comprising of the compressor and turbine impellers connected on a single shaft passing through the journal bearings, is bal-
anced by machining off some mass from the nut holding the compressor impeller. Doing so prevents any irregular rotordynamic behavior in the long run due to coupling of two rotating masses, i.e., the compressor and turbine impellers. For the DoE study, preparation of separate cartridge assemblies for each impeller was not feasible cost-wise, including the time constraints set by the busy test schedule at MTEE. Therefore, the results from unbalanced cartridge assembly have been accepted.

All the tests have been performed at a mass flow rate of 60g/s at 220,000rpm rotational speed, corresponding to the best-efficiency operating point simulated for the computational evaluation of manufacturing uncertainties. The tests have been corrected to ambient temperature and pressure of 293.15K and 101.325kPa, respectively. The TIT has been maintained approximately at 450K to minimize the heat transfer effects on compressor performance. Sufficient stabilization time is given before each measurement in order to minimize the test bench measurement uncertainties. The data is recorded at a preset frequency (measurement repetitions), which is then averaged and used in the test bench calculation sequence to obtain the final measured outputs in form of overall compressor pressure ratio and isentropic efficiency.

7.3 Results and Discussions

The experimental DoE responses have been used to construct the metamodels, which have been evaluated for fit and convergence, and subsequently used to conduct the compressor performance assessment under manufacturing uncertainties in the impeller geometry. A comparison has been made between the computational and experimental evaluations in the end.

7.3.1 Metamodel Construction and Evaluation

The experimental DoE responses for the compressor pressure ratio and isentropic efficiency have been fitted with second-order or quadratic response surfaces in Matlab, which act as metamodels to predict the compressor performance at new observations. To check the usefulness of the metamodels in predicting the correct responses, the coefficient of determination $R^2$ and its adjusted form $R^2_{adj}$ have

<table>
<thead>
<tr>
<th>Measure</th>
<th>Responses</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R^2_{32}$</td>
<td>0.987</td>
</tr>
<tr>
<td>$R^2_{adj,32}$</td>
<td>0.985</td>
</tr>
</tbody>
</table>

Table 7.2: Experimental response surface model diagnostics
7.3. RESULTS AND DISCUSSIONS

Figure 7.4: Comparison between the measured DoE responses and the experimental response surface model outcomes for the test case compressor (For confidentiality, $t_{clr,1}$ and $t_{clr,2}$ are represented as -1, 0 and +1 levels of factorial design)
been calculated. Table 7.2 presents the two coefficients of determination for the compressor pressure ratio and isentropic efficiency. The response surface models show a good fit, since $R^2$ and $R^2_{adj}$ lie in a satisfactory range of 0.9-1.0. However, the coefficients of determination obtained for the response surface model representing the compressor isentropic efficiency are considerably lower than the ones for response surface model representing the compressor pressure ratio.

To evaluate the metamodel convergence, a direct comparison has first been made between the measured DoE responses and the corresponding predictions made by the response surface models for the lower, nominal and upper values of impeller geometry given in Table 7.1. Figure 7.4a shows a comparison between the test and response surface outcomes for inducer tip-clearance $t_{clr,1}$. A reasonably good agreement can be seen between the test and the metamodel responses for both compressor pressure ratio and isentropic efficiency. A similar trend is seen in Fig. 7.4b for the exducer tip-clearance $t_{clr,2}$. The predictions made by the metamodels for tip-radius $r_2$, as displayed in Fig. 7.4c, show a relatively large disagreement with the test data, especially for the compressor isentropic efficiency.

The inconsistencies in the results predicted by the experimental metamodels can be associated to the measurement uncertainties of the test bench propagated in the DoE responses, and the lack of sufficient number of samples to fill the design space and represent the mutual interaction between the input geometric parameters. Furthermore, the measurement of compressor isentropic efficiency can be more uncertain than measuring the compressor pressure ratio due to many uncertain measured parameters involved in its calculation.

As a second convergence evaluation, the experimental metamodel results for compressor performance with deviations in the impeller geometry have been compared with the corresponding computational metamodel outcomes. However, before the evaluation, it is important to highlight a couple of aspects related to the differences found in the computational and experimental evaluations:

- **Nominal tip-clearances.** As mentioned in section 7.2.1, it was expected that the tip-clearance will reduce to 0.15mm (approximately 6% of tip height $b_2$) during compressor operation at high rotational speeds from the nominal steady-state tip-clearance value. However, the variation in $t_{clr,1}$ and $t_{clr,2}$ was found to be marginal during the tests, thus invalidating the nominal tip-clearance values used in the computational evaluations. To resolve this contradiction, the experimental metamodels have been used to extrapolate the compressor performance at lower values of inducer tip-clearance $t_{clr,1}$ and exducer tip-clearance $t_{clr,2}$, corresponding to the nominal operating tip-clearance of 0.15mm used in the computational evaluations. Although the extrapolation of new observations outside the design space, for which the response surface has not been constructed, will be questionable, it may still be worthwhile to do so considering the amount of effort that will be required to perform either of the computational and experimental evaluations all over again with correct nominal tip-clearances.
7.3. RESULTS AND DISCUSSIONS

Figure 7.5: Performance sensitivity evaluation and comparison between the CFD, computational metamodel and the experimental metamodel results (the term ‘RSM’ represents a response surface model)
• **Measurement locations.** The measurement locations are also different in the computational and experimental setups. In the computational evaluations, only the impeller performance has been considered and measured. This is contrary to the experimental evaluations, where the performance of the complete compressor; from compressor inlet to volute outlet, has been measured on the test bench. To account for the difference in measurement locations, the relative change in impeller and compressor performance, predicted by the computational and experimental response surface models, respectively, has been considered.

Figure 7.5 shows the analysis made by deviating the selected impeller geometric parameters inside the range defined in Table 7.1. The CFD data obtained from the evaluation presented in section 4.6.2 of chapter 4 has been added in the plots as an additional reference. Considering the change in performance caused by the inducer tip-clearance \( t_{\text{clr},1} \) shown in Fig. 7.5a, a good agreement is observed between the three data sets for pressure ratio; contrary to the isentropic efficiency. The pressure ratio and isentropic efficiency variation due to deviations in the exducer tip-clearance \( t_{\text{clr},2} \), shown in Fig. 7.5b has been predicted well by the experimental metamodels. For the deviations in tip radius \( r_2 \), the pressure ratio predictions made by the experimental response surface model are in a reasonable agreement with the other two data sets, whereas an opposite trend is observed for the isentropic efficiency. After an evaluation of the impeller geometry, it has been observed that this contradiction is caused by the wrap angle and the associated backsweep angle \( \beta_{2b} \), which also changes with deviations in \( r_2 \). For instance, as \( r_2 \) is reduced, the wrap angle decreases, thereby resulting in a lesser \( \beta_{2b} \), and vice versa. Contrary to this, the computational evaluations have been made at a constant \( \beta_{2b} \) for varying \( r_2 \). However, this disagreement has been caused by the lack of sufficient information related to compressor geometry and its different features, whereas the methodology behind the computational evaluations is correct.

On the whole, although being extrapolated, predictions made by the experimental response surfaces are in a reasonable agreement with the CFD and computational response surface data, especially in the case of pressure ratio. The comparison, therefore, has provided a decent validation to the computational methodology and the results obtained in this work for the test case impeller.

### 7.3.2 Uncertainty Quantification

The experimental metamodel based MCS has been applied for uncertainty propagation. A total of hundred thousand uniformly distributed samples have been generated in Matlab inside the deviation ranges (now considered as tolerances) given in Table 7.1 for the three geometric parameters; inducer tip-clearance \( t_{\text{clr},1} \), exducer tip-clearance \( t_{\text{clr},2} \) and tip radius \( r_2 \). The Monte Carlo samples have been evaluated deterministically with the experimental metamodels. The output performance variation has been statistically evaluated to determine the mean and
7.3. RESULTS AND DISCUSSIONS

(a) Impeller performance variation predicted by the computational metamodel

(b) Compressor performance variation predicted by the experimental metamodel

Figure 7.6: Performance variation under manufacturing uncertainties predicted by metamodel assisted MCS

the standard deviation of compressor performance due to the manufacturing uncertainties introduced in the impeller geometry. For comparison, the computational metamodel based MCS has also been performed using the three impeller geometric parameters and their respective tolerances given in Table 7.1.

Figure 7.6 presents the probability distributions, along with the statistical data produced by the computational and experimental metamodel based MCS. As expected, the pressure ratio variation is closely predicted by the computational and experimental response surface models, where both are having comparable probability distributions. The mean values for pressure ratio are similar in the two
evaluations since the experimental response surface has used to extrapolate the compressor performance at lower nominal tip-clearance values (see section 7.3.1), thereby resulting in a higher mean pressure ratio. Nonetheless, their standard deviations, which represent the variability in pressure ratio due to manufacturing uncertainties, are similar. The computational metamodel based MCS results are, therefore, being validated by the experimental counterpart. In comparison, predictions made by the computational metamodel based MCS for isentropic efficiency do not correspond well to the experimental outcomes as expected. The statistical data, along with the probability distributions for the computational and experimental isentropic efficiency evaluations are in a disagreement.

7.4 Conclusions and Recommendations

The following conclusions can be drawn from the experimental validation study:

- Many discrepancies have been found in the impeller geometries used for the computational and experimental evaluations, including the differences in nominal tip-clearances at steady-state and operating conditions. These differences have been translated into the disagreements in the metamodel assisted computational and experimental analysis of the test case impeller. Therefore, obtaining a high quality geometric model is obligatory.

- The differences in DoE methods, number of DoE samples and the measurement uncertainties in the test bench are also contributing to the overall inconsistency found between the computational and experimental evaluations.

- The experimental outcomes, although being extrapolated to correct for the differences in steady-state and operating tip-clearances, have given a reasonable validation to the computational methods applied for manufacturing uncertainty quantification of the test case impeller. The variability in compressor pressure ratio has been predicted reasonably well during the process.

References

General Conclusions

Automotive turbochargers are being produced in millions every year with their demand on the rise to compensate for the performance trade-off in the internal combustion (IC) engines as they are downsized in order to meet the stringent emission regulations. Although being a small machine having few components, the turbocharger is a critical subsystem, which determines the final output performance of a downsized IC engine. A high functional quality, i.e., top performance at design and off-design operating conditions is required from an automotive turbocharger in a tough working environment close to the hot sections of the IC engine with large magnitudes of heat transfer. Furthermore, precision manufacturing and high quality control is required to satisfy the miniature dimensions and tolerances set to control the manufacturing uncertainties and the resulting dimensional deviations, inherent to every manufacturing process. Being low Reynolds number turbomachines, turbochargers are susceptible to losing a great deal of performance with small deviations in component geometry, including the variation in operating conditions as well.

A unique application of an automotive turbocharger has been made in the form of a micro gas turbine or microturbine for combined heat and power (CHP) generation, designed by a Dutch company named Micro Turbine Technology BV (MTT). The turbocharger has been integrated with a generator, combustor and a recuperator, along with other subsystems, thereby forming the complete microturbine/CHP assembly. Preliminary design and development of a microturbine using an automotive turbocharger has been a successful attempt. However, considering the difference in applications, MTT would require an indigenous design especially for the microturbine application. Moreover, considering the market projections for the microturbine, it is likely that the microturbine will be produced in large numbers. A high quality product is, therefore, necessary in this case.

The robust design methodology considers both the functionality and manufacturability of a product during the initial phases of its design. The methodology is particularly advantageous for complex designs, such as turbomachinery, in order
to achieve a product, which delivers a high level of performance, along with an insensitivity to stochastic or random dimensional deviations caused by the manufacturing uncertainties. A generic methodology has been presented in the thesis to implement the robust design methodology on a micro centrifugal compressor impeller belonging to an automotive turbocharger using a versatile selection of engineering models, which have been properly assessed, adapted with additional submodels, integrated and applied as a unit to eventually perform the manufacturing uncertainty quantification (UQ) and robust design optimization.

In line with the objectives of the present research, following conclusions can be drawn from the complete set of evaluations presented in this dissertation for the test case micro centrifugal compressor impeller:

- **Centrifugal compressor performance modeling.** Performance modeling of centrifugal compressors is a difficult task due to the presence of complex three-dimensional (3D) flow field in its different components. A main contributor to the loss in compressor efficiency is the impeller due to the presence of low-momentum boundary layers and leakage flow, composing a distinctive secondary flow zone. A 1D meanline model based on the two-zone methodology is logically applied considering the fact that there are indeed two distinct zones of flow in the impeller passage, which eventually undergo a rapid mixing downstream of the impeller. A provision has been made in the standard two-zone model by decomposing the entropy production in the secondary zone into its sources, by using the empirical loss correlations generally used in the single-zone meanline modeling method. The 1D meanline model underwent many adaptations in its empirical modules using the data obtained from a comprehensive 3D performance analysis of the test case compressor carried out using computational fluid dynamics (CFD).

- **Metamodeling of centrifugal compressor performance.** The 1D meanline model was quick in calculating the centrifugal compressor performance. However, the lack in accuracy to predict the performance variation as a result of deviations in the impeller geometry, in comparison to the high fidelity CFD modeling, resulted in abolishing the 1D meanline model from further use in the research. A large database comprising of both simulated and experimental data sets, is needed to effectively advance the 1D meanline model for improving its accuracy in predicting the centrifugal compressor performance. Instead, the metamodel has proved to be a fine alternative to the 1D meanline model. An initial set of response data has been prepared by carrying out a design of experiments (DoE) using the space-filling Latin hypercube sampling (LHS) in CFD. Response surface has been a very useful metamodel for its simplicity and for its rate of convergence, using a relatively lesser number of DoE samples compared to the kriging metamodel, also considered in the study. The metamodels have been validated against the CFD data, and also the experimental data in later part of the thesis, as a confirmation to their accuracy of prediction.
• **Probabilistic analysis of manufacturing uncertainties.** Requirement for a quick performance analysis tool, such as the 1D meanline model and the metamodel, for the micro centrifugal compressor was set by the intended probabilistic analysis to be carried out using the Monte Carlo simulation (MCS). High fidelity and deterministic performance calculations for a large number of Monte Carlo samples (a hundred thousand samples used in the present study) would be impractical to conduct with 3D CFD modeling. After discarding the 1D meanline model, a metamodel based MCS has been performed to propagate the influence of random geometric deviations in the impeller geometry on its performance. The most sensitive of the impeller geometric parameters have been used as input variables to save further computational time. Later, the analysis of variance (ANOVA) identified the most uncertain of these geometric parameters. Finally, the most sensitive and the most uncertain of the impeller geometric parameters have been successfully recognized through the probabilistic evaluation.

• **Robust design optimization.** The metamodel assisted MCS setup has been integrated with a multi-objective genetic algorithm (MOGA) to conduct the maximization of test case impeller’s mean performance, while minimizing the performance variation due to geometric deviations under an uncertain manufacturing environment. The most sensitive and uncertain impeller geometric parameters have been used to carry out the robust design optimization, which were identified during UQ. The final selection of a suitably robust impeller out of different Pareto optimal solutions is an open question and is largely based on how much trade-off can be made between performance and robustness, since they are conflicting each other. Nonetheless, trimming of the impeller shroud profile has been realized to be a sensitive operation, which can ultimately influence the tip-clearance gap between the impeller and the compressor cover, thereby causing a considerable variability in performance if deviated from its nominal value during manufacturing. A robust definition of the impeller shroud profile, as demonstrated in the thesis, can reduce its sensitivity to geometric deviations caused by the intrinsic manufacturing uncertainties. This also means that during the turbocharger operation, the compressor may be insensitive to any rotor-dynamics related deviations in the clearance gap, thus delivering the target performance consistently.

**Recommendations for Future Work**

Probabilistic turbomachinery analysis and design are a relatively new and unconventional area of research. Nonetheless, the Dutch gas turbine industry, being manufacturers of high quality gas turbine components, can benefit in developing their design expertise and assessing the influence of manufacturing deviations on
turbomachinery performance using the methods and tools presented in this thesis. Some open items have been left in the study, which could be evaluated and implemented in future research activities in this field. Following are the recommendations deduced from the present work:

- The first and foremost attention should be paid to the quality of the geometric model and the amount of information available (part drawings, tolerances, test data, etc.) available for the turbomachinery case under evaluation. Lack of relevant geometric data, including information related to steady-state and operating tip-clearances for the test case micro centrifugal compressor, has led to many discrepancies in modeling its performance, as revealed by the comprehensive experimental evaluation of manufacturing uncertainties presented in the thesis. Therefore, as a good start, it is important to collect as much information as possible about the test case geometry in order to carry out a high quality analysis.

- 1D meanline modeling for predicting centrifugal compressor performance has been found impractical. A great deal of research has been made on the tool in the past decades without producing a definite modeling approach that is universally acceptable in the engineering community. The 1D models have undergone considerable adaptations and as demonstrated in the thesis, the present 1D model had to be tuned further to improve its accuracy of prediction for the test case turbocharger compressor, especially to implement the geometric deviations for UQ and subsequent analyses.

- Data assimilation is important for determining the probability distribution function (PDF) of an input variable in order to model the influence of manufacturing uncertainties on performance accurately. Such data was not available in the present case for the test case compressor. For more advanced probabilistic analysis, it is highly recommended to acquire the PDFs of the input variables from the manufacturer and implement them in the UQ.

- A single operating point has been used for robust design optimization of the test case impeller. As a result, a robust impeller may have different surge and choke margins, which can be critical for an automotive turbocharger application. Hence, robust design optimization should also consider compressor performance at off-design operating points.

- The structural aspect of design is missing in the present evaluation. Multi-disciplinary optimization, where both aerodynamic and structural features of turbomachinery could be considered simultaneously, will be highly advantageous if included in a future research work. Furthermore, the influence of surface roughnesses on compressor performance has been neglected in the present study and may be considered as well, thus adding more precision and value to the work.
Acknowledgements

During the course of five years in the Netherlands for my PhD, I have made many professional and personal achievements, gained a lot of knowledge, met many people, visited many places, and married a lovely woman. Sadly, I also lost three of the most important persons in my life — my grandfather, who was my mentor, my grandmother, who was my dear friend, and my mother, who sacrificed so much for the success of her children. Nonetheless, life goes on and in the words of John Lennon, life is what happens while you are busy making other plans.

First of all, I want to express my gratitude to my promotor, Jos van Buijtenen for selecting me for the PhD position and allowing me to study at TU Delft, which is one of the most prestigious engineering universities in the world. I would like to acknowledge the freedom and encouragement he gave me during the research. Thank you very much for your guidance and support at both professional and personal levels.

Special thanks goes to Rene Pecnik who was a true and critical adviser in the course of this research. His supervision proved to be the guardian angel which always showed me the direction when I needed it the most. Thank you very much for your support throughout my study.

I would like to thank the team from Micro Turbine Technology BV (MTT) for their support, particularly Wilfried Visser, Savad Shakariyants and Nicholas George for the useful discussions.

I wish to thank my colleagues at Mitsubishi Turbocharger and Engine Europe BV (MTEE) as well. I am grateful to Edwin Kamphues, my manager, for his support, and for allowing me to write my thesis on Fridays. Many thanks to Rogier Lammers and Berend Jonkers for their continuous push to finish my PhD. I still remember Rogier saying “finish your PhD Adeel!” and here I am now, with my PhD degree. Thank you, Rogier and Berend, for translating the summary and the prepositions in my thesis to Dutch.

I found a great friend in Mattia Olivero during my PhD. Together, we did good research, laughed so much talking about the film Snatch and South Park cartoons, played Need for Speed, had biryani, talked about cricket, etc. Thank you for your friendship and support throughout these years and hopefully, I will host you in Pakistan in the near future and show you my country and culture.

My appreciation is extended to my other office-mates – Michel, Sowande, Teo and Bart. From Michel, I learnt management skills and Matlab (thanks for the
help Michel!). From Sowande, I learnt how to be cool as a cucumber in difficult situations. Teo taught me how to do quality work and sleep at the same time, while Bart showed me that scientists can also be theatre artists. Guys, thank you very much for the good time!

Let me also acknowledge the support of Process and Energy department’s staff, especially Eveline van der Veer and Judith Blommaart, fellow PhD and post doc students, my MSc students, Sin-Yun Yang and Tarik Hartuc, and the technical staff in the laboratory.

I joined the Concordia DSV cricket club in Delft back in 2008 and since then, enjoyed playing cricket with a bunch of lovely people. Even though I moved to Almere in 2012, I just could not leave the club and continued playing for it for the good company and fun. Jan Willem, Manu, Nauman, Deep, Patel, Rupert, Kamal and rest of the gang, thank you for the nice companionship and some good cricketing moments.

I came to know many Pakistani students while studying at TU Delft, many of whom are now good friends. Thank you Omer Naeem, Faisal Nadeem, Seyab Khan, Umer Altaf, Fakhar Anjam, Muhammad Akram, Abdul Hanan, Usama Malik, Shah Muhammad, Shahriyar Khan, Zubair Khan, Umair Ismail, Zubair Nawaz, Tariq Abdullah, Aqel Wahlah, and many others, for the good company. Special thanks to Sana Aamir, Aisha Qadir, Maham Aftab, Asma Qadir, Yasir Rafiq, Aqel Ali, Ali Waqar and Umer Ijaz Chaudhry, for all the good fun over these years.

I have my deepest regards for my wife and I wish to thank her for being a part of my life. She has been a great support in good and bad times. Saba, thank you very much for your patience and support, especially for taking care of the family during my mother’s illness and passing away, and while I was writing the thesis. I could not have done it without you!

Last, but not the least, I would like to thank my parents for making it all possible. Ami, I will always love you and I am sorry for being away from you all these years. I will miss you dearly at my defense ceremony, but I know in my heart that you would have been a proud mother. Thank you papa, for your continuous support. You have been my inspiration in life for your hard work and selfless dedication towards your family. My dearest ami and papa, you are my heroes!

Adeel Javed
Almere, May 30, 2014
About the Author

Adeel Javed was born on January 24, 1984 in Rawalpindi, Pakistan. After completing his higher secondary school education in 2001, he joined the College of Aeronautical Engineering (CAE), an esteemed aviation institute of the National University of Science and Technology (NUST) in Pakistan. He completed his BE Aerospace degree in 2005 and in 2006, proceeded to Cranfield University, UK for MSc in Thermal Power. He obtained the MSc degree and a specialization in aerospace propulsion in 2007 with distinction. In 2008, he was awarded a PhD position at Delft University of Technology. His research focussed on the performance and manufacturing aspects of gas turbine parts in order to support concurrent engineering and design for robustness against manufacturing uncertainties.

Adeel has a keen interest in topics related to gas turbine performance, turbo-machinery design, computational fluid dynamics (CFD), aircraft performance and design, experimentation, design optimization and uncertainty management. He has published several articles in various areas of gas turbine engineering and has mentored many BE and MSc students.

In his free time, Adeel enjoys playing cricket, reading books and articles, building scale models, listening to music, watching movies, travelling and sight seeing, cooking, etc. Moreover, he is an enthusiast of aircrafts and cars and loves to collect information about their history and technological development.
Selected Publications


