

**T.C.
GEBZE TECHNICAL UNIVERSITY
GRADUATE SCHOOL**

**AERODYNAMIC DESIGN AND ANALYSIS OF TURBINE WITH
MID FRAME**



EREN IŞIK

**A THESIS OF MASTER OF SCIENCE
DEPARTMENT OF MECHANICAL ENGINEERING**

ADVISOR: ASSIST. PROF. DR. SALİH ÖZEN ÜNVERDİ

FEBRUARY 2026

**T.C.
GEBZE TECHNICAL UNIVERSITY
GRADUATE SCHOOL**

**AERODYNAMIC DESIGN AND ANALYSIS OF
TURBINE WITH MID FRAME**

EREN IŞIK

**A THESIS OF MASTER OF SCIENCE
DEPARTMENT OF MECHANICAL ENGINEERING**

ADVISOR: ASSIST. PROF. DR. SALİH ÖZEN ÜNVERDİ

FEBRUARY 2026

T.C.
GEBZE TEKNİK ÜNİVERSİTESİ
LİSANSÜSTÜ EĞİTİM ENSTİTÜSÜ

TÜRBİN ORTA ÇERÇEVELERİNİN AERODİNAMİK
TASARIMI VE ANALİZİ

EREN IŞIK

YÜKSEK LİSANS TEZİ
MAKİNE MÜHENDİSLİĞİ ANABİLİM DALI

DANIŞMAN: DR. ÖĞR. ÜYESİ SALİH ÖZEN ÜNVERDİ

ŞUBAT 2026

MASTER of SCIENCE JURY APPROVAL FORM

A thesis submitted by Eren Işık, defended on 02/02/2026 before the jury formed with the 20/01/2026 date and 2026/05 numbered decision of the GTU Graduate Administration Board, has been accepted as a MASTER of SCIENCE thesis in the Department of Mechanical Engineering.

JURY

MEMBER

(THESIS ADVISOR) : Assist. Prof. Dr. Salih Özen Ünverdi

MEMBER

: Prof. Dr. Süleyman Karsli

MEMBER

: Assoc. Prof. Dr. Sercan Acarer

APPROVAL

Gebze Technical University Graduate Administration Board
...../...../..... date and/..... numbered decision.

SIGNATURE/SEAL

ABSTRACT

This thesis investigates the aerodynamic characteristics of a turbine mid-frame concept in axial turbines. A reference two-stage axial turbine, specifically the NASA/GE E3 high-pressure turbine, is utilized as a benchmark for this study. The turbine model is reconstructed and validated through blade-to-blade (B2B) computational fluid dynamics (CFD) calculations at the hub, mean, and tip sections, in alignment with available literature data. Subsequently, the model is validated at the design point and selected off-design conditions by comparing the corrected mass flow rates and the turbine's cooled total-to-total efficiency, which, in other words, represents the thermodynamic efficiency. The calculated thermodynamic efficiency at the design point differs by 0.68 percentage points when compared to the experimental data.

Following the validation process, the stator of the second-stage turbine is redesigned using two distinct mid-frame configurations: a single-frame configuration (MF1) and a splitter-equipped frame configuration (MF2S). The geometric properties of these configurations are non-dimensionalized and parameterized for control via pre-conditioned functions, including Bezier point control functions for beta distributions and an asymmetric power law for thickness distributions. The aerodynamic characteristics of these configurations are examined using B2B CFD calculations at the mean section under varying inflow conditions and profile variations as part of a Design of Experiments (DoE) study. The results indicate that the profile loss behavior aligns with the Kacker-Okapuu turbine loss model, exhibiting strong sensitivity to factors such as exit flow angle, pitch-to-axial chord ratio, and the inlet-to-exit Mach number ratio. Additionally, the impact of splitter angle differences, relative to the corresponding main frame angle, on losses is analyzed using a reduced surrogate model.

Finally, the selected MF1 and MF2S configurations are integrated into the benchmark turbine model to compare their aerodynamic performance. Both configurations demonstrate favorable incidence tolerance relative to the baseline model due to their larger leading-edge radius and reduced front-side curvature, although they experience shock losses near the throat. In conclusion, both designs exhibit promising aerodynamic characteristics in the mean section, as indicated by the B2B CFD calculations. The turbine mid-frame concept is identified as a promising approach for shortening the axial length of main engines by replacing the aft frame with a turbine stator row.

Keywords: Axial turbine, E3 High-pressure turbine (HPT), Mid turbine frame, Computational fluid dynamics (CFD), Blade-to-blade (B2B), Design of experiments (DoE)

ÖZET

Bu tezin amacı, eksenel bir türbinde türbin orta gövde taşıyıcı çerçevesi (turbine mid frame) konseptinin aerodinamik karakteristiklerini ortaya koymaktır. Bu kapsamda, NASA/GE E3 yüksek basınç türbini olarak bilinen iki kademeli eksenel bir türbin referans alınarak bir karşılaştırma çalışması gerçekleştirilmiştir. Türbin geometrisi literatürde yer alan veriler doğrultusunda yeniden oluşturulmuş ve kök, orta ve uç kesitlerde kanattan kanata (blade-to-blade) hesaplamalı akışkanlar dinamiği (HAD) yöntemi kullanılarak doğrulanmıştır. Bunu takiben model, tasarım noktası ve seçilmiş tasarım dışı çalışma koşullarında, düzeltilmiş kütleli debi ile türbinin soğutmalı toplamdan-toplama verimi (diğer bir ifadeyle termodinamik verimi) karşılaştırılarak doğrulanmıştır. Tasarım noktasındaki termodinamik verim, deneysel test verilerine kıyasla 0,68 yüzde puanlık bir farkla hesaplanmıştır.

Doğrulama çalışmasının tamamlanmasının ardından, ikinci kademe türbin statoru iki farklı orta çerçeve konfigürasyonu olacak şekilde yeniden tasarlanmıştır: tek ana kanatçıklı MF1 konfigürasyonu ve ana kanatçık ile ayırıcı kanat içeren MF2S konfigürasyonu. Bu konfigürasyonların geometrik özellikleri boyutsuzlaştırılmış ve açı dağılımlarının kontrolü için Bezier noktası tabanlı kontrol fonksiyonları, kalınlık dağılımı için ise asimetrik üstel fonksiyonlar kullanılarak parametreleştirilmiştir. Aerodinamik karakteristikler, deney tasarımı (Design of Experiments, DoE) çalışması kapsamında, farklı giriş koşulları ve farklı profil geometrileri altında orta kesitte gerçekleştirilen kanattan kanata HAD hesaplamaları ile incelenmiştir. Elde edilen sonuçlar, profil kaybı davranışına ilişkin genel eğilimlerin Kacker–Okapuu türbin kayıp modelindeki profil kayıpları ile uyumlu olduğunu ve kayıpların özellikle akış çıkış açısı, kanatlar arası adımın eksenel kanat uzunluğuna oranı ile giriş ve çıkış Mach sayıları oranına güçlü biçimde bağlı olduğunu göstermiştir. Ayrıca, ayırıcı kanadın giriş metal açısı ile ana kanadın karşılık gelen konumdaki metal açısı arasındaki farkın kayıplar üzerindeki etkisi, indirgenmiş bir yaklaşım modeli kullanılarak incelenmiştir.

Son olarak, seçilen MF1 ve MF2S adayları aerodinamik karakteristiklerinin karşılaştırılması amacıyla referans (benchmark) modele entegre edilmiştir. Her iki tasarım da, daha büyük hücum kenarı yarıçapı ve ön yüzey eğriliğinin azaltılması sayesinde referans tasarıma kıyasla daha iyi bir giriş açısı kaçıklığı toleransı sergilemiştir; ancak boğaz bölgesi yakınında şok kaynaklı kayıplar gözlemlenmiştir. Genel olarak, her iki tasarım da orta kesitte gerçekleştirilen kanattan kanata HAD hesaplamalarında umut vadeden aerodinamik performans göstermiştir. Türbin orta gövde taşıyıcı çerçevesi konsepti, arka çerçevenin bir türbin sabit kanat sırası ile değiştirilmesi yoluyla ana motorun eksenel uzunluğunu azaltabilmesi nedeniyle avantajlı bir yaklaşım olarak değerlendirilmektedir.

Anahtar Kelimeler: Eksenel türbin, E3 Yüksek basınç türbini (YBT), Türbin orta çerçevesi, Hesaplamalı akışkanlar dinamiği (HAD), Kanattan kanata (Blade-To-Blade), Deney tasarımı



*To my wife
and family*

ACKNOWLEDGEMENTS

I would like to express my sincere gratitude to my esteemed advisor, Dr. Salih Özen Ünverdi, for his guidance, expertise, and valuable insights throughout my master's studies and academic career. His continuous support and encouragement have been instrumental in the completion of this work.

I am profoundly grateful to my wife and my family, whose unwavering encouragement, patience and understanding have been invaluable throughout this journey. Their support has been a constant source of strength and motivation.

I also extend my deepest appreciation to my colleagues at TRMotor, who generously shared their knowledge and experience during this study and consistently provided their support.

It is my hope that this study will serve as a useful reference for future aerodynamics engineers working in this field and contribute meaningfully to existing literature.

TABLE OF CONTENTS

	<u>Page</u>
ABSTRACT	vi
ÖZET	vii
ACKNOWLEDGEMENTS	viii
TABLE OF CONTENTS	ix
LIST OF SYMBOLS AND ABBREVIATIONS	xi
LIST OF FIGURES	xiv
LIST OF TABLES	xvi
1. INTRODUCTION	1
1.1. Aerodynamic Role of Mid Turbine Frames	1
1.2. NASA GE E3 HPT as Benchmark Configuration	3
2. METHODOLOGY	6
2.1. Theoretical Background on Axial Turbine Flow	6
2.1.1. The H-s (Mollier) Diagram and Expansion Process	7
2.1.2. Kinematics and the Euler Turbine Equation	9
2.1.3. Work Coefficient, Flow Coefficient and Degree of Reaction	13
2.2. Loss Models in Axial Turbines	15
2.2.1. Kacker-Okapuu Loss Model	15
2.2.1.1. Profile Loss	16
2.2.1.2. Secondary Loss	17
2.2.1.3. Trailing-Edge Thickness Loss	19
2.2.1.4. Tip Clearance Loss	20
2.3. Pritchard Eleven-Element Method	20
2.3.1. Model Description	21
2.3.2. Geometric Construction	22
2.4. Computational Fluid Dynamics Fundamentals	22
2.4.1. Governing Equations	23
2.4.2. Turbulence Modeling	24
2.4.2.1. $k-\omega$ SST Model	24
2.4.3. Blade-to-Blade (B2B) Numerical Analysis	25
2.4.4. Quasi-3D Approximation	25
2.4.5. Grid Topology and Near-Wall Resolution	26
2.4.6. Discretization and Solver Settings	26
2.4.7. Boundary Conditions and Convergence	27
3. BASELINE MODEL AND VALIDATION	29
3.1. NASA E3 HPT as Reference Design	29
3.2. Numerical Method	36
3.3. Validation Results	45
4. MID FRAME DESIGN CONCEPT & CONFIGURATIONS	55
4.1. Transition from Stator to Mid Frame	55
4.2. Physical Model and Pre-Setups	60

4.3. Parametric Study (DoE)	68
5. COMPARISON OF RESULTS	81
6. DISCUSSION AND FUTURE WORKS	92
REFERENCES	94
BIOGRAPHY	97
PUBLICATIONS AND PRESENTATIONS FROM THE THESIS	98
APPENDICES	99



LIST OF SYMBOLS AND ABBREVIATIONS

A_1	: Inlet Flow Area
A_{th}	: Throat Area
a	: Speed of Sound
η_b	: Blading Efficiency
C	: Magnitude of Absolute Velocity
C_x	: Axial Absolute Velocity
C_θ	: Tangential Absolute Velocity
c	: Chord
C_{ax}	: Axial Chord
C_l	: Lift Coefficient
C_m	: Meridional Velocity
C_p	: Specific Heat at Constant Pressure
C_t	: Tangential Chord
d	: Deviation Angle
F_1	: SST Blending Function
H_0	: Total Enthalpy
H_s	: Static Enthalpy
h	: Blade Height
i	: Incidence Angle
k	: Turbulent Kinetic Energy
K_1, K_2	: Mach Number Correction Factors
K_3	: K_s Coefficient
K_P	: Mach Number Correction Factor
K_s	: Compressibility Correction Factor
Ma	: Mach Number
Mr	: Relative Mach Number
$\dot{m}$	: Mass Flow Rate
$\dot{m}_{corr}$	: Corrected Mass Flow Rate
N_b	: Blade Count
N	: Rotational Speed
o	: Throat Width
P_0	: Total Pressure
P_s	: Static Pressure
P_{ref}	: Reference Pressure
PWR	: Power
R	: Specific Gas Constant,
$^\circ R$	: Degree of Reaction
r	: Radius
R_{LE}	: Leading-Edge Radius
R_{TE}	: Trailing-Edge Radius
R_H	: Hub Radius
R_T	: Tip Radius
Re	: Reynolds Number
f_{RE}	: Reynolds Number Correction Factor
R^2	: Coefficient of Determination
s	: Entropy

S	: Pitch
T_0	: Total Temperature
T_s	: Static Temperature
$T_{0,c}$	: Cooling Total Temperature
T_{ref}	: Reference Temperature
T_{max}	: Maximum Thickness
u_i, u_j	: Mean Velocity Velocity Components
u_i', u_j'	: Fluctuating Velocity Components
U	: Rotational Velocity
W	: Relative Velocity Magnitude
W_x	: Axial Relative Velocity
W_θ	: Tangential Relative Velocity
y	: First Cell Distance
y^+	: Y Plus
Y_P	: Pressure Loss Coefficient
Y_S	: Secondary Loss Coefficient
Y_T	: Total Loss Coefficient
Y_{TC}	: Tip Clearance Loss Coefficient
Y_{TET}	: Trailing-Edge Thickness Loss Coefficient
Y_{shock}	: Shock Loss Term
$Y_{P,AMD}$	: Ainley-Mathieson Reference Profile Loss
$Y_{S,AMD}$	: Ainley-Mathieson Reference Secondary Loss
α	: Absolute Flow Angle
β	: Relative Flow Angle
β_k	: Blade Metal Angle
$\Delta\beta$	: Blade Turning Angle
$\Delta\beta_u$	: Unguided Turning Angle
$\beta_{1s,offset}$	: Splitter Inlet Angle Offset
Δk	: Tip Clearance
Δ	: Delta
γ	: Specific Heat Ratio
η_{tt}	: Total-to-total Efficiency
η_{TH}	: Thermodynamic Efficiency for Cooled Turbine
$\Delta\eta/\eta_0$	: Efficiency Loss Due to Tip Clearance
Ω	: Angular Velocity
ϕ	: Flow Coefficient
ψ	: Work Coefficient
ω	: Wedge Angle
ρ	: Density
τ_w	: Wall Shear Stress
μ	: Dynamic Viscosity
ξ	: Stagger Angle
1D	: One-Dimensional
2D	: Two-Dimensional
3D	: Three-Dimensional
AR	: Aspect Ratio
AMD	: Ainley-Mathieson Reference Loss
AUSM	: Advection Upstream Splitting Method
AVDR	: Axial Velocity Density Ratio

B2B	: Blade-to-Blade
CFD	: Computational Fluid Dynamics
CTI	: Combustor Turbine Interaction
DNS	: Direct Numerical Simulation
DoE	: Design of Experiment
E3	: Energy Efficient Engine, EEE
EDP	: Engine Design Point
GE	: General Electric
HPT	: High Pressure Turbine
LE	: Leading-Edge
LES	: Large Eddy Simulation
LHS	: Latin Hypercubic Sampling
LPT	: Low Pressure Turbine
MF1	: Mid Frame Configuration 1
MF2S	: Mid Frame Configuration 2 with Splitter
MTF	: Mid Turbine Frame
NASA	: National Aeronautics and Space Administration
PR	: Pressure Ratio
PS	: Pressure Side
RANS	: Reynolds Averaged Navier Stokes
RDG	: Reading
RPM	: Revolutions Per Minute
RMSE	: Root Mean Square Error
R1	: Rotor 1
R2	: Rotor 2
S1	: Stator 1
S2	: Stator 2
SS	: Suction Side
SST	: Shear Stress Transport
TE	: Trailing-Edge
TMTF	: Turning Mid Turbine Frame

LIST OF FIGURES

	<u>Page</u>
Figure 1.1: NASA/GE Energy Efficient Engine [12].	3
Figure 2.1: H-s diagram of axial turbine [1].	8
Figure 2.2: Velocity diagram of axial turbine [1].	11
Figure 2.3: Axial turbine flow [20].	12
Figure 2.4: Smith Chart [23].	14
Figure 2.5: Profile loss coefficient for exit angles and S/C_{ax} ratio [27].	16
Figure 2.6: Mach number correction factors [26].	17
Figure 2.7: Secondary flows [27].	18
Figure 2.8: Trailing edge loss energy coefficient correlation.	19
Figure 2.9: Leakage flow visualization.	20
Figure 3.1: Benchmark flowpath [11].	29
Figure 3.2: S1 hub, mean and tip profiles.	31
Figure 3.3: R1 hub, mean and tip profiles.	32
Figure 3.4: S2 hub, mean and tip profiles.	33
Figure 3.5: R2 hub, mean and tip profiles.	34
Figure 3.6: 1D flowpath of E3 HPT.	37
Figure 3.7: 2D flowpath of E3 HPT.	37
Figure 3.8: S1 hub section control point distribution as example.	40
Figure 3.9: Grid distribution.	42
Figure 3.10: O grid distribution.	43
Figure 3.11: Boundary layer of S1 TE region.	43
Figure 3.12: Mesh maximum skew angle distribution [°].	44
Figure 3.13: Grid aspect ratio distribution.	44
Figure 3.14: Mesh dependency study.	45
Figure 3.15: S1 hub section blade loading.	47
Figure 3.16: S1 mean section blade loading.	48
Figure 3.17: S1 tip section blade loading.	48
Figure 3.18: B2B hub section rel Mach number contour at EDP.	49
Figure 3.19: B2B mean section rel Mach number contour at EDP.	50
Figure 3.20: B2B tip section rel Mach number contour at EDP.	50
Figure 3.21: Corrected mass flow rate comparison chart.	51
Figure 3.22: Efficiency data comparison chart.	52
Figure 3.23: Static pressure vs axial station at design point conditions.	54
Figure 4.1: Base flowpath comparison.	56
Figure 4.2: Profiles of base blades.	57
Figure 4.3: Thickness distribution of MF1.	58
Figure 4.4: Thickness distribution of MF2S.	58
Figure 4.5: Beta distribution of MF1.	59
Figure 4.6: Beta distribution of MF2S.	60
Figure 4.7: Uneven passage area distribution.	63
Figure 4.8: Flowchart of DoE study.	64
Figure 4.9: LHS variable distribution scatter plot.	66

Figure 4.10: MF2S mesh.	67
Figure 4.11: MF1 mesh.	67
Figure 4.12: MF2S Y_p histogram.	68
Figure 4.13: MF1 Y_p histogram.	69
Figure 4.14: MF2S corrected mass flow rate versus pressure ratio.	69
Figure 4.15: MF1 corrected mass flow rate versus pressure ratio.	70
Figure 4.16: MF2S Smith Chart.	70
Figure 4.17: MF1 Smith Chart.	71
Figure 4.18: MF2S exit flow angle effect on Y_p .	71
Figure 4.19: MF1 exit flow angle effect on Y_p .	72
Figure 4.20: MF2S exit Mach number versus pressure loss plot.	72
Figure 4.21: MF1 exit Mach number versus pressure loss plot.	73
Figure 4.22: MF2S Mach number effect on Y_p .	73
Figure 4.23: MF1 Mach number effect on Y_p .	74
Figure 4.24: MF2S effect of pitch to axial chord on Y_p .	74
Figure 4.25: MF1 effect of pitch to axial chord on Y_p .	75
Figure 4.26: Scatter plot of incidence angle versus pressure loss.	76
Figure 4.27: Scatter plot of blade turning angle versus pressure loss.	76
Figure 4.28: Scatter plot of splitter inlet offset angle versus pressure loss.	76
Figure 4.29: Comparison of actual and surrogate-predicted pressure loss.	77
Figure 4.30: Surrogate model prediction of pressure loss versus mid-frame α_2 .	78
Figure 4.31: Scraped effect of incidence angle versus Y_p .	78
Figure 4.32: Model results on deviation angle versus pressure loss.	79
Figure 4.33: 2D response surface of Y_p as function of $\beta_{1s,offset}$ and frame (i).	79
Figure 4.34: 2D response surface of Y_p as function of $\beta_{1s,offset}$ and $\Delta\beta_{main}$.	80
Figure 5.1: MF1 selection filter plot.	81
Figure 5.2: MF2S selection filter plot.	82
Figure 5.3: Profiles of selected configuration.	83
Figure 5.4: E3 3D geometry with mean section B2B CFD Ma contour plot.	84
Figure 5.5: MF1 geometry with mean section B2B CFD Ma contour plot.	84
Figure 5.6: MF2S geometry with mean section B2B CFD Ma contour plot.	84
Figure 5.7: MF1 Mach number contour plot.	85
Figure 5.8: MF2S Mach number contour plot.	86
Figure 5.9: MF1 LE Streamlines.	87
Figure 5.10: MF2S LE Streamlines.	87
Figure 5.11: MF1 mid passage Mach number contour plot.	88
Figure 5.12: MF2S mid passage Mach number contour plot.	89
Figure 5.13: Second stage stator loadings.	90

LIST OF TABLES

	<u>Page</u>
Table 3.1: Literature geometry data for stage 1.	30
Table 3.2: Literature geometry data for stage 2.	30
Table 3.3: EDP boundary conditions [12].	35
Table 3.4: RDG 12 boundary conditions.	35
Table 3.5: Aerodynamic map simulation points [14].	35
Table 3.6: Blade section wise geometrical parameters.	38
Table 3.7: Blade section wise geometrical parameters (cont).	39
Table 3.8: Mesh metrics.	42
Table 3.9: RDG 12 Cooling Flow Details.	46
Table 3.10: Corrected mass flow rate comparison table.	52
Table 3.11: Efficiency data comparison table.	53
Table 4.1: Geometrical parameters of base models.	56
Table 4.2: Fixed, independent and dependent variable list.	61
Table 5.1: Selected MF1 and MF2S parameters.	82
Table 5.2: Flow parameters of E3 - MF1 - MF2S at mean section.	85
Table A.1: Stage 1 hub comparison.	99
Table A.2: Stage 1 mean comparison.	99
Table A.3: Stage 1 tip comparison.	100
Table A.4: Stage 2 hub comparison.	100
Table A.5: Stage 2 mean comparison.	101
Table A.6: Stage 2 tip comparison.	101

1. INTRODUCTION

Turbomachines are devices that exchange energy from flow through the motion of rotating blades. A rotating blade row, which is called as a rotor, changes the stagnation enthalpy of the working fluid by either adding or extracting work from it [1]. These processes change the pressure and temperature state of the flow.

Turbomachines are grouped into two classes. Power-absorbing machines use shaft power to raise the pressure or head of a fluid, such as pumps, fans, and compressors. In power-producing machines the process is reversed, and power is generated by expanding the fluid. While the geometries of these machines vary a lot, the principle is the same; energy is converted between the mechanical domain and the fluid domain. The aerothermal design and analysis of such systems rely on principles from thermodynamics, fluid mechanics, and heat transfer disciplines.

The aerodynamic design of turbine section not only determines thermal efficiency and power production but also it determines the preliminary structural requirements and environmental performance of the engine. As engine architecture has evolved toward higher efficiency and lower weight, intermediate components such as turbine mid frames have become important as they provide structural support also guiding the flow between turbine stages [2]. The analysis and design of experiment study of such components form the focus of this thesis.

1.1. Aerodynamic Role of Mid Turbine Frames

The mid turbine frame MTF is located as inter turbine module in between the high-pressure turbine HPT and the low-pressure turbine LPT. MTFs are designed in both aerothermal and structural manners. The frame is guiding swirling HPT exit flow to the LPT rotor in the required flow angle and velocity distribution for aerodynamics, providing channels for oil and cooling network for thermal management also carries thrust, bending, torsion and bearing loads as a main structural part [2].

The aerodynamic challenge of MTF is low aspect ratio and large thickness-to-chord ratio constraints which are negatively affecting the blading efficiency. Such geometric

constraints are tending strong wakes and end-wall driven secondary flows which are causing non-uniform total pressure and swirl distributions at LPT [2], [3]. To mitigate these effects, turning mid turbine frames TMTFs are introduced. These frames are cambered struts so they can provide limited flow turning also they are partially compensating the high swirl generated by the HPT [2].

Experimental studies, which are by Santner [2] and Paradiso et al. [3], shows that even the optimized TMTFs can produce circumferential gradients in pressure distribution and flow angle distribution. Such findings highlight how geometry of strut and flow turning influence losses under realistic HPT exit conditions. Additionally, Krajnc et al. [4] demonstrated that adding a second splitter can improve flow uniformity and sensitivity to off-design conditions. In addition, Yasa et al. [5] investigated multi-splitter architectures and found that while their performance is comparable to conventional designs at nominal points, they are highly sensitive to off-design conditions. Their study specifically highlighted that negative flow incidence can cause significant flow separation on the structural vanes, leading to increased aerodynamic losses. Numerical investigations, which are by Liu et al. [6] and Xu et al. [7], showed that the axial length and diffusion characteristics of the inter-turbine duct influences flow separation and radial distribution of fluid.

While the flow through a TMTF is three-dimensional and influenced by secondary flows [8], the primary loss mechanisms can be studied in two-dimensional blade-to-blade (B2B) domains. Previous works by Orsenigo [8] and Russo et al. [9] show that 2D parametric analyses can isolate the first-order aerodynamic trends associated with turning angle, thickness ratio, and pitch-to-chord spacing. This method allows an efficient evaluation of geometric sensitivities without the computational cost of fully 3D CFD simulations. Additionally, Riebl et al. [10] highlighted the importance of adaptive geometry parameterization to efficiently represent such complex multi-body configurations in optimization loops.

In the scope of this thesis, the aerodynamic characteristics of a mid-turbine frame are analyzed by 2D B2B CFD calculations. Two concepts have been used for calculations which are single main vane named MF1 and main frame with splitter named as MF2S and then resulting characteristics are compared. Firstly, the upstream boundary conditions are derived from the open literature reference concept of the NASA GE E3

HPT for calculations [11]. The E3 HPT serves as a benchmark configuration for establishing realistic inflow conditions and ensuring the physical relevance of the MTF analyses. After validating E3 HPT flow path, stator of second stage is replaced by MF1 and MF2S concepts and their geometrical properties are parameterized to run design of experiment DoE study. In results aerodynamic characteristics and effects of geometrical parameters are presented. Finally, two concepts are compared at modified E3 HPT flow path.

This work aims to identify the aerodynamic mechanisms of turning struts and to quantify their profile loss trends on 2D domain. The results are expected to provide insight into the design of compact architectures where structural and aerothermal functions are in a single module.

1.2. NASA GE E3 HPT as Benchmark Configuration

The NASA Energy Efficient Engine E3 program was launched in the late 1970s. The objective was to develop advanced propulsion technologies for future high-efficiency engines. One of the key outcomes of this program was the design and experimental investigation of a two-stage, cooled high-pressure turbine (HPT) of General Electric GE. Because of the available design and experimental data, this turbine has become one of the most referenced configurations in open literature.

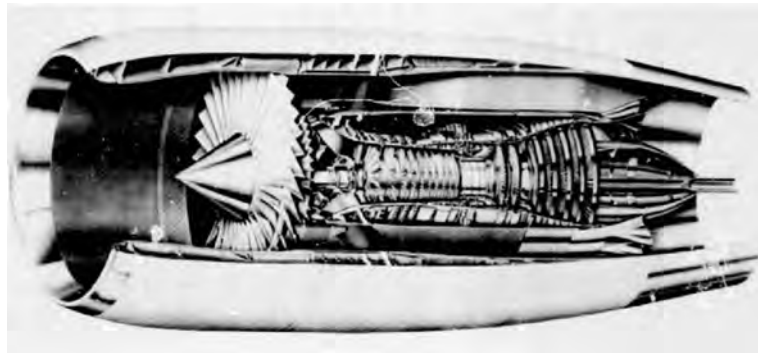


Figure 1.1: NASA/GE Energy Efficient Engine [12].

The experimental work reported by Timko et al. [11] shows turbine rig tests results under aerodynamically scaled turbine operating conditions. These tests provided measurements of stage efficiency, degree of reaction, exit swirl, and Reynolds-number sensitivity. While geometry and test results are reported by Timko et al, cooling

configuration of E3 HPT has been presented at the report by Halila et al. [13] in which internal and external network and blade film hole locations are available.

The E3 HPT is characterized by high aerodynamic loading, a two-stage axial layout, and advanced air-cooling technology, achieving design-point efficiencies exceeding 90% [11]. Both annular cascade tests and full-scale turbine rig experiments were executed by using aerodynamically corrected inlet total pressure, total temperature, and exit flow conditions of from map of HPT. During these experiments, cooling outputs to core flow were controlled to reflect realistic operating conditions. These results establish a reliable dataset for numerical validation studies.

Detailed design is provided in the High-Pressure Turbine Component Test Performance Report and the High-Pressure Turbine Test Hardware Detailed Design Report [11], [13]. These documents include vane and blade coordinates, spanwise Mach-number and temperature distributions, as well as cooling-scheme specifications.

Due to this level of documentation, the E3 HPT has been widely used for CFD validation studies. Murari et al. [14] reproduced aerodynamic maps from experimental data as comparing source-term cooling and geometrical modeling of cooling methodologies and also compared results from two commercial CFD tools. In follow up work, these results were extended to unsteady simulations which are demonstrating that both steady and time-accurate approaches can capture stage losses and overall performance trends [15].

Further studies expanded the scope of validation to higher-fidelity and system-level analyses. Claus et al. [12] compared zero-dimensional engine cycle model with 3D CFD simulations, validating the E3 geometry within different level of modeling. This work reinforced the suitability of the E3 turbine as a reliable digital baseline for advanced propulsion research.

Higher-fidelity approaches were later applied by several authors. Miki et al. [16] implemented a source-term-based cooling-air modeling within large-eddy simulations LES. They achieved agreement with measured temperature and pressure distributions with combustor interaction. Wehrel and Carvalho [17] employed the E3 configuration to develop consistent performance models for both E3 versions of GE and Pratt & Whitney.

The E3 HPT has also been used in combustor–turbine interaction (CTI) studies. Yang et al. and Zhang et al. investigated the effects of hot-streak and swirl clocking, demonstrating that upstream circumferential non-uniformities strongly influence the radial migration of thermal loads [18], [19]. These effects were shown to have a direct impact on surface heat-flux distributions and on the inflow conditions experienced by downstream components, including mid-frame structures.

In the present study, the exit flow of the first stage of E3 HPT is adopted as the inlet condition for the MTF simulations. Flow characteristics derived from NASA experimental datasets [11] and also performance cycle of Claus et al. [12] and validation results are compared with the numerical studies of Murari et al. [14] and experimental data [11]. After validation study, first stage of HPT exit flow has been used as inflow conditions for blade-to-blade CFD calculation of MTF to maintain realistic engine operating conditions.

2. METHODOLOGY

2.1. Theoretical Background on Axial Turbine Flow

Axial turbines are energy conversion devices that generate mechanical work from an air, combustion products and e.g. by expanding the flow through stationary vanes rows stators and followed by rotating blades rows rotors [1]. During this process, thermal energy is converted into kinetic energy and eventually into shaft work.

Axial turbine performance is commonly analyzed using a combination of static and stagnation (total) thermodynamic quantities. For a perfect gas, the total enthalpy is defined as the sum of static enthalpy and kinetic energy per unit mass:

$$H_0 = H_s + \frac{C^2}{2} \quad (2.1)$$

Where H_s is the static enthalpy and C is the magnitude of the absolute velocity [20]. Assuming a constant specific heat capacity at constant pressure, C_p , the total enthalpy relation can be expressed in terms of temperature:

$$T_0 = T_s + \frac{C^2}{2C_p} \quad (2.2)$$

Where T_0 and T_s represent the total and static temperatures, respectively.

For an adiabatic and reversible by means isentropic flow of a perfect gas, total quantities are constant. Under these conditions, the static and total states are related through compressible-flow relations:

$$\frac{P_0}{P_s} = \left(\frac{T_0}{T_s}\right)^{\frac{\gamma}{\gamma-1}} = \left(1 + \frac{\gamma-1}{2} Ma^2\right)^{\frac{\gamma}{\gamma-1}} \quad (2.3)$$

Where P_0 and P_s are the stagnation and static pressures, respectively, γ is the ratio of specific heats, and Ma denotes the Mach number.

The local Mach number is defined as the ratio of the flow velocity to the local speed of sound [20]:

$$\text{Ma} = \frac{C}{a}, a = \sqrt{\gamma R T_s} \quad (2.4)$$

Where a is the local speed of sound and R is the specific gas constant.

However, the expansion process is not isentropic in real turbomachinery flows. Irreversibility such as viscous dissipation, shock waves, secondary flows, and mixing losses lead to an increase in entropy and a corresponding loss in stagnation pressure across blade rows. In contrast, the stagnation temperature remains unchanged in stationary blade rows under adiabatic assumption and varies only in rotating rows where mechanical work is exchanged between the fluid and the rotor [21].

The efficiency of an axial turbine is calculated by using the total-to-total efficiency, denoted as η_{tt} . It is defined as the ratio of the actual specific work extracted by the turbine to the ideal isentropic work corresponding to the same pressure ratio:

$$\eta_{tt} = \frac{H_{01} - H_{03}}{H_{01} - H_{03,s}} \quad (2.5)$$

Where subscripts 1 and 3 denote the stage inlet and exit, respectively. The subscript s refers to the isentropic exit state obtained for the same inlet total conditions and exit total pressure [1].

2.1.1. The H-s (Mollier) Diagram and Expansion Process

The energy transformation in turbine can be tracked on enthalpy–entropy (H–s) diagram, which is also referred to as the Mollier diagram. This diagram provides a clear visualization of the ideal and real expansion processes, as well as the associated losses within the stator and rotor blade rows.

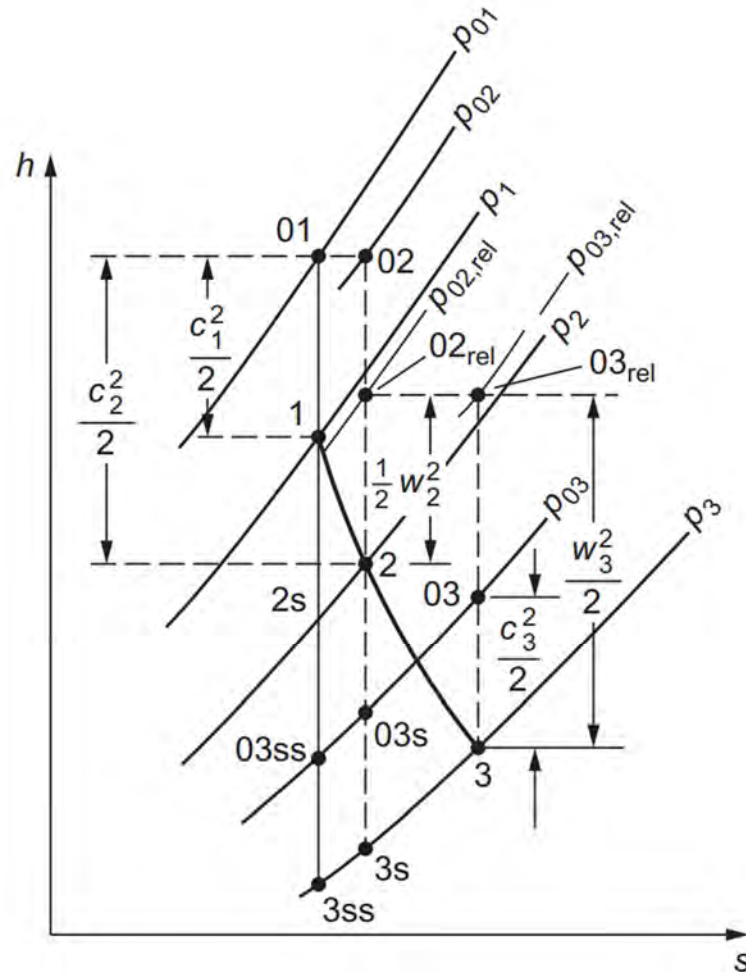


Figure 2.1: H-s diagram of axial turbine [1].

The flow is characterized by a total state defined by the total enthalpy H_{01} and entropy s_1 at the stage inlet. For a given inlet total pressure and temperature and for a specified exit static pressure, the ideal isentropic expansion to the stage exit is represented by a vertical line on the H-s plane, corresponding to constant entropy. This process terminates at the isentropic exit state denoted by $H_{3,ss}$ at the same entropy level s_1 . The corresponding static exit state is obtained by subtracting the kinetic energy associated with the outlet velocity from the stagnation enthalpy.

In an actual turbine stage, the expansion is distributed in between the stator and the rotor. Ideally, the stator expands the flow from state 1 to an intermediate isentropic state $2s$, while the rotor completes the expansion from $2s$ to the final isentropic exit state $3s$. However, irreversible losses are occurring in each blade row, causing a deviation at the actual expansion from the ideal isentropic lines.

Across the stator, the real process, which is from state 1 to state 2, accompanied by an increase in entropy from s_1 to s_2 , where s_2 is greater than s_1 . During this process, the static pressure decreases from P_{s1} to P_{s2} , and the flow accelerates as it passes through the nozzle passages. The stagnation enthalpy remains almost constant for adiabatic stator, such that H_{01} is nearly equal to H_{02} , while the stagnation pressure decreases from P_{01} to P_{02} due to profile losses, secondary flows, and boundary-layer effects [22].

The expansion through the rotor occurs from state 2 to state 3, with a further increase in entropy, s_3 . In contrast to the stator, both the static and stagnation enthalpies decrease across the rotor, as the fluid energy is converted into mechanical work on the shaft. As a result, the stagnation enthalpy drops from H_{02} to H_{03} . The stagnation pressure also decreases from P_{02} to P_{03} due to viscous dissipation and mixing losses within and downstream of the rotor passages [22].

On the Mollier diagram, the vertical distance between the inlet and exit stagnation enthalpy represents the specific work extracted by the turbine stage. Which is given by:

$$\Delta H_0 = H_{01} - H_{03} \cong H_{02} - H_{03} \quad (2.6)$$

The difference between the inlet stagnation enthalpy H_{01} and the isentropic exit stagnation enthalpy $H_{03,s}$ corresponds to the ideal specific work for the same inlet and exit pressures. Thus, the total-to-total efficiency defined in Equation (2.5) can be interpreted as the ratio of the actual enthalpy drop to the ideal isentropic drop on the H - s diagram.

The Mollier diagram also shows the distribution of the enthalpy drop between the stator and rotor. The segments corresponding to states 1–2 and 2–3 represent the static enthalpy changes across each blade row. Their relative magnitudes are directly related to the degree of reaction and have a strong influence on the velocity triangles, blade loading, and overall blade geometry [21].

2.1.2. Kinematics and the Euler Turbine Equation

It is necessary to introduce the kinematics of axial turbine flow using absolute and relative velocity triangles to relate the thermodynamic description presented in the

previous section. These triangles describe the flow direction, blade loading, and work exchange in both stationery and rotating frames of reference.

At any given radius, as radial component is neglected, the absolute velocity vector C can be decomposed into axial and tangential components, denoted by C_x and C_θ , respectively. This decomposition forms the basis of the velocity triangle representation:

$$\vec{C} = \begin{bmatrix} C_x \\ C_\theta \end{bmatrix} \quad (2.7)$$

The magnitude of the absolute velocity is obtained from its components as:

$$C = \sqrt{C_x^2 + C_\theta^2} \quad (2.8)$$

The blade speed U is tangential, and it depends on the angular velocity of the rotor and the local radius. In vector form, the blade speed can be expressed as a tangential velocity component, while its magnitude is given by the product of angular velocity and radius:

$$U = \begin{bmatrix} 0 \\ U \end{bmatrix}, U = \Omega r \quad (2.9)$$

The velocity of the fluid relative to the rotating blade, which governs the aerodynamic loading and incidence on the rotor, is obtained by subtracting the blade speed from the absolute velocity:

$$W = C - U \quad (2.10)$$

The absolute and relative flow angles are defined using the ratios of tangential to axial velocity components. The absolute flow angle α characterizes the direction of the flow in the stationary frame, while the relative flow angle β defines the flow direction as seen by the rotating blade:

$$\tan \alpha = \frac{C_\theta}{C_x}, \tan \beta = \frac{W_\theta}{W_x} \quad (2.11)$$

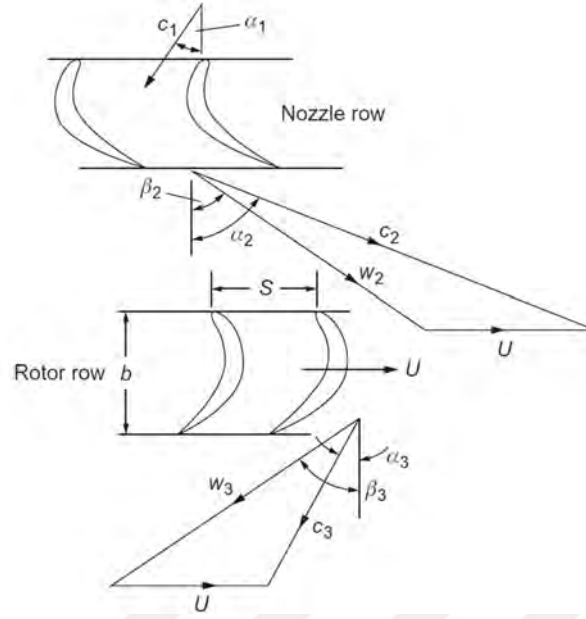


Figure 2.2: Velocity diagram of axial turbine [1].

In **Figure 2.2** illustrates the velocity triangles at the three stations of a axial turbine stage. Station 1 corresponds to the stator inlet, where the flow enters with absolute velocity C_1 and angle α_1 . Station 2 represents the stator exit and rotor inlet, where the flow is accelerated and turned to angle α_2 , reaching velocity magnitude C_2 . Station 3 denotes the rotor exit, where the flow leaves the stage with absolute velocity C_3 and angle α_3 . Within the rotor, the corresponding relative velocities W_2 and W_3 form angles β_2 and β_3 with the axial direction.

These kinematic quantities are directly related to the work exchange within the turbine through the Euler turbine equation. For a rotor operating at a constant mean radius, the specific work extracted from the flow is equal to the change in tangential momentum across the rotor [20]. This relationship can be expressed in terms of the tangential components of the absolute velocity as:

$$\Delta H_0 = W_t = U(C_{\theta 2} - C_{\theta 3}) \quad (2.12)$$

This expression shows that the work output of the turbine stage is directly proportional to the change in whirl velocity between the rotor inlet and exit. Consequently, the stator exit swirl, and the rotor exit flow angle plays a central role in determining the stage loading and overall turbine performance.

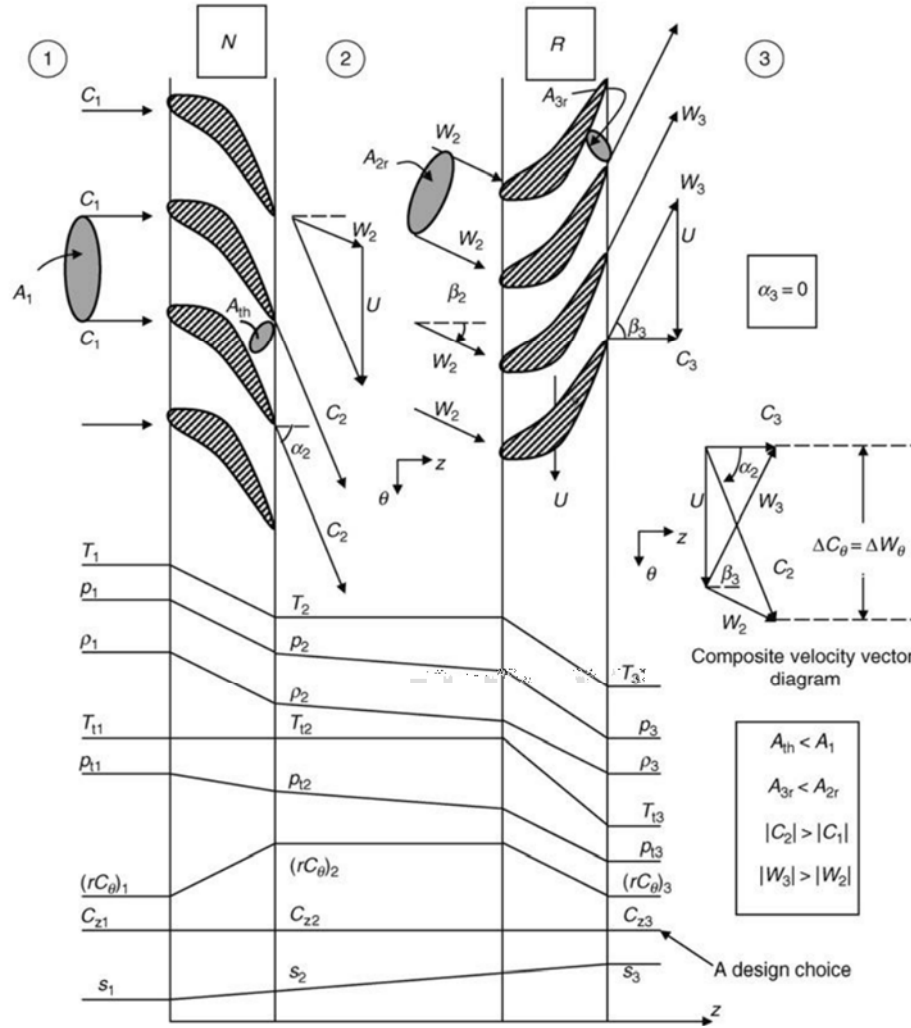


Figure 2.3: Axial turbine flow [20].

Figure 2.3 summarize the flow through an axial turbine stage, showing the stator row between stations 1 and 2, the rotor row between stations 2 and 3, and the downstream exit plane.

The stator is converting thermal energy into kinetic energy by expanding the flow through a converging passage. As the effective flow area decreases from A_1 to the throat area A_{th} , the absolute velocity increases from C_1 to C_2 and the flow is turned to angle α_2 . During this process, the static pressure and static temperature decrease from P_{s1} and T_{s1} to P_{s2} and T_{s2} . Under adiabatic conditions and in the absence of shaft work, the stagnation enthalpy and stagnation temperature remain approximately constant across the stator, such that H_{01} is nearly equal to H_{02} and T_{01} is nearly equal to T_{02} . Viscous effects and secondary flows, however, lead to a reduction in stagnation pressure from P_{01} to P_{02} and a corresponding increase in entropy from s_1 to s_2 . The

stator also establishes the radial distribution of rC_θ , generating the swirl required to drive the rotor.

The rotor blades rotate with circumferential speed U and receive high-energy, swirling flow leaving the stator. In the rotating frame, the relative velocity given by W is simply equal to absolute velocity subtracted by rotational velocity. The relative inlet and outlet velocities W_2 and W_3 form angles β_2 and β_3 with the axial direction, respectively. For highly loaded turbine stages, the magnitude of the relative velocity typically increases across the rotor, indicating that the flow is accelerated in the rotating frame even though it is decelerated in the absolute frame. In the absolute frame, the tangential velocity component decreases from $C_{\theta 2}$ to $C_{\theta 3}$, corresponding to a reduction in angular momentum and, through the Euler turbine equation, the extraction of mechanical work.

2.1.3. Work Coefficient, Flow Coefficient and Degree of Reaction

To compare stage behavior, turbines are often summarized using non-dimensional parameters. The most widely used of these are the flow coefficient, the stage work (or loading) coefficient, and the degree of reaction. Together, these parameters provide a compact description of the stage that directly links the velocity triangles, pressure distribution, and blade geometry.

The flow coefficient, denoted by ϕ , expresses the ratio of the axial velocity component to the blade speed at the mean radius and is defined as [20]:

$$\phi = \frac{C_x}{U} \quad (2.13)$$

For a given mean radius and rotational speed, the stage mass flow rate is approximately proportional to the flow coefficient through the continuity relation. Low values of ϕ are associated with highly loaded stages, characterized by strong flow turning and large relative flow angles. In contrast, higher values of ϕ correspond to a more axial flow field, reduced turning, and lighter aerodynamic loading.

The stage loading coefficient, denoted by ψ , provides a non-dimensional measure of the specific work extracted by the turbine stage relative to the square of the blade speed. Using the Euler turbine equation, the loading coefficient can be written as:

$$\psi = \frac{\Delta H_0}{U^2} = \frac{U(C_{\theta 2} - C_{\theta 3})}{U^2} = \frac{C_{\theta 2} - C_{\theta 3}}{U} \quad (2.14)$$

High values of ψ indicate that a larger amount of work is extracted in stage. However, increased stage loading generally requires stronger flow turning which is related to higher profile and secondary losses [1]. As a result, experimental and numerical investigations show that turbine efficiency typically reaches a maximum within a certain region of the ϕ – ψ design space, which is commonly represented using Smith charts for axial turbines.

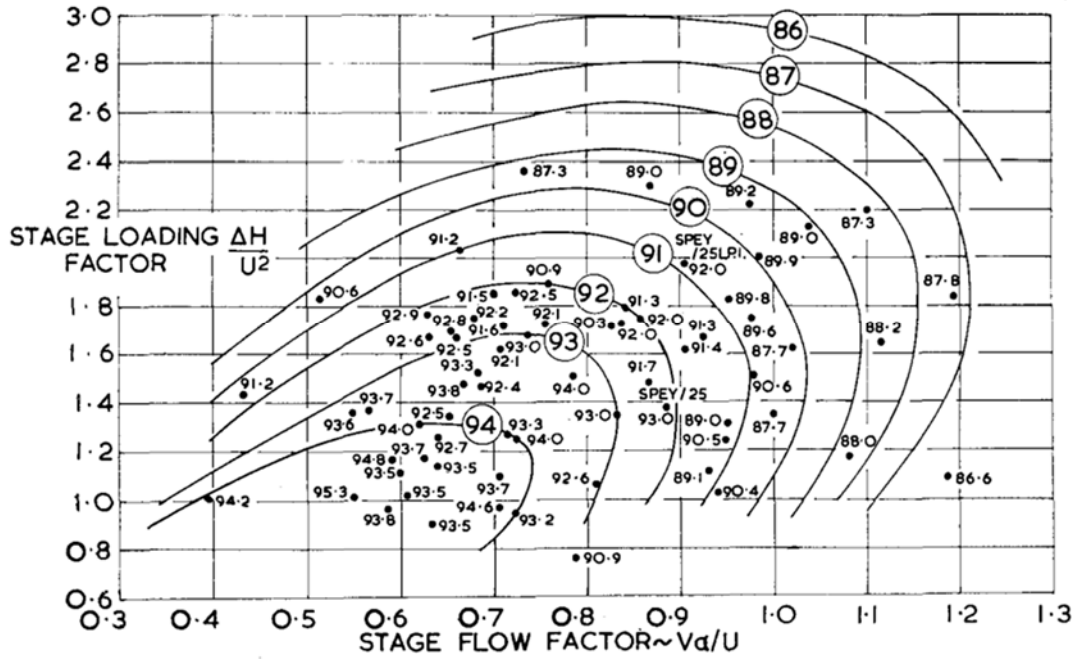


Figure 2.4: Smith Chart [23].

The degree of reaction, denoted by R , describes the fraction of the static pressure drop that occurs within the rotor relative to the total static-to-stagnation pressure change across the stage. In the present work, the degree of reaction is defined using static pressure as:

$$^{\circ}R = \frac{P_{s2} - P_{s3}}{P_{01} - P_{s3}} \quad (2.15)$$

A value of R equal to 0.5 corresponds to an equally distributed static pressure drop between the stator and the rotor. Such stages exhibit nearly symmetric velocity triangles and similar blade loading in both rows and are often referred to as 50-percent-

reaction or Parsons-type stages. Lower values of R indicate impulse-dominated behavior, in which most of the pressure drop occurs in the stator.

Taken together, the parameter set consisting of ϕ , ψ , and R provides a parameterization of an axial turbine that connects the thermodynamic representation on the Mollier diagram with the kinematic description provided by the velocity triangles.

2.2. Loss Models in Axial Turbines

Aerodynamic losses are critical to predict turbine performance and efficiency. In axial turbines, flow losses occur because of viscous dissipation, secondary flow interactions, and mixing phenomena. Decomposition of losses into classes such as profile losses, secondary losses, tip-clearance losses, and trailing-edge losses helps to assess performance degradation.

Profile losses occur due to boundary-layer growth and wake mixing through the suction and pressure surfaces of blades. Secondary losses develop near the hub and shroud regions. Tip-clearance losses are from leakage flow across the rotor tip gap. While trailing edge losses occur from wake formation and shock interactions downstream of the trailing edge.

Because of the highly three-dimensional and viscous effects of flow, direct analytical approach of losses is impractical. As a result, empirical mean line methods have been developed based on experimental cascade data. The first loss model is the Ainley–Mathieson [24] model. Later, The Craig–Cox Model [25] has refinement, which includes Reynolds-number corrections and improved trailing-edge treatments. Afterwards, Kacker and Okapuu (1982) [26] formulated a merged and robust model that remains widely applied mean line prediction method in turbine design.

2.2.1. Kacker-Okapuu Loss Model

Kacker and Okapuu combined the dominant loss mechanisms into a single empirical expression for the total loss coefficient:

$$Y_T = Y_P f_{Re} + Y_S + Y_{TET} + Y_{TC} \quad (2.16)$$

Where Y_p is profile loss, f_{Re} is Reynolds-number correction factor, Y_s is secondary losses, Y_{TET} is trailing-edge thickness loss, Y_{TC} is tip-clearance loss. Each term represents a distinct physical source of irreversible entropy generation within the turbine stage.

$$f_{Re} = \begin{cases} \left(\frac{Re}{2.10^5}\right)^{-0.4} & Re \leq 2.10^5 \\ 1.0 & 2.10^5 < Re < 10^6 \\ \left(\frac{Re}{10^6}\right)^{-0.2} & Re > 10^6 \end{cases} \quad (2.17)$$

2.2.1.1. Profile Loss

Profile losses result from viscous dissipation in the boundary layer and wake, influenced by the blade geometry, flow acceleration, and local Mach number [21]. The baseline expression extends the Ainley–Mathieson correlation and includes compressibility and shock effects:

$$Y_p = 0.914 \left(\frac{2}{3} Y_{p,AMD} K_p + Y_{shock} \right) \quad (2.18)$$

Here, $Y_{p,AMD}$ is the reference profile loss derived from low-speed cascade tests can be obtained by **Figure 2.5**, K_p is a Mach-number correction factor, and Y_{shock} represents additional loss caused by suction-side shock waves at high inlet Mach numbers.

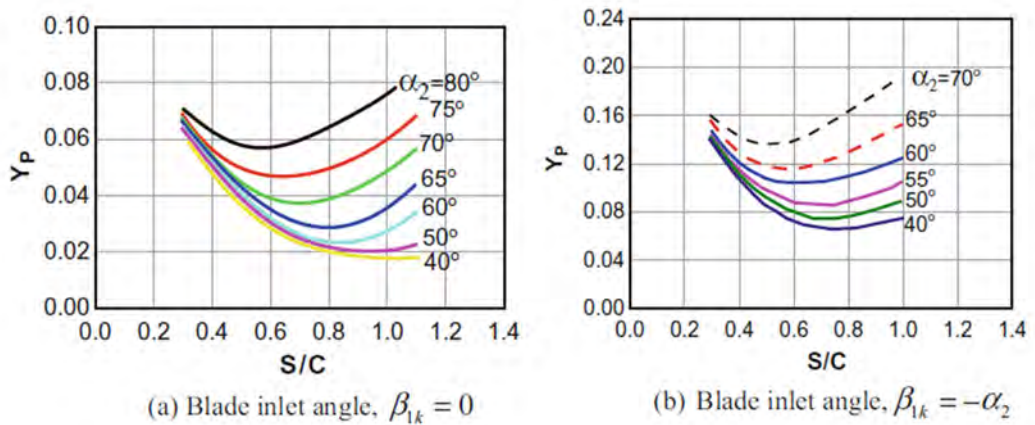


Figure 2.5: Profile loss coefficient for exit angles and S/C_{ax} ratio [27].

The shock term is calculated as:

$$Y_{\text{shock}} = 0.75(M_{1,\text{hub}} - 0.4)^{1.75} \left(\frac{R_H}{R_T} \right) \quad (2.19)$$

This formulation accounts for compressibility effects that become significant in transonic regimes where local accelerations lead to shock formation.

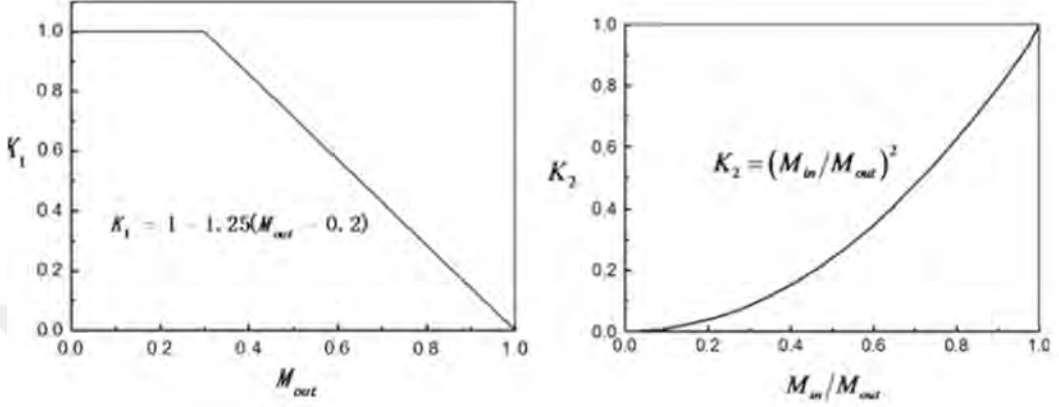


Figure 2.6: Mach number correction factors [26].

The Mach number correction factor K_P can be calculated through the equations (2.20), (2.21) and (2.22).

$$K_1 = 1 - 1.25(Ma_{out} - 0.2) \quad (2.20)$$

$$K_2 = \left(\frac{Ma_{in}}{Ma_{out}} \right)^2 \quad (2.21)$$

$$K_P = 1 - K_2(1 - K_1) \quad (2.22)$$

2.2.1.2. Secondary Loss

The interaction between the incoming end wall boundary layer and the transverse pressure gradients triggers the formation of secondary flow structures. Which arises due to the combined effects of pressure gradients and boundary-layer separation.

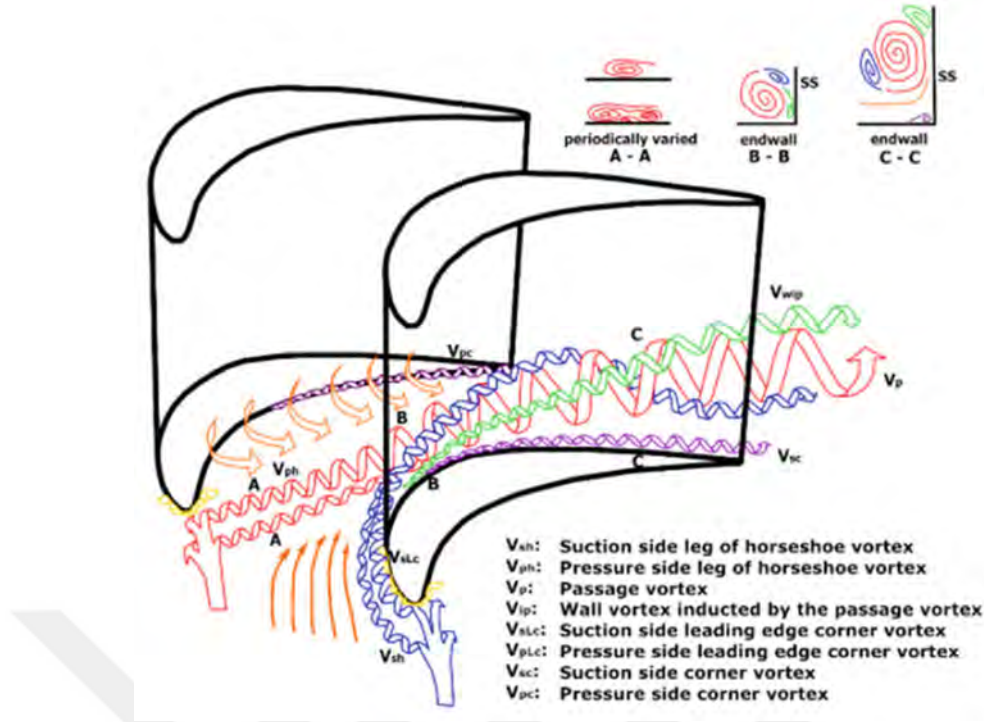


Figure 2.7: Secondary flows [27].

Kacker and Okapuu adopted a modified version of the Dunham–Came relation to represent these effects:

$$Y_S = 1.2Y_{S,AMD}K_S \quad (2.23)$$

Where $Y_{S,AMD}$ is referencing secondary flow loss.

$$Y_{S,AMD} = 0.0334f(AR)\left(\frac{C_l}{s/c}\right)^2 \frac{\cos^2 \alpha_2}{\cos^2 \alpha_m} \quad (2.24)$$

$C_l/s/c$ lift to pitch-to-chord ratio:

$$\begin{aligned} \frac{C_l}{s/c} &= 2(\tan \alpha_1 + \tan \alpha_2) \cos \alpha_m \\ \alpha_m &= \tan^{-1} \left[\frac{1}{2} (\tan \alpha_1 - \tan \alpha_2) \right] \end{aligned} \quad (2.25)$$

Where $f(AR)$ is an aspect-ratio-dependent correction:

$$f(AR) = \begin{cases} 1 - 0.25\sqrt{2 - h/c}, & h/c \leq 2 \\ h/c, & h/c > 2 \end{cases} \quad (2.26)$$

Finally, compressibility correction factor K_S is,

$$K_s = 1 - K_3(1 - K_p) \quad (2.27)$$

Secondary losses typically increase with lower aspect ratios and larger flow turning angles, where vortex strength and mixing intensity are greater.

2.2.1.3. Trailing-Edge Thickness Loss

The finite geometry of the trailing edge induces a base pressure deficit where it is leading to wake region that interacts with the primary flow, an effect that is non-linearly amplified in transonic regimes. This term depends primarily on the outlet Mach number and the thickness-to-throat ratio R_{TE}/o . Kacker and Okapuu introduced the following formulation:

$$Y'_{TET} = \left[1 - \frac{(k-1)}{2} M_2^2 \left(\frac{1}{1 - \Delta\phi_{TET}^2} - 1 \right) \right]^{-\frac{\gamma}{\gamma-1}} - 1 \quad (2.28)$$

The empirical coefficient $\Delta\phi_{TET}$ interpolates between axial-entry nozzle and impulse-blade limits. Its correlation has been shown in the **Figure 2.8**. Thicker trailing edges increase wake thickness and contribute to additional total-pressure loss, especially at higher Mach numbers.

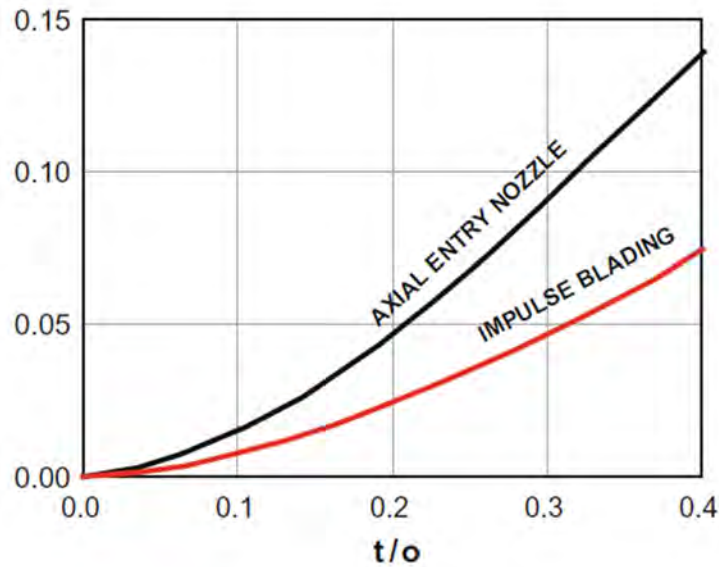


Figure 2.8: Trailing edge loss energy coefficient correlation.

2.2.1.4. Tip Clearance Loss

Tip-clearance losses occur in rotor rows where a finite radial gap exists between the blade tip and the casing. Pressure difference between the suction and pressure sides rolls up into a tip vortex, which is generating blockage and irreversible losses. The **Figure 2.9** visualize the leakage flow.

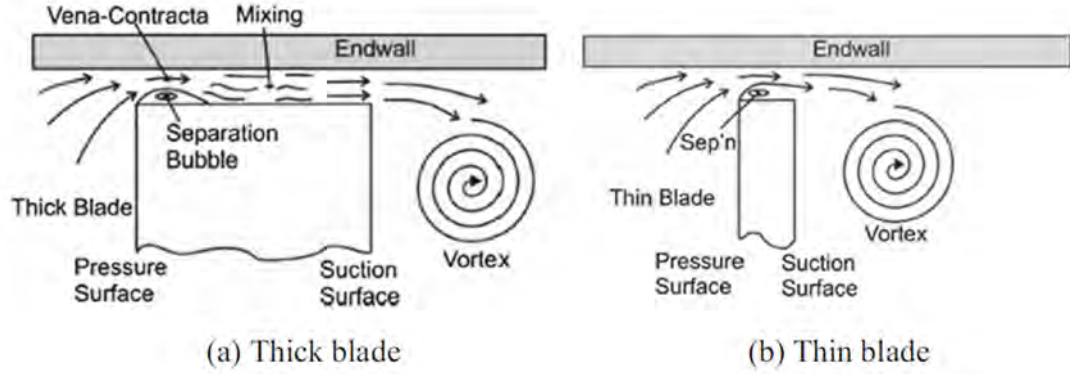


Figure 2.9: Leakage flow visualization.

The efficiency loss caused by tip clearance is estimated as:

$$\frac{\Delta\eta}{\eta_0} = 0.93 \left(\frac{\Delta k}{h \cos \alpha_2} \right) \left(\frac{R_{TIP}}{R_{HUB}} \right) \quad (2.29)$$

This term is highly sensitive to clearance height and blade loading.

The Kacker–Okapuu loss model remains a cornerstone of axial turbine mean line design, offering a good balance between physical accuracy and computational simplicity. It continues to serve as the empirical foundation for validating CFD simulations and for correlating design-stage predictions with experimental data.

2.3. Pritchard Eleven-Element Method

Pritchard [28] introduced an analytical geometry model by employing a set of eleven geometric variables. This method effectively removes dependency on dense point-coordinate data for blade section definition.

2.3.1. Model Description

In this formulation, the complete airfoil at a given spanwise station is described by the following independent eleven geometric parameters, each representing a physical or aerodynamic feature of the blade [28]:

Airfoil Radius (R): Radial position of the section, used for spanwise interpolation from hub to tip.

Axial Chord (C_{ax}): The chord length projected on the axial direction; determines blade spacing and compactness.

Tangential Chord (C_t): The projection of the chord in the circumferential direction; governs turning and work extraction capability.

Throat (σ): Minimum flow passage between adjacent blades which defines the mass-flow capacity and Mach number distribution.

Unguided Turning ($\Delta\beta_u$): Difference between the geometric exit angle and the flow exit angle; accounts for deviation and curvature effects.

Inlet Metal Angle (β_{k1}): Blade metal angle at the leading edge relative to the axial direction; determines flow incidence and turning.

Exit Metal Angle (β_{k2}): Blade metal angle at the trailing edge; defines outlet flow direction.

Inlet Wedge Angle (ω_1): The included angle between pressure and suction sides at the leading edge; affects stagnation-point curvature and acceleration onset.

Leading-Edge Radius (R_{LE}): Controls flow attachment and sensitivity to incidence variations.

Trailing-Edge Radius (R_{TE}): Influences wake thickness and mixing losses downstream.

Number of Blades (B.No): Defines the blade pitch and total flow area across the annulus.

Several cascade-level quantities are not specified independently in the eleven-element method; instead, they are derived from the independent set. Typical dependent quantities include pitch s , solidity C_{ax}/s , stagger angle ξ and the suction-pressure side

polynomial coefficients and control-point locations used to generate the final coordinates.

2.3.2. Geometric Construction

The airfoil surface is generated by determining five control points where they are located at the leading-edge, trailing-edge and throat regions and connecting them through smooth analytical curves [28].

The leading-edge and trailing-edge surfaces are defined by circular arc and the rest of surfaces are expressed as a cubic polynomial:

$$y(x) = a + bx + cx^2 + dx^3 \quad (2.30)$$

The coefficients are derived from endpoint coordinates and tangent directions, ensuring first-derivative continuity at all junctions. This provides a fully analytical, smooth profile that can be directly used in computational or CAD-based design environments.

The eleven-element model thus allows direct geometric manipulation based on aerodynamic parameters such as flow turning, throat area, and blade loading. Because of its simplicity and robustness, it is a widely used geometry-generation method for axial turbine design workflows.

2.4. Computational Fluid Dynamics Fundamentals

Computational Fluid Dynamics (CFD) serves as an analytical tool in turbomachinery design to analyze aerodynamic performance and flow behavior. It's not only cost effective and practical but also provides high fidelity data which is often inaccessible or expensive to capture through testing. In general, three-dimensional CFD simulations can capture complex flows such as flow separations and flow turning into turbine passages and secondary flows.

In this thesis, CFD is applied using a quasi-three-dimensional Blade-to-Blade B2B approach on selected stream surfaces. The purpose of this approach is to analyze blade loading, profile losses and flow turning at desired spanwise locations. Fully three-

dimensional RANS simulations are not performed within the scope of this work. Whenever 3D effects are discussed, they are referenced through literature.

All aerodynamic evaluations and design decisions in the present study are therefore based on B2B CFD results, supported by classical turbomachinery correlations and available experimental data. This approach allows an efficient investigation of the baseline turbine and provides physically consistent inflow conditions for the subsequent mid turbine frame analyses.

2.4.1. Governing Equations

The flow is modelled as a steady, compressible gas with temperature-dependent properties. Under the steady Reynolds-averaged Navier–Stokes (RANS) assumption, the governing equations express conservation of mass, momentum, and energy for the mean flow field. In compact tensor notation [29], the equations are;

Continuity equation:

$$\frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad (2.31)$$

Momentum equation:

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} - \frac{\partial}{\partial x_j} \left(\overline{\rho u'_i u'_j} \right) \quad (2.32)$$

Energy equation:

$$\frac{\partial(\rho H_0 u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right) + \Phi \quad (2.33)$$

Here, ρ is the density, u_i are the mean velocity components, P is the static pressure, T is the static temperature, H_0 is the total enthalpy, k is the thermal conductivity and Φ represents viscous dissipation. The viscous stress tensor is denoted by τ_{ij} . The term $-\overline{\rho u'_i u'_j}$ represents the Reynolds stresses introduced by Reynolds averaging. These additional terms give rise to the turbulence-closure problem and are modelled using an appropriate turbulence model, as described in the next subsection.

The mathematical system is closed by the ideal-gas law which links pressure, density, and temperature:

$$P = \rho RT \quad (2.34)$$

These equations form a system of nonlinear partial differential equations solved iteratively using a finite-volume approach.

2.4.2. Turbulence Modeling

Because fully resolving all turbulent eddies using Direct Numerical Simulation (DNS) is computationally infeasible for high-Reynolds-number turbomachinery flows, the Reynolds-averaged Navier–Stokes (RANS) formulation is adopted. In this approach, the effects of turbulence on the mean flow field are represented through a turbulence model, which provides closure for the Reynolds stress terms appearing in the momentum equations.

2.4.2.1. k – ω SST Model

In the present study, the Shear Stress Transport (SST) k – ω turbulence model is used [2]. The SST model combines the near-wall accuracy of the k – ω formulation with the free-stream robustness of the k – ε model. It solves two additional transport equations for the turbulent kinetic energy k and the specific dissipation rate ω :

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_j)}{\partial x_j} = P_k - \beta^* \rho k \omega + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right] \quad (2.35)$$

$$\begin{aligned} \frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \omega u_j)}{\partial x_j} &= \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2(1 \\ &\quad - F_1) \frac{\rho \sigma_{\omega 2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \end{aligned} \quad (2.36)$$

Here, P_k denotes the production of turbulent kinetic energy, μ_t is the eddy viscosity, and F_1 is the blending function that controls the transition between the k – ω and k – ε formulations.

The SST model has become a common choice in turbomachinery applications due to its robust performance in flows with adverse pressure gradients and separation. Near solid walls, the model behaves as a $k-\omega$ formulation, allowing integration down to the wall without relying on empirical wall functions. Away from the wall, the blending functions gradually shift the model behavior towards $k-\epsilon$, reducing sensitivity to the free-stream value of ω . This hybrid formulation provides a reliable balance between near-wall accuracy and numerical robustness for the present blade-to-blade simulations.

2.4.3. Blade-to-Blade (B2B) Numerical Analysis

While full 3D CFD provides the most detailed representation of the flow, it is still relatively expensive when many design evaluations must be completed. Therefore, a quasi-3D Blade-to-Blade (B2B) approach is adopted in. The B2B method is placed between simple 1D calculations and fully three-dimensional simulations. It resolves the detailed pressure distribution around the airfoil while still treating the flow in a two-dimensional sense on a selected stream surface.

2.4.4. Quasi-3D Approximation

The B2B analyses are performed using a commercial turbomachinery CFD tool. The governing equations are solved on an axisymmetric stream surface of revolution, commonly referred to as the S1 surface [30]. This surface corresponds to a particular radial location in the annulus and represents a cut of the flow field in the meridional direction. The circumferential variation is assumed to be periodic over one blade pitch, so that only a single blade passage needs to be considered.

The main effect of the three-dimensional geometry is the variation of annulus height in the axial direction due to hub and shroud contours. This change in flow area is accounted for through the Axial Velocity Density Ratio (AVDR). AVDR represents the ratio of the local axial mass flux to its circumferentially averaged reference value. In the quasi-3D formulation, AVDR appears in the continuity equation as a source term to effectively model the acceleration or deceleration associated with area contraction or expansion.

By including AVDR, the B2B solution on the S1 surface can calculate the correct meridional acceleration of the flow even the calculation itself is on a 2.5D dimensional domain with %5 arbitrary height on S1 surface. Secondary flows, tip-clearance vortices and other inherently three-dimensional phenomena are not resolved explicitly in this calculation. Instead, the B2B results are primarily used for blading, preliminary performance trends and for providing boundary conditions and design input to subsequent 3D simulations.

2.4.5. Grid Topology and Near-Wall Resolution

The computational domain around the airfoil is discretized using a structured multi-block grid created in commercial tool.

While creating a grid an H–O–H topology is adopted. An O-type block is wrapped around the airfoil surface to maintain high orthogonality and smooth spacing in the wall-normal direction. Upstream and downstream of the blade, H-type blocks are used to align the grid with the main flow direction and to simplify the imposition of inlet and outlet boundary conditions. The inlet and outlet extensions are chosen sufficiently long to minimize any artificial influence of the boundaries on the blade loading.

Accurate prediction of viscous losses and possible boundary-layer separation require sufficient grid resolution near the solid surfaces. The mesh is therefore refined strongly in the wall-normal direction to achieve a dimensionless wall distance of $y^+ \leq 5$ at the first cell center on the blade and end wall surfaces. This level of resolution allows the SST turbulence model to be integrated directly to the wall without using empirical wall functions, which generally leads to more reliable predictions of wall shear stress and heat transfer. Additional clustering is applied near the leading and trailing edges to resolve the rapid changes in curvature and pressure in these regions.

2.4.6. Discretization and Solver Settings

The governing equations on the S1 surface are solved using a pressure-based finite volume method for compressible flow. Conserved variables are stored at cell centers, and numerical fluxes are evaluated at cell faces.

Convective fluxes are discretized using a third-order upwind Advection Upstream Splitting Method (AUSM). AUSM-type schemes are well suited for turbomachinery applications, as they can robustly handle subsonic and transonic flow regimes while maintaining low numerical dissipation.

An implicit Gauss–Seidel time-stepping scheme is employed to march the solution towards a steady state. The implicit formulation improves numerical robustness and allows the use of higher Courant numbers compared to explicit schemes, which is advantageous for compressible turbomachinery flows.

To accelerate convergence, a two-level multigrid method is applied, enabling rapid damping of low-frequency error components. The solution is iteratively advanced on both coarse and fine grids until convergence is achieved.

To improve numerical stability during the initial stages of the computation, the solver is first initialized using a first-order accurate discretization of the convective terms. After the flow field has stabilized and the residuals have decreased by several orders of magnitude (typically after approximately 30 iterations), the discretization is switched to the higher-order AUSM scheme to obtain the final solution accuracy.

2.4.7. Boundary Conditions and Convergence

At the inlet plane, circumferential distributions of total pressure, total temperature, and flow angle are prescribed. These quantities are consistent with the upstream mean line design and with the intended operating point of the turbine mid frame. The inlet turbulence intensity is set to 5%, and the corresponding turbulence quantities are initialized accordingly.

At the outlet plane, a mass-averaged static pressure is imposed. This condition fixes the stage pressure ratio and allows the solver to adjust the mass flow rate until a consistent solution is reached.

Translational periodicity is applied in the circumferential direction, so that only a single blade pitch is modelled while representing an infinite, periodically repeating cascade.

The B2B solution is considered converged when the following criteria are satisfied. The residuals of mass, momentum, and energy decrease below 10^{-5} . The global mass-

flow imbalance falls below 0.01%, ensuring strict overall mass conservation. In addition, key aerodynamic quantities, such as the total-pressure loss coefficient and the flow turning angle, reach steady values and exhibit no systematic drift with further iterations.

$$\text{MFR}_{\text{imbalance}} = |\dot{m}_{\text{in}} - \dot{m}_{\text{out}}| / \dot{m}_{\text{in}} \quad (2.37)$$

Under these conditions, the B2B results provide a consistent and sufficiently accurate basis for comparing blade sections and for supplying boundary conditions and design input for the subsequent three-dimensional MTF analyses.



3. BASELINE MODEL AND VALIDATION

In this work, firstly the verified and validated well-documented high-pressure turbine configuration is used as benchmark before it is applied to the turbine mid frame studies. For this purpose, the NASA E3 High Pressure Turbine (HPT) is selected as a reference design. The E3 turbine provides an extensive experimental database, including detailed aerodynamic measurements at several spanwise locations and operating conditions, which makes it an appropriate benchmark for assessing both the physical modeling and the numerical implementation used in the present study.

3.1. NASA E3 HPT as Reference Design

The reference geometry chosen for validation is the High-Pressure Turbine (HPT) from the NASA Energy Efficient Engine E3 project, developed by General Electric.

The reconstructed flow path is as seen **Figure 3.1 [11]**:

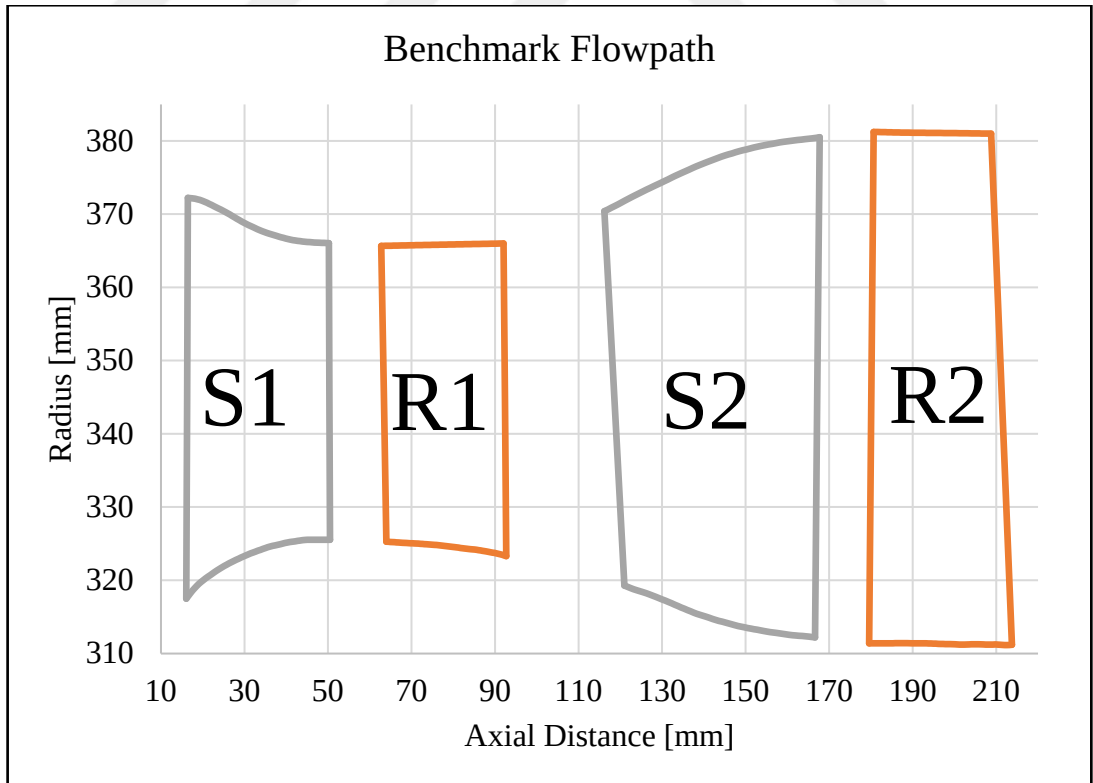


Figure 3.1: Benchmark flowpath [11].

The spanwise geometrical parameters for the stator and rotor rows are summarized in **Table 3.1** and **Table 3.2** [11];

Table 3.1: Literature geometry data for stage 1.

Component	Parameter	Hub	Mean	Tip
Stator 1	Blade Count: 46			
	Radius [mm]	325.76	345.76	365.76
	Axial Chord [mm]	33.76	33.78	33.83
	TE Thickness [mm]	0.965	0.965	0.965
	Unguided Turning [°]	9.2	8.4	8.7
	TE Wedge Angle [°]	10.2	9.2	9
Rotor 1	Blade Count: 76			
	Radius [mm]	323.37	344.68	366.01
	Axial Width [mm]	28.7	28.7	28.7
	TE Thickness [mm]	0.965	0.965	0.965
	Unguided Turning [°]	13	13	13
	TE Wedge Angle [°]	12.5	12.5	12.5
	Tip Clearance			

Table 3.2: Literature geometry data for stage 2.

Component	Parameter	Hub	Mean	Tip
Stator 2	Blade Count: 48			
	Radius [mm]	312.17	346.33	380.49
	Axial Width [mm]	45.03	49.05	53.09
	TE Thickness [mm]	0.965	0.965	0.965
	Unguided Turning [°]	10.5	11	11.5
	TE Wedge Angle [°]	9	9.5	10
Rotor 2	Blade Count: 70			
	Radius [mm]	311.15	346.08	381
	Axial Width [mm]	33.53	30.73	27.94
	TE Thickness [mm]	1.575	1.118	1.27
	Unguided Turning [°]	14.5	15.5	12
	TE Wedge Angle [°]	14	14	9
	Tip Clearance			

The blade profiles used for the validation study were reconstructed based on reference points provided in Timko's report [11]. The profiles for Stator 1 (S1), Rotor 1 (R1), Stator 2 (S2), and Rotor 2 (R2) are depicted in **Figure 3.2**, **Figure 3.3**, **Figure 3.4** and **Figure 3.5**, respectively.

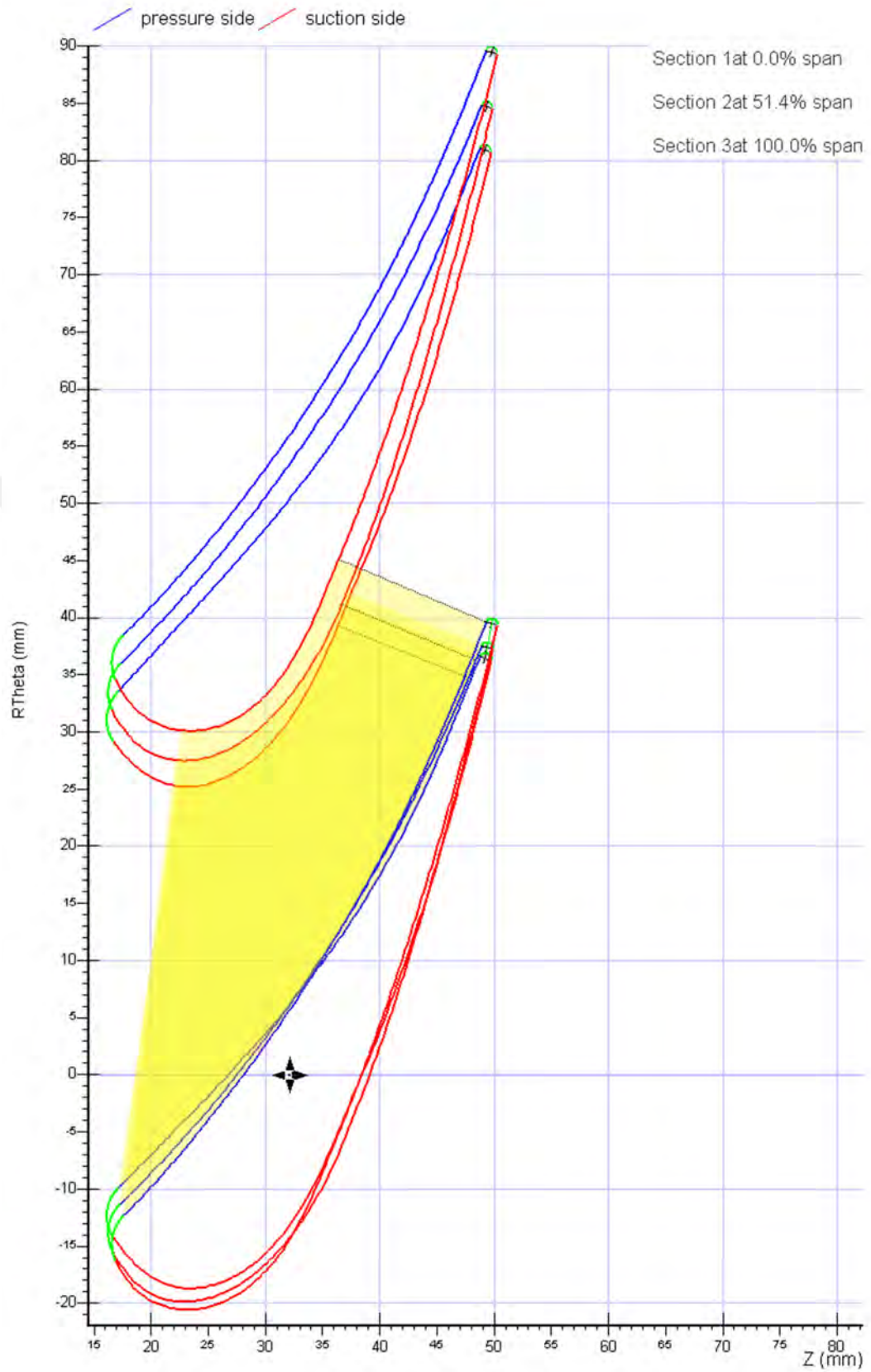


Figure 3.2: S1 hub, mean and tip profiles.

The R1 profiles are plotted in **Figure 3.3**.

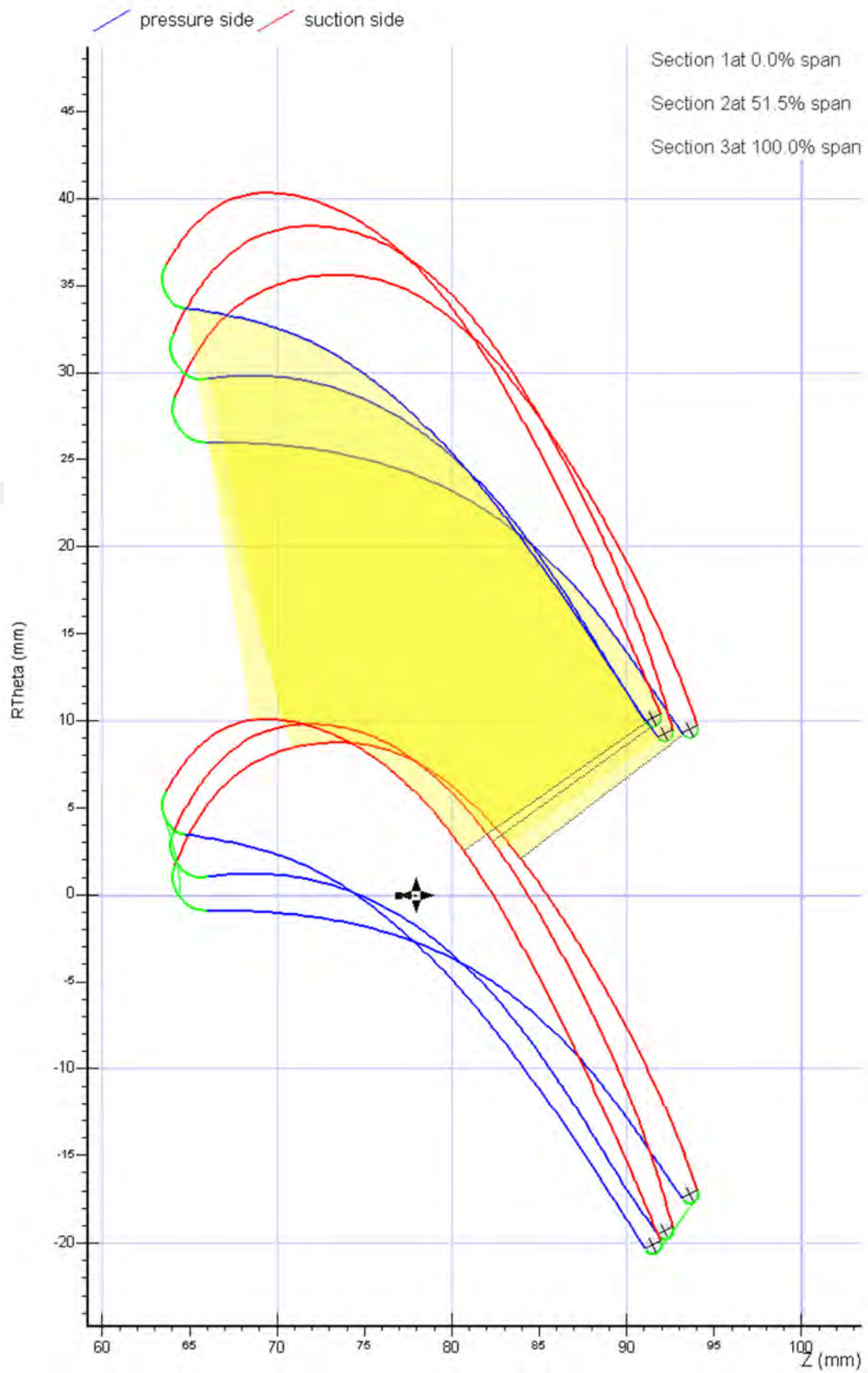


Figure 3.3: R1 hub, mean and tip profiles.

The S2 profiles are plotted in **Figure 3.4**.

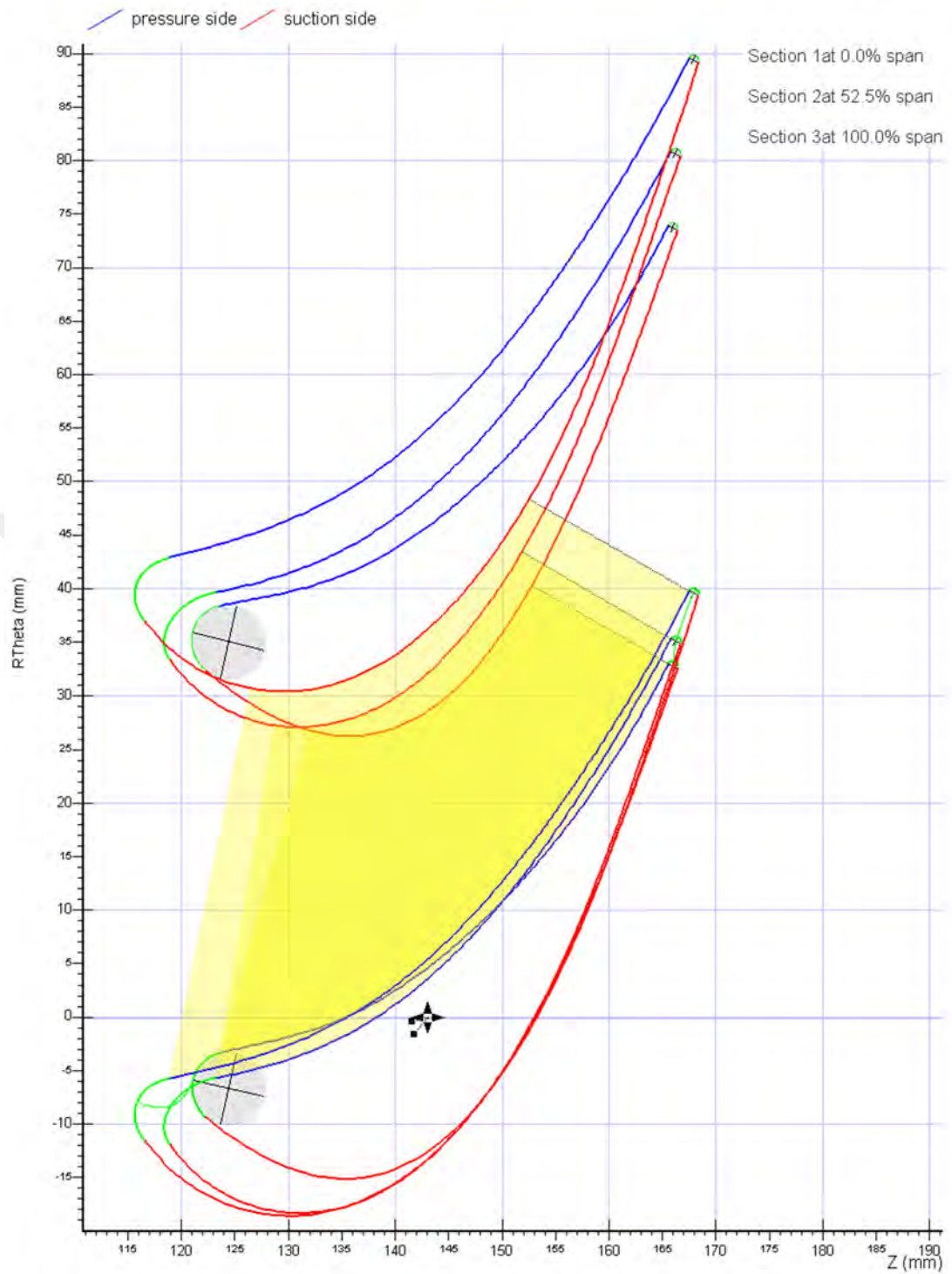


Figure 3.4: S2 hub, mean and tip profiles.

The R2 profiles are plotted in **Figure 3.5**.

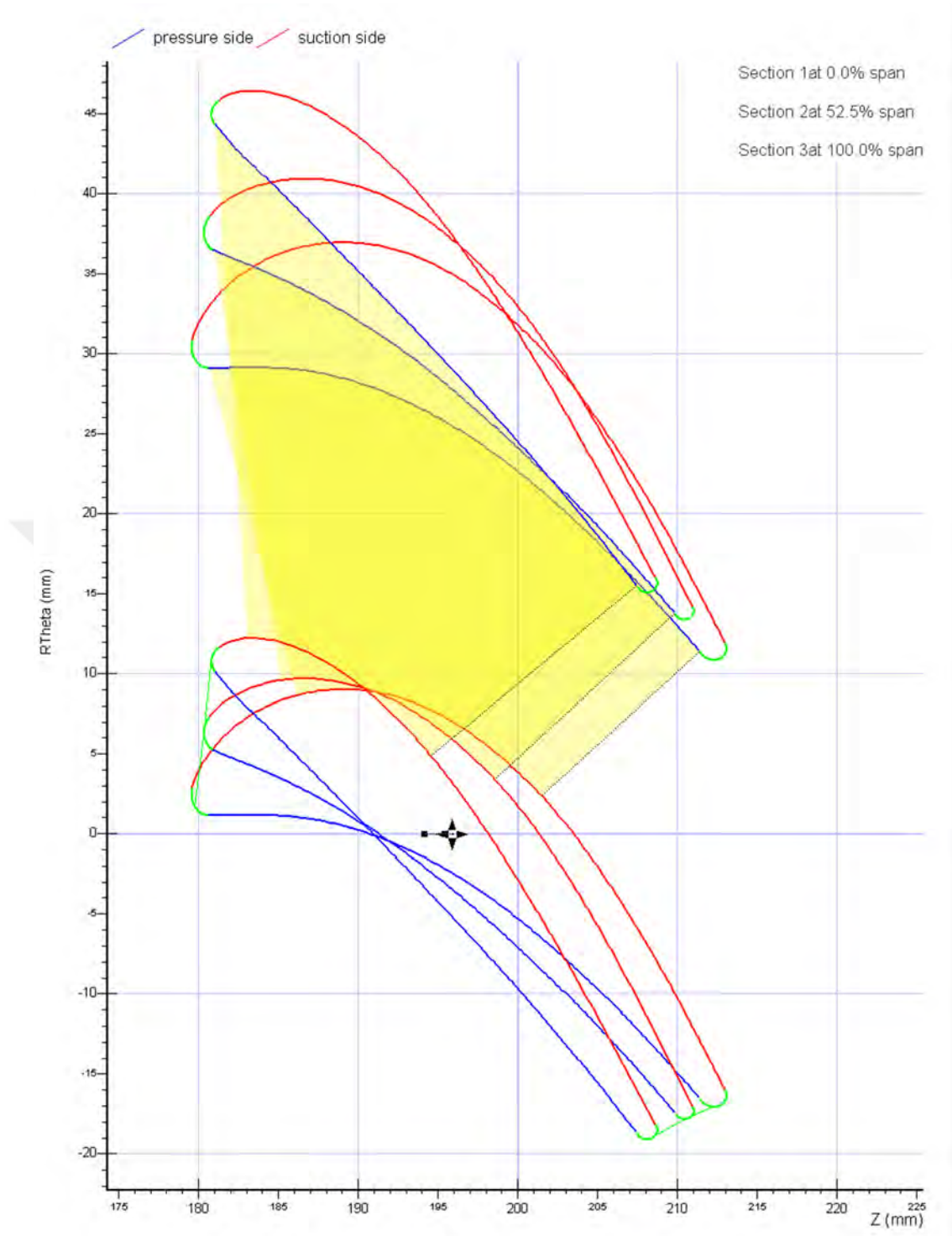


Figure 3.5: R2 hub, mean and tip profiles.

To validate the aerodynamic performance of the HPT across its entire operating envelope, several cases representing different engine missions were analyzed. These cases involve significantly varying boundary conditions, allowing for a thorough characterization of the turbine.

The engine cycle design point is adopted from Claus et al [12] and detailed below:

Table 3.3: EDP boundary conditions [12].

Parameter	Value
T_0 [K]	1527.2
P_0 [kPa]	1257.5
Inlet $\dot{m}$ [kg/s]	27.07
$\dot{m}_{corr}$ [kg/s]	5.02
RPM	12384
PR	4.945
η_{tt}	0.927
PWR [MW]	14.73

Here is also aerodynamically scaled condition which is used by Murari et al. [14] at CFD studies and Timko et al. [11] while collecting experimental data.

Table 3.4: RDG 12 boundary conditions.

Parameter	Symbol	Unit	Value
Total Inlet Pressure	$P_{0,in}$	kPa	345.98
Total Inlet Temperature	$T_{0,in}$	K	709.37
Inlet Mass Flow Rate	$\dot{m}$	kg/s	10.9
Rotational Speed	N	rpm	8295
Outlet Static Pressure	$P_{s,out}$	kPa	62

At their studies this condition is stated as Design Point by Murari et. al [14] and RDG 12 by Timko et al [11].

In this thesis, the specific test cases are primarily selected from Murari et al. [14] to investigate the turbine characteristics under off-design conditions. The selected validation cases are listed in **Table 3.5**.

Table 3.5: Aerodynamic map simulation points [14].

Case #	RDG #	$\dot{m}_{corr}$ [kg/s]	N[RPM]	P_{0in} [kPa]	T_{0in} [K]	PR
Design pt	12	10.9	8295	345.98	709.37	4.96
Test pt-1	51	10.9	8885	346.05	708.33	5.87
Test pt-4	227	10.59	8194	345.63	709.32	2.25
Test pt-6	249	7.3	2534	345.98	714.33	1.2
Test pt-8	75	10.94	5734	345.7	708.97	3.73
Test pt-10	59	11.12	4827	350.53	710.29	5.49

3.2. Numerical Method

The aerodynamic design and analysis process follows a hierarchical multi-fidelity approach, progressing from lower-order modeling to high-fidelity simulations. This strategy ensures both computational efficiency and high accuracy in resolving complex flow features.

Initially, a one-dimensional (1D) analysis was conducted to rapidly establish the fundamental flow properties, velocity triangles, and thermodynamic states at the design point. This preliminary step allowed for the quick verification of the cycle matching and the definition of initial geometrical constraints.

During the 1D calculation the governing equations and loss correlations employed are detailed in Chapter 2. This preliminary step aimed to confirm the cycle matching and establish boundary conditions consistent with the experimental data. It served as a starting point for the detailed numerical simulations.

After quickly initializing 1D geometry, 2D design is conducted. From 1D to 2D these parameters are transferred as initials.

- Basic Flowpath
- Basic Blade Parameters, (Angles, Thicknesses)
- Basic Flow Parameters, (Flow Angles, Mach Numbers, Pressure, Temperature)

While the 1D analysis provides a quick estimation using simplified linear geometry, it neglects the impact of end wall curvature. To address this, detailed hub and shroud contours were defined in the 2D study. These contours are critical for two main reasons:

1. They define the meridional flow path area, which controls the flow acceleration and diffusion through the passage.
2. The curvature of the hub and shroud induces radial forces, causing a redistribution of mass flow and pressure from hub to tip.

Therefore, modeling these contours in 2D is essential to capture the spanwise variations that the 1D model cannot resolve.

The difference between 1D and 2D flow path of E3 HPT can be seen from **Figure 3.6** and **Figure 3.7**.

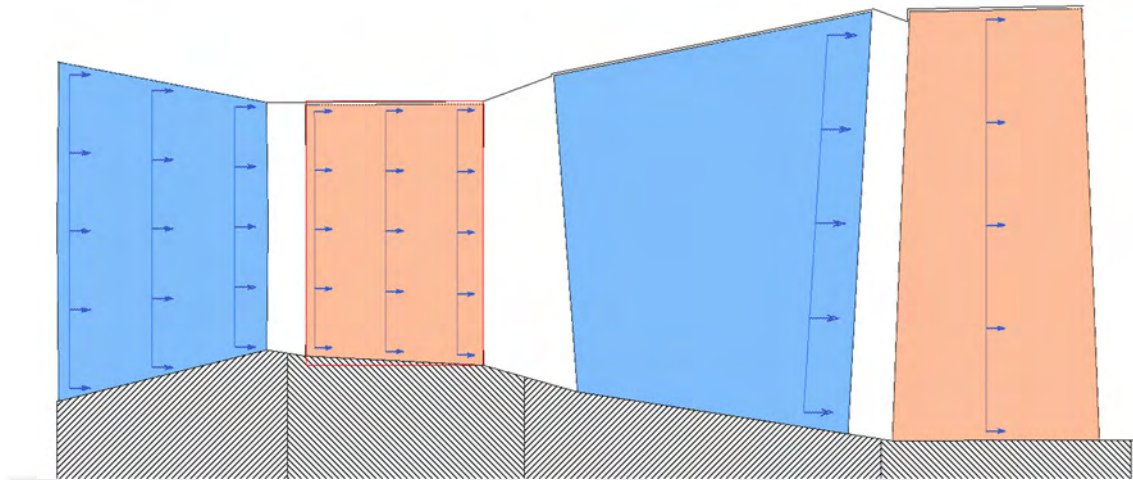


Figure 3.6: 1D flowpath of E3 HPT.

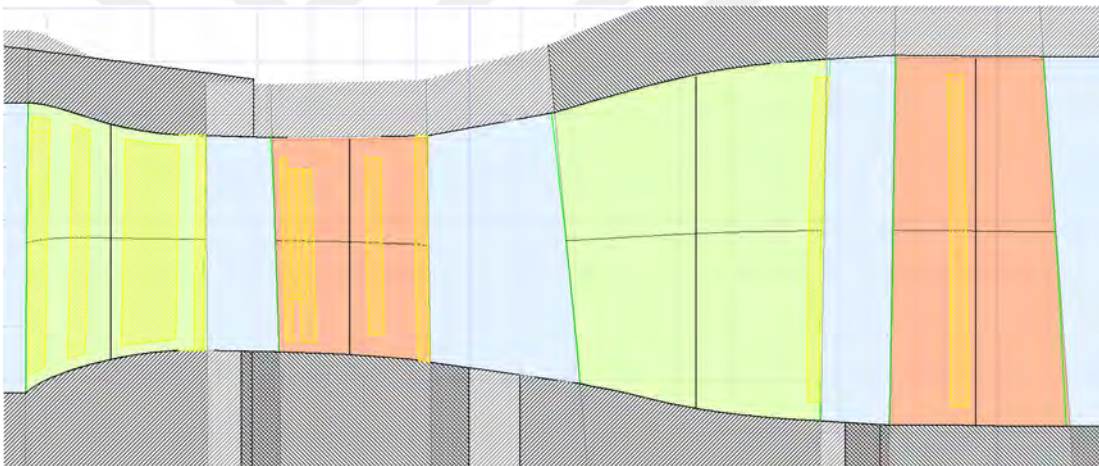


Figure 3.7: 2D flowpath of E3 HPT.

After defining the reference hub and shroud contour, the detailed blading of three sections is conducted. During the blading there are several methods available. The point cloud data taken from Timko's report [11] is firstly applied to the commercial tool NREC AxCent, later, for the curvature smoothing the Pritchard Method is used for this thesis.

2D Blading process is involved in these following steps:

- Raw data extraction
- Section editing with Pritchard Method

- Fine tuning based on the results of 2D B2B CFD calculations

With help of Pritchard Method, blade section curvatures are smoothed on LE-Throat, Throat-TE and PS curves so the blade loadings have resulted in a good sense. Finally, minor adjustments have been made to find good radial equilibrium on blade aerodynamic outputs.

As a results, the blade profile data is generated as seen on **Table 3.6** and **Table 3.7**.

Table 3.6: Blade section wise geometrical parameters.

	Units	S1	R1	S2	R2	Design Choice
Inlet Tip Radius	[mm]	372.3	365.8	370.2	380.5	
Outlet Tip Radius	[mm]	365.8	366	380.5	381	
Inlet Hub Radius	[mm]	317.5	324.8	319	311.1	
Outlet Hub Radius	[mm]	325.8	323.3	312.2	311.2	
Tip Inlet Metal Angle	[°]	-5.41	24.54	-19.7	-9.02	
Mean Inlet Metal Angle	[°]	-8.66	36.8	-20.6	13.26	
Hub Inlet Metal Angle	[°]	-13.69	36.62	-12.22	30	
Tip Outlet Metal Angle	[°]	74.21	-64.5	67.61	-57.31	Direct Input
Mean Outlet Metal Angle	[°]	75.68	-62.8	65.34	-56.86	
Hub Outlet Metal Angle	[°]	71.75	-61.95	66.7	-56.3	
Unguided Turning tip	[°]	9.2	13	11.5	12	
Unguided Turning mean	[°]	8.4	13	11	15.5	
Unguided Turning hub	[°]	8.7	13	10.5	14.5	
LE radius Tip	[mm]	3.45	1.6	3.44	1.03	
LE radius Mean	[mm]	3.45	1.8	3.44	1.3	
LE radius Hub	[mm]	3.45	1.95	3.44	1.31	

While using the Pritchard Method, the blade inlet metal angles and inlet wedge angles are input. For the outlet section, blade gauge angle, exit wedge angle and unguided turning angle.

Table 3.7: Blade section wise geometrical parameters (cont).

	Units	S1	R1	S2	R2	
LE Wedge Angle Tip	[°]	110	70	65.52	85.3	Design Choice
LE Wedge Angle Mean	[°]	105	66	63.32	81.06	
LE Wedge Angle Hub	[°]	100	75	52.9	74.8	
TE Thickness Tip	[mm]	0.97	0.97	0.97	0.97	
TE Thickness Mean	[mm]	0.97	0.97	0.97	0.97	
TE Thickness Hub	[mm]	0.97	0.97	0.97	0.97	
TE Wedge Angle Tip	[°]	8	12.5	10	9	
TE Wedge Angle Mean	[°]	9.2	12.5	9.5	9	Direct Input
TE Wedge Angle Hub	[°]	10.2	12.5	9	9	
Stagger angle tip	[°]	59	-41.3	42.8	-47.6	
Stagger angle Mean	[°]	57	-37.2	42.5	-38.5	
Stagger angle hub	[°]	55	-32	41	-30	
Tmax/Chord tip		0.17	0.2	0.21	0.15	
Tmax/Chord mean		0.19	0.21	0.2	0.19	
Tmax/Chord hub		0.19	0.22	0.22	0.21	

The blade chord length was input for all rows. Finally, LE and TE thicknesses are defined. For design iteration, mostly stagger and TE gauge angle is adjusted.

The curvature of all rows are controlled by Bezier points as seen on **Figure 3.8**.

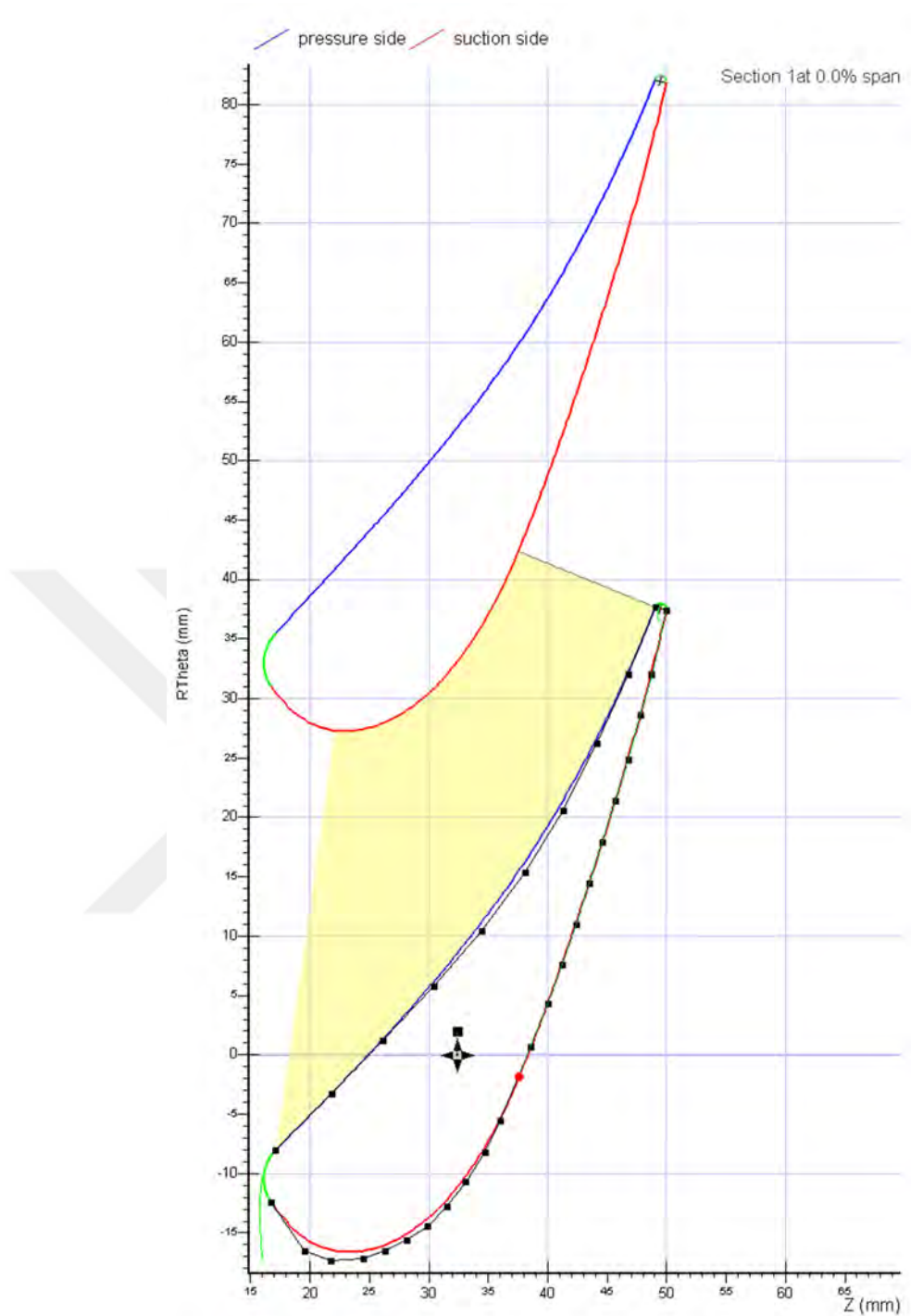


Figure 3.8: S1 hub section control point distribution as example.

After completing the blading of each spanwise section, the aerodynamic characteristics are evaluated using two-dimensional blade-to-blade (B2B) CFD analyses. Since the blade design is defined at three representative spanwise locations, the CFD calculations are primarily performed at these sections, namely the 10% span (hub), 51% span (mid-span), and 90% span (tip). These locations are selected to account for the

variation of blade loading and effective incidence along the span. So, blade profiles are iterated until reaching desired aerodynamic results.

The B2B calculation is conducted with structured OH grid type in which O grids are around the blade surface and H grids are distributed along the flow path.

During the meshing process, several criteria must be satisfied to ensure both numerical accuracy and computational efficiency. These criteria primarily include mesh quality metrics, near-wall resolution expressed by the non-dimensional wall distance y^+ , and the associated impact on solution time.

In this study, a structured meshing approach based on the B2B Mesher of AxCent was used with an OH-hybrid topology. Mesh quality was evaluated using commonly accepted metrics such as grid aspect ratio, maximum skewness angle.

The grid aspect ratio was treated differently in the near-wall and core-flow regions. High aspect ratios are unavoidable in the boundary-layer O-grid due to the need for fine resolution normal to the blade surface. In these regions, aspect ratios on the order of 100-400 were considered acceptable. In the H-grid passage and wake regions, the aspect ratio was kept below approximately 100 to limit numerical diffusion.

The maximum skewness angle was monitored throughout the domain. For most of the mesh, skewness values were kept below 70° . Slightly higher values, up to about 80° , were observed locally near the leading and trailing edges, where strong grid curvature is present.

The near-wall mesh resolution was assessed using the non-dimensional wall distance y^+ , defined as

$$y^+ = \frac{\rho u_\tau y}{\mu} \quad (3.1)$$

Where ρ is the fluid density, μ is the dynamic viscosity, y is the distance from the wall to the center of the first cell, and u_τ is the friction velocity,

$$u_\tau = \sqrt{\frac{\tau_w}{\rho}} \quad (3.2)$$

In the present simulations, the first cell height was selected to maintain y^+ values below 5 along the blade surfaces. This resolution allows the viscous sublayer to be adequately resolved within a wall-resolved RANS framework, without relying on wall-function assumptions. To ensure consistency across the grid sensitivity study, the near-wall O-grid topology and resolution were kept unchanged for all mesh levels, while refinement was applied primarily in the H-grid region to assess the influence of passage resolution on the solution.

The mesh details can be seen from **Table 3.8** and grid density can be seen from **Figure 3.9**.

Table 3.8: Mesh metrics.

Mesh Details	S1	R1	S2	R2
Circumferential Nodes	209	129	141	129
O Block	21	21	21	21
Total H Block	96	96	100	96
Relative Thickness	0.1	0.12	0.15	0.15
Y^+	3.19	2.79	2.8	2.23
Max Skewness Angle [°]	75	70	69	60
Max Grid Aspect Ratio	92	154	110	115

The mesh density can be seen as;

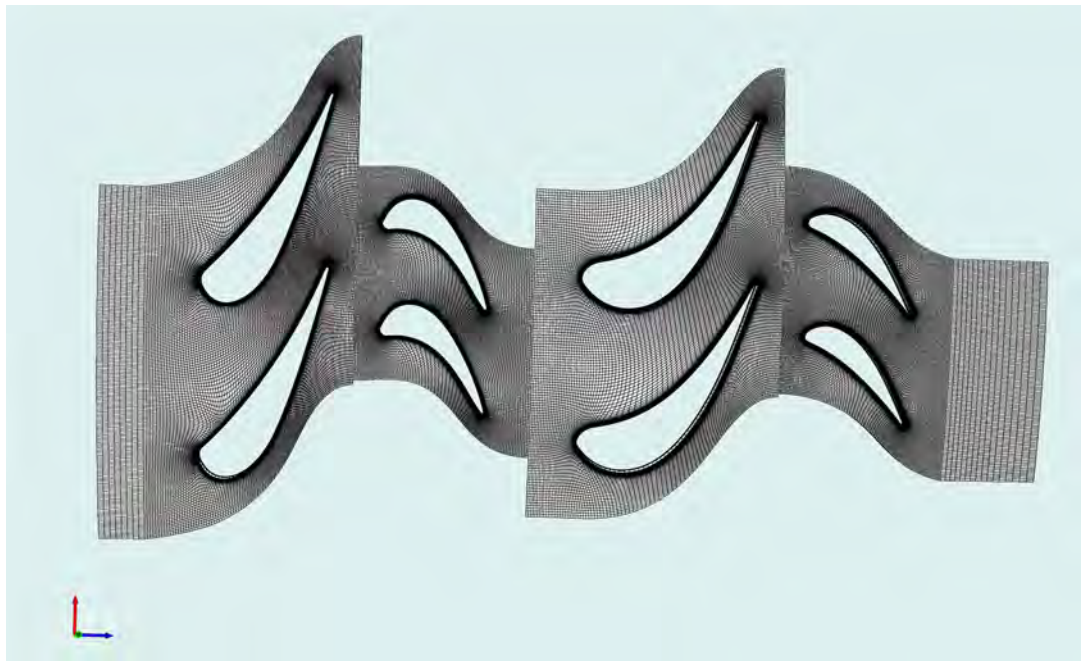


Figure 3.9: Grid distribution.

The O grid distribution is determined by several CFD calculations with observing boundary layer behavior of solution. The grid distribution can be seen from **Figure 3.10**.

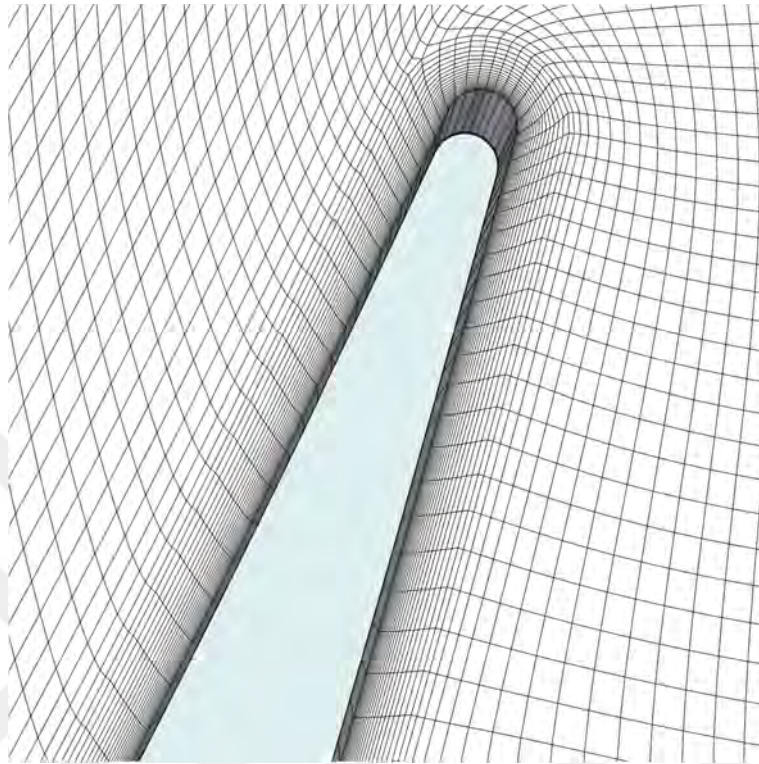


Figure 3.10: O grid distribution.

The boundary layer resolution can be seen from **Figure 3.11**.

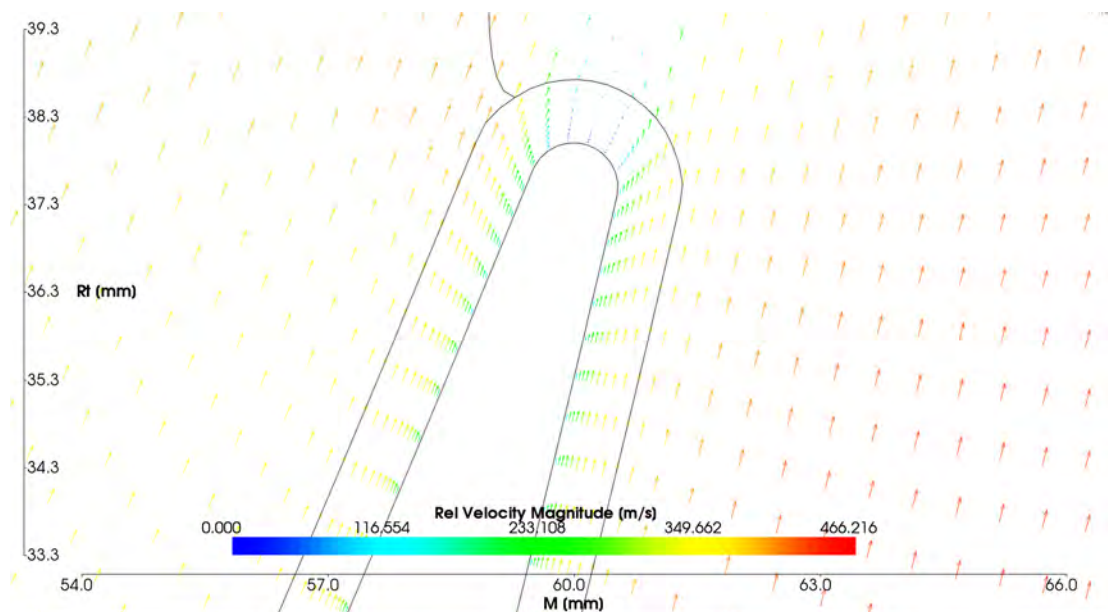


Figure 3.11: Boundary layer of S1 TE region.

Local mesh quality deviations were confined to small regions and were found to have no significant impact on the solution.

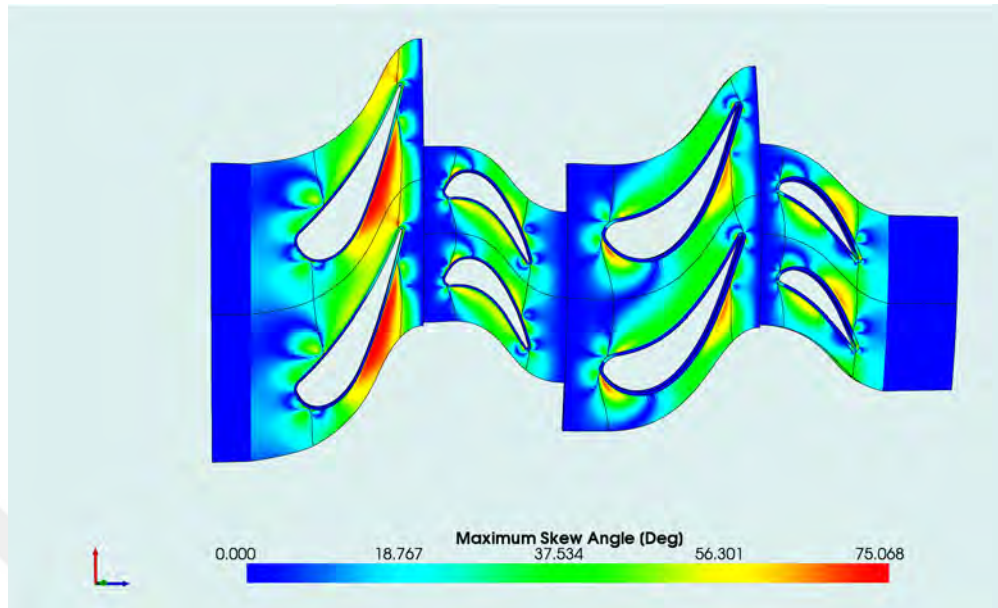


Figure 3.12: Mesh maximum skew angle distribution [°].

Overall mesh quality contours can be seen from **Figure 3.12** and **Figure 3.13**.

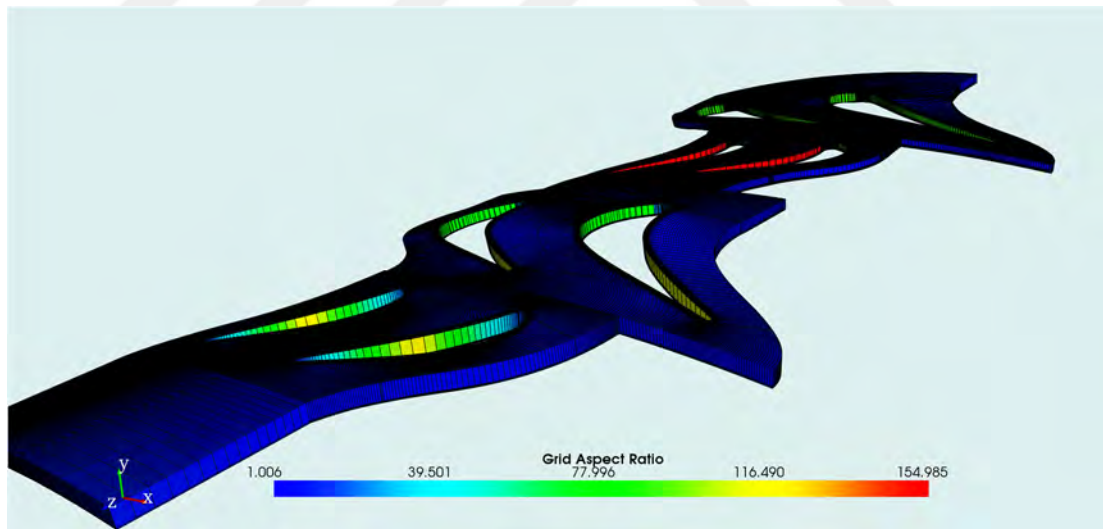


Figure 3.13: Grid aspect ratio distribution.

Based on the grid quality metrics presented above, a mesh dependency study was conducted to assess the sensitivity of the numerical solution to grid resolution.

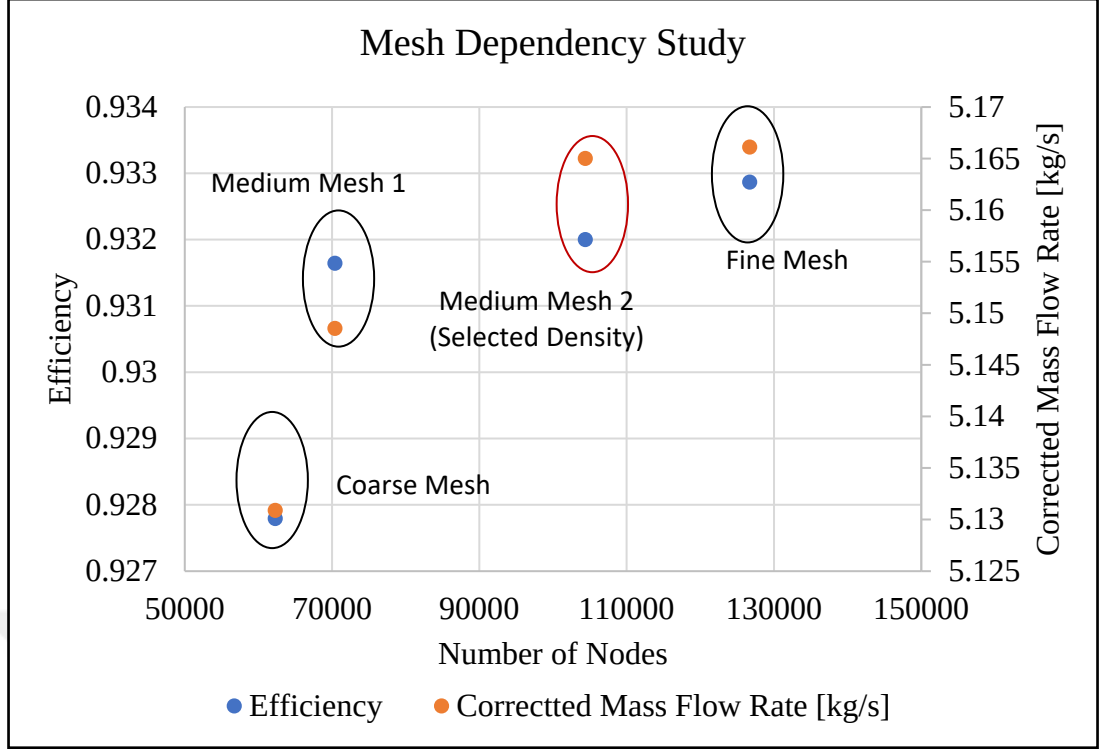


Figure 3.14: Mesh dependency study.

Figure 3.14 illustrates the mesh dependency of the predicted efficiency and corrected mass flow rate as a function of the total number of nodes. The results show that both quantities approach asymptotic values beyond the Medium Mesh 2 density, which is therefore selected as a suitable compromise between numerical accuracy and computational cost.

The solver setup follows the formulation described in Chapter 2 and is applied here within a Blade-to-Blade (B2B) CFD framework. A steady SST 2 equation turbulence model, integrated to the wall using a low-Reynolds-number treatment consistent with the $y^+ < 5$ resolution. Convective fluxes are discretized using a third-order upwind AUSM scheme, and convergence is accelerated using a two-level multigrid approach. Each operating point is iterated for a minimum of 250 iterations, and convergence is assessed based on residual reduction and stabilization of key aerodynamic quantities.

3.3. Validation Results

In this section, the aerodynamic performance of the reconstructed NASA E3 high-pressure turbine is validated against available experimental data and published numerical results. The comparison focuses on global performance parameters, namely

thermodynamic efficiency and corrected mass flow rate, evaluated over the design and selected off-design operating conditions.

In this thesis, the cooled turbine efficiency definition proposed by Timko et al. [11] is adopted. The efficiency is defined as the ratio of the actual work extracted by the turbine to the ideal isentropic work potential of the combined core and cooling flows,

$$\eta_{TH} = \frac{\dot{m}_{total} \Delta H_{turbine}}{(\dot{m}_{41} \Delta H_{is} + \sum \dot{m}_{cool} \Delta H_{is,cool})} \quad (3.3)$$

In addition to efficiency, the corrected mass flow rate is used as a key validation parameter, as it enables consistent comparison between different operating points with varying inlet conditions. It is expressed as:

$$\dot{m}_{corr} = \dot{m} \frac{\sqrt{T_{0,in}/T_{ref}}}{P_{0,in}/P_{ref}} \quad (3.4)$$

Where T_{ref} and P_{ref} denote the reference total temperature and pressure, respectively.

The design boundary conditions are adopted from the NASA E3 experimental database and corresponding numerical reconstructions reported in the literature. While the original studies are reported using imperial units, all parameters are converted to SI units for consistency with the present work.

The solver setup is defined using prescribed inlet total pressure and temperature, outlet static pressure, and rotational speed. The resulting operating point closely matches the E3 design intent, with a pressure ratio of approximately 4.96, corresponding to RDG 12 in the NASA E3 test campaign [11].

Cooling flows are modeled using source-term injections, with mass flow rates and total temperatures summarized in **Table 3.9**. The locations of cooling injection are estimated based on the E3 Detailed Design Report [13].

Table 3.9: RDG 12 Cooling Flow Details.

Component	Total $\dot{m}$ (kg/s)	Total Temperature Range, $T_{0,c}$ (K)
Stator 1	0.8459	326.39 K – 352.19 K
Rotor 1	0.6407	345.28 K
Stator 2	0.122	356.43 K
Rotor 2	0.0344	345.28 K

The validation procedure is carried out in multiple steps to ensure consistency at both local and global levels.

First, the blading of the benchmark and analyses at the engine design condition has been made to avoid Reynolds-number effects associated with aerodynamic scaling. Using boundary conditions taken from cycle of Claus et al. [12]. Under these conditions, spanwise distributions of flow angles and Mach numbers at the hub, mean, and tip sections are compared against the axisymmetric reference results provided by Timko et al. [11]. These comparisons are documented in Appendix A.

The comparison of blade loading distributions obtained from the present B2B simulations with those reported in the NASA E3 experiments [11] is shown in **Figure 3.15**, **Figure 3.16** and **Figure 3.17**. The results demonstrate good agreement in terms of suction-side acceleration and pressure-side diffusion trends. Minor deviations observed near the leading and trailing edges are attributed to the quasi-two-dimensional B2B assumption, the absence of end wall effects, and uncertainties in the exact locations of the cooling holes.

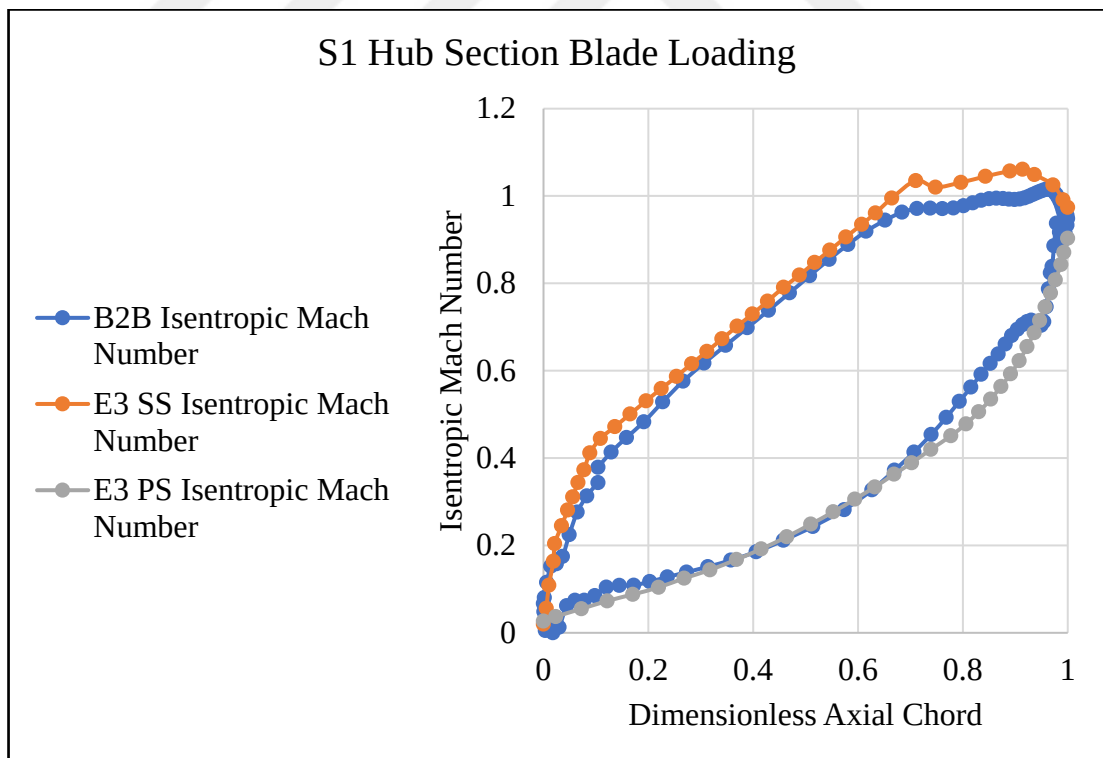


Figure 3.15: S1 hub section blade loading.

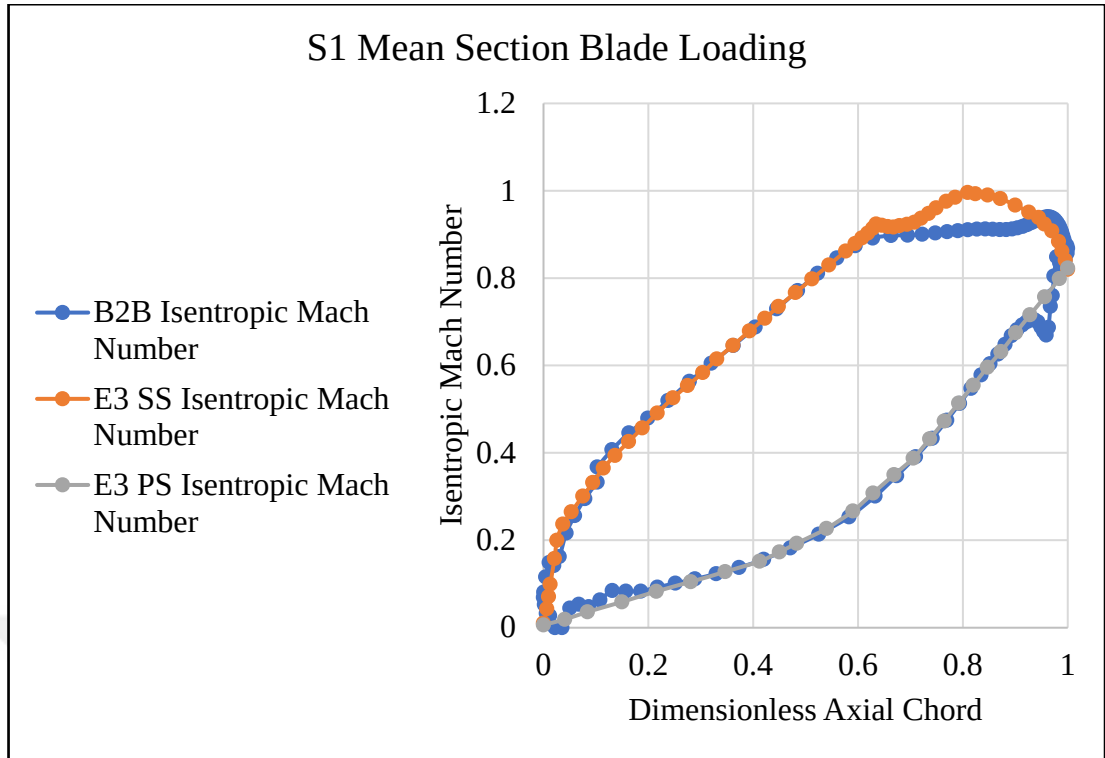


Figure 3.16: S1 mean section blade loading.

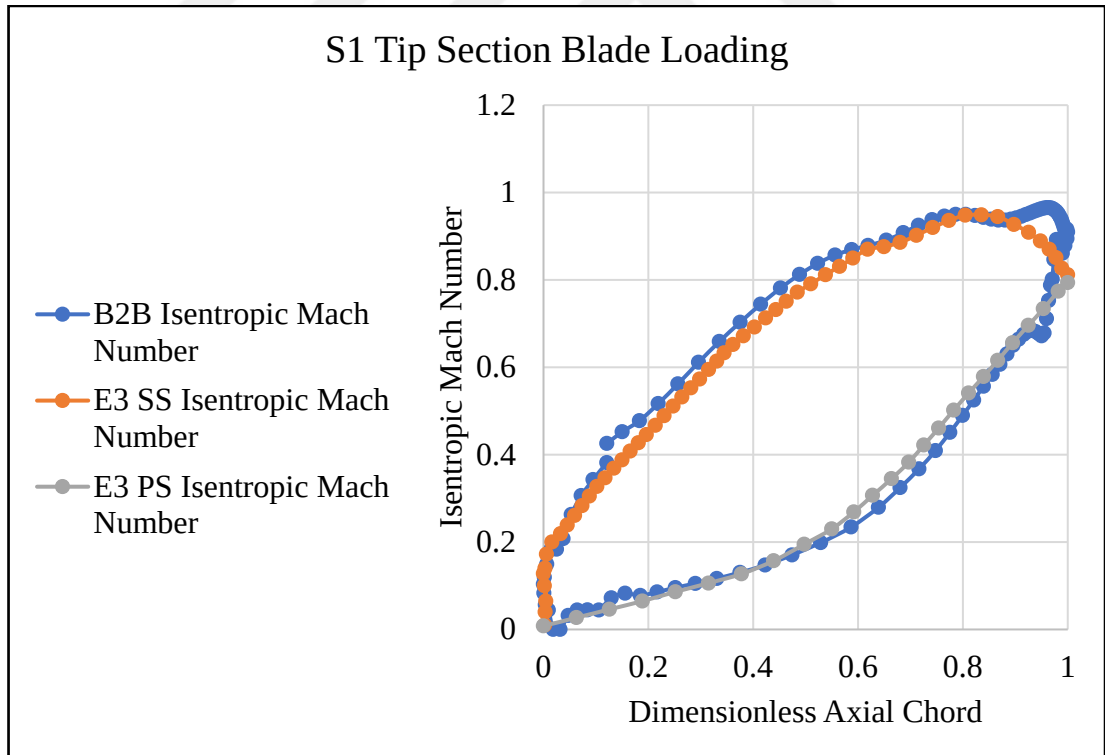


Figure 3.17: S1 tip section blade loading.

For all sections except the hub, flow behavior at front side is well captured. For the hub section flow isentropic Mach number is slightly low at 0.1 axial chord location

than E3 report [11]. This might be because of unclear hub-to-shroud distribution of cooling flow. Also, flow accelerates less than NASA E3 data at throat section for hub and mean span. Even though it is seen that required mass flow rate at stator exit is matched, minor adjustments may be considered for throat region.

Figure 3.18, Figure 3.19 and Figure 3.20 present the relative Mach number contours at the hub, mean, and tip sections under engine design point conditions. The contours confirm physically consistent acceleration and diffusion patterns, supporting the validity of the reconstructed geometry and numerical setup.

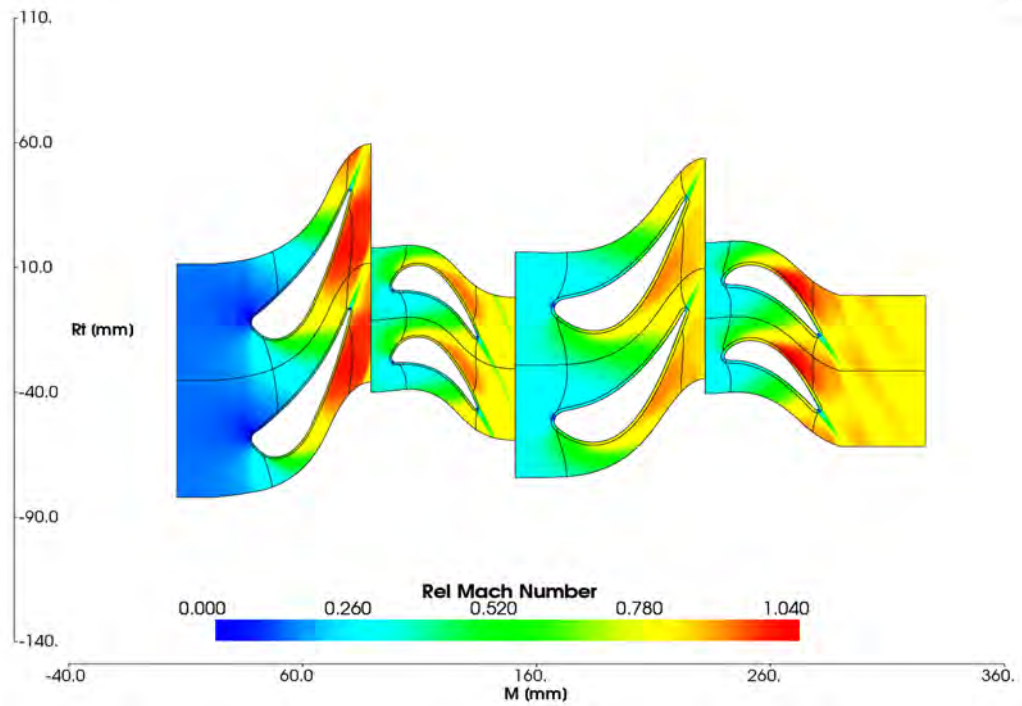


Figure 3.18: B2B hub section rel Mach number contour at EDP.

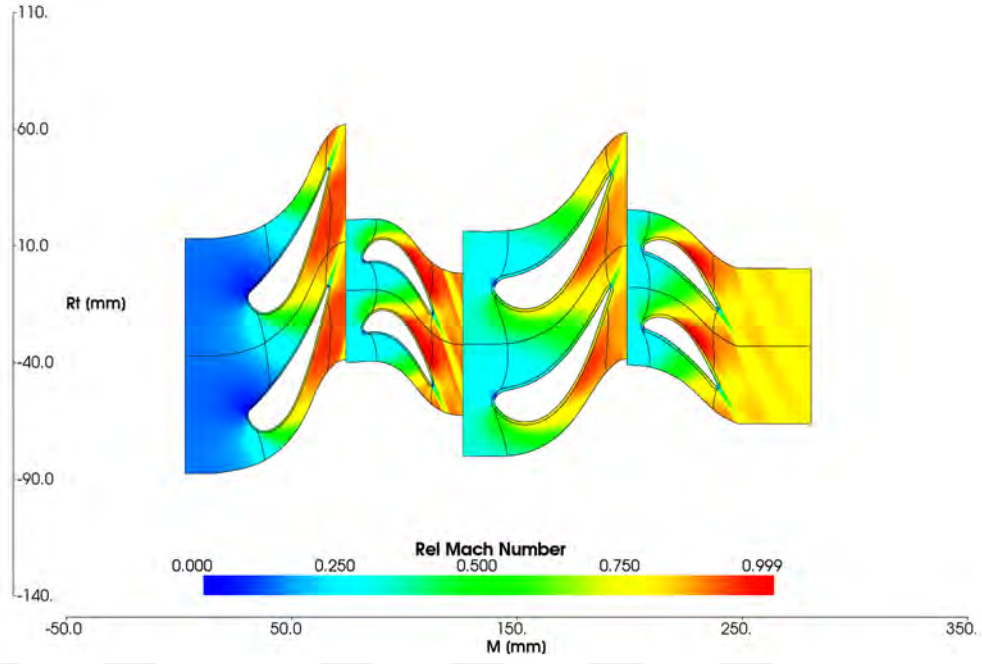


Figure 3.19: B2B mean section rel Mach number contour at EDP.

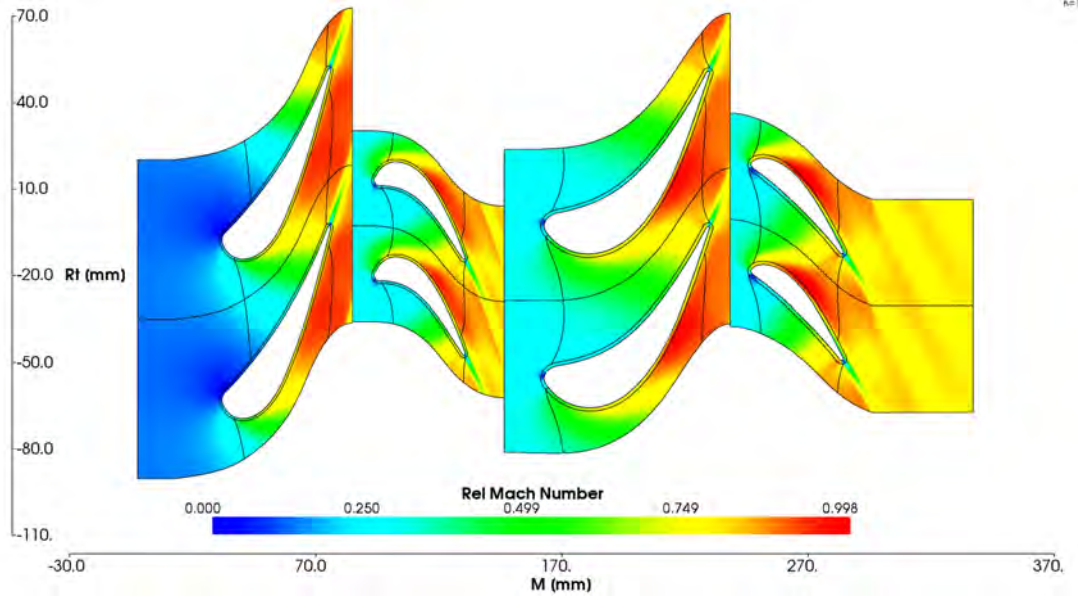


Figure 3.20: B2B tip section rel Mach number contour at EDP.

Following geometric and kinematic validation, global performance parameters are evaluated at the design point and selected off-design conditions. Experimental measurements from the NASA E3 database [11] and published numerical results by Murari et al. [14] are used as reference benchmarks.

Since the Blade-to-Blade (B2B) simulations provide solutions at discrete spanwise locations (hub, mean, and tip), a weighted numerical integration approach is employed to reconstruct the overall stage performance.

Rather than averaging local efficiency directly, the actual extracted power and the ideal work potential are integrated separately across the span in order to preserve energy consistency. The stage-averaged thermodynamic efficiency is then obtained from the ratio of the integrated quantities. This approach ensures that the reconstructed efficiency reflects the correct contribution of each spanwise section.

The comparison of corrected mass flow rate and thermodynamic efficiency is presented in **Figure 3.21** and **Figure 3.22** and **Table 3.10** and **Table 3.11**. The B2B predictions show good agreement with experimental data and published numerical results over a wide range of operating conditions.

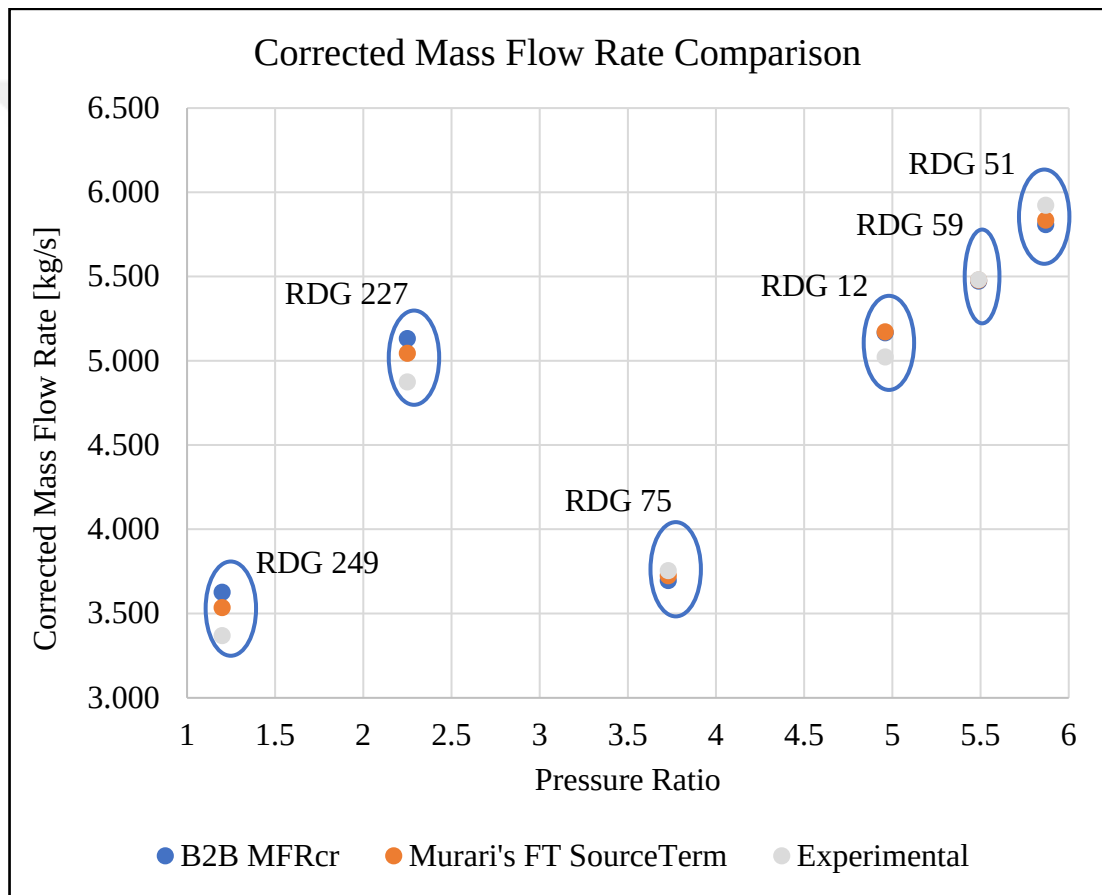


Figure 3.21: Corrected mass flow rate comparison chart.

In addition to the graphical comparison, the corrected mass flow values are reported in tabulated form to enable a direct point-by-point assessment. Table 3.10 lists the operating points (RDG cases) together with the corresponding pressure ratios, allowing small deviations between datasets to be quantified. As also indicated by Figure 3.21, the B2B predictions closely follow both the experimental trend and the

published FineTurbo reference across the full pressure-ratio range. The remaining differences are generally modest and show no systematic bias with operating condition.

Table 3.10: Corrected mass flow rate comparison table.

Case No.		Pressure Ratio [PR]	Test Data $\dot{m}_{\text{corr}}$ [kg/s]	FineTurbo Source Term $\dot{m}_{\text{corr}}$ [kg/s]	B2B $\dot{m}_{\text{corr}}$ [kg/s]
Design Pt	RDG 12	4.96	5.022	5.171	5.165
Test Pt-1	RDG 51	5.87	5.922	5.834	5.807
Test Pt-4	RDG 227	2.25	4.874	5.044	5.131
Test Pt-6	RDG 249	1.2	3.368	3.535	3.626
Test Pt-8	RDG 75	3.73	3.754	3.724	3.695
Test Pt-10	RDG 59	5.49	5.481	5.478	5.472

And the efficiency comparison is given in;

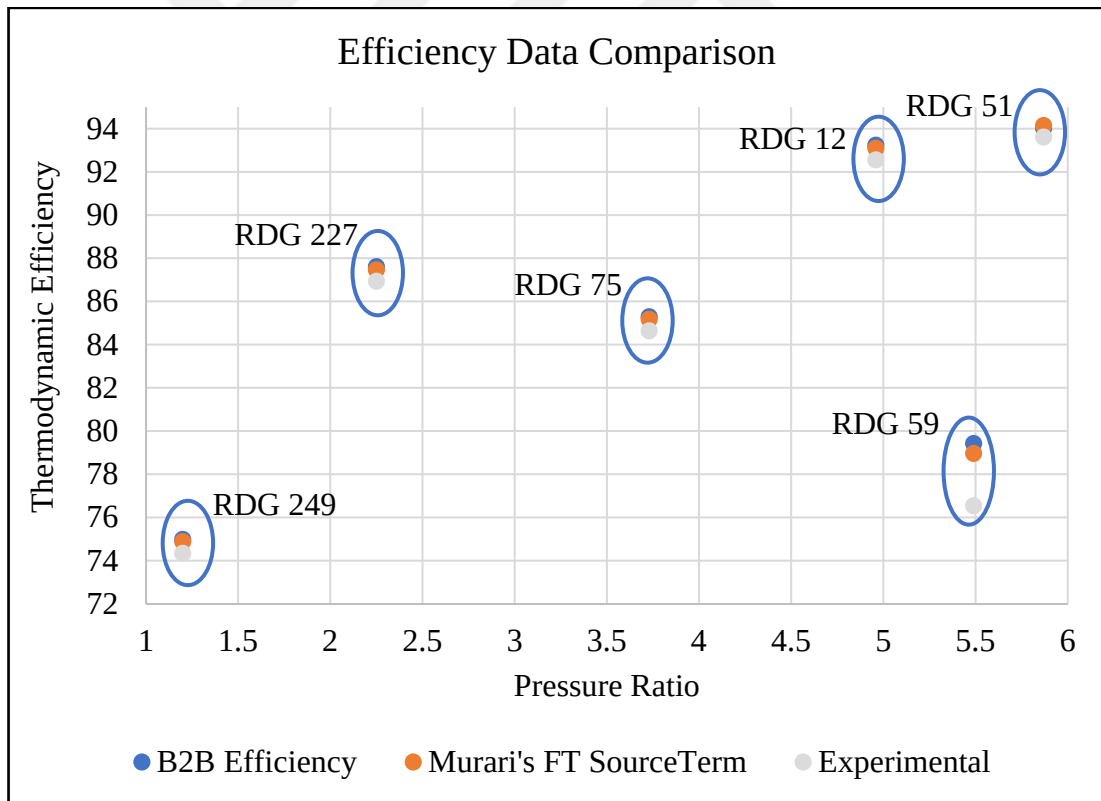


Figure 3.22: Efficiency data comparison chart.

A similar agreement is observed for the stage-averaged thermodynamic efficiency. Figure 3.22 illustrates the efficiency trend with pressure ratio, while Table 3.11 provides a point-by-point comparison for each RDG operating condition. Overall, the

B2B results capture the relative levels and variations of efficiency and remain close to both the experimental data and the FineTurbo source-term reference. Any remaining differences are discussed below in the context of the modelling assumptions..

Table 3.11: Efficiency data comparison table.

Case No.		Pressure Ratio [PR]	Test Data η_{th}	FineTurbo SourceTerm η_{th}	B2B Results η_{th}
Design Pt	RDG 12	4.96	92.55	93.09	93.23
Test Pt-1	RDG 51	5.87	93.61	94.14	94.01
Test Pt-4	RDG 227	2.25	86.93	87.461	87.61
Test Pt-6	RDG 249	1.2	74.34	74.88	74.98
Test Pt-8	RDG 75	3.73	84.64	85.17	85.29
Test Pt-10	RDG 59	5.49	76.54	78.96	79.42

The remaining discrepancies between the B2B results and experimental data are within the expected range for quasi-3D simulations and are primarily associated with the absence of secondary flows, tip-leakage effects and full three-dimensional mixing losses.

Finally, static pressure distribution from stage 1 inlet to stage 2 outlet along the hub and tip section is observed to validate pressure reactions of each stage. The static pressure distribution at RDG 12 versus B2B calculation can be seen at **Figure 3.23**.

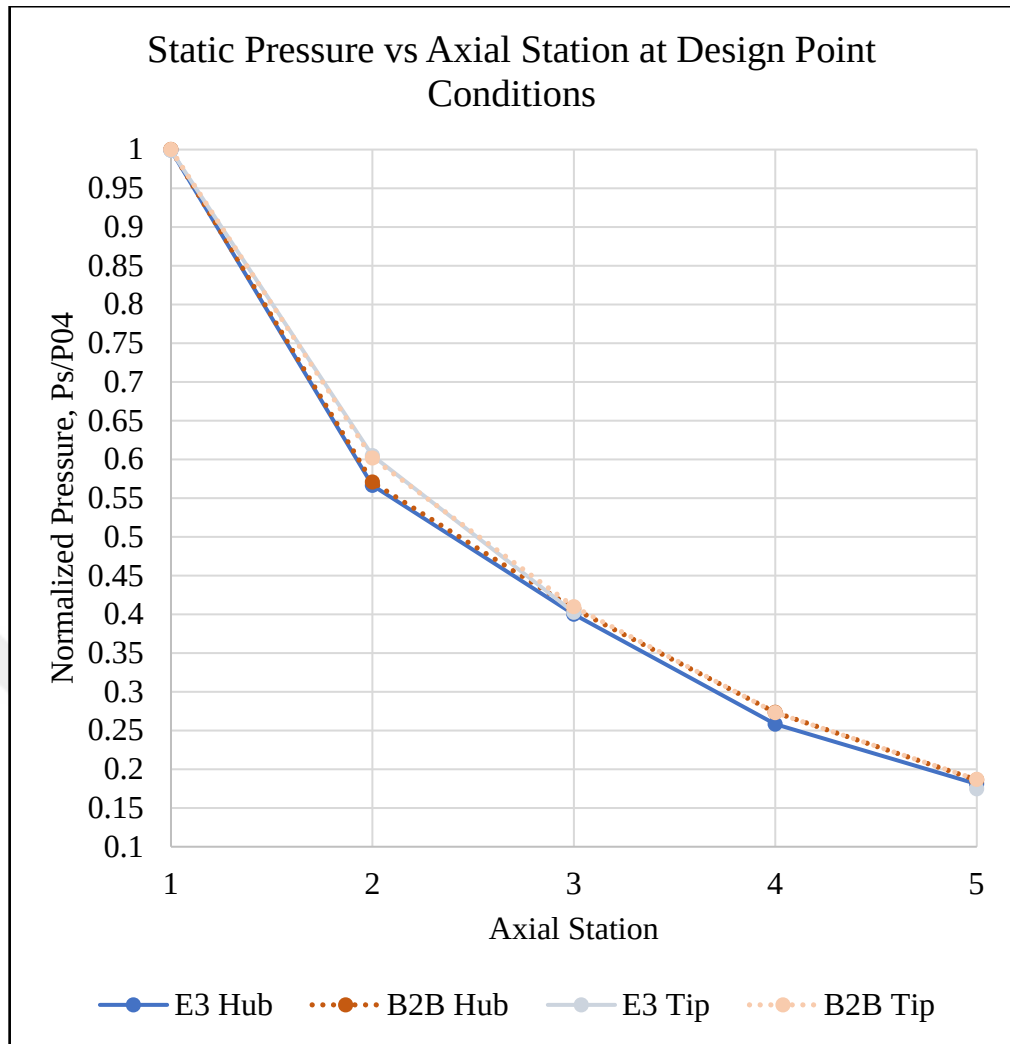


Figure 3.23: Static pressure vs axial station at design point conditions.

The normalized static pressure at tip and hub section both are relatively close to the experimental data at S1 outlet and R1 outlet, which means there is a good agreement at pressure reaction at stage 1.

For stage 2, there are minor deflections such at R2 outlet. This is caused by mainly adjusting the static outlet pressure to reach the desired total-to-total pressure ratio. Also probed pressure value can vary a little depending on reading location.

4. MID FRAME DESIGN CONCEPT & CONFIGURATIONS

The present chapter introduces a parametric exploration study inspired by mid frame concept. The objectives are to investigate how a mid-frame responds to both geometric and operating-condition variations and to quantify the resulting trends using a systematic Design of Experiments (DoE) approach.

4.1. Transition from Stator to Mid Frame

The stator of the second stage of the validated HPT baseline is redesigned as strut vanes. While redesigning, instead of performing a full three-dimensional coupled analysis, the problem is reduced to a 2D B2B CFD to enable wide parametric changes. This modeling choice targets the first-order sensitivities of profile losses and flow turning with respect to controlled changes in geometry and inlet conditions.

Two configurations are considered:

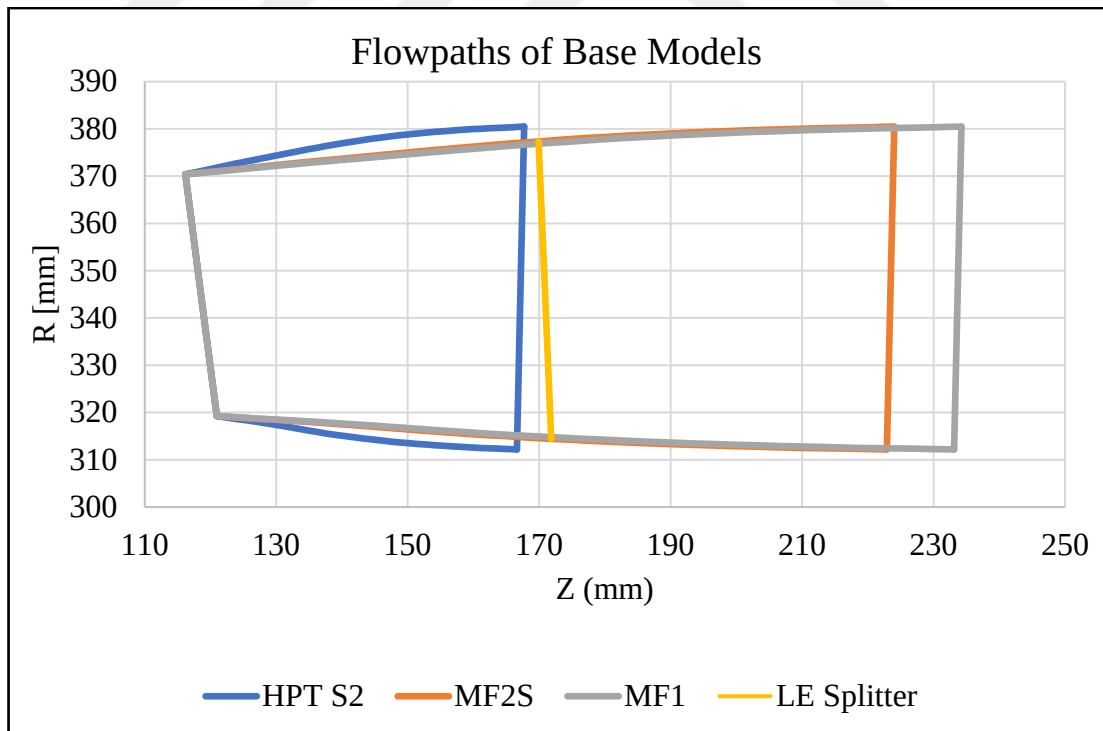
1. MF1: A single main vane row designed to deliver a predetermined turning while satisfying geometric constraints. This configuration serves as the reference for evaluating the performance penalty introduced by the mid frame structural requirements.
2. MF2S: Second configuration in which splitter vane in between mid-frames is designed while aiming to redistribute loading and modify wake characteristics.

Geometric parameters, flow paths and mean blade profiles are as shown at **Table 4.1**, **Figure 4.1** and **Figure 4.2** respectively.

Table 4.1: Geometrical parameters of base models.

Parameters	Units	HPT Second Stage Stator (S2)	MF1	MF2S	Splitter
C_{ax}	[mm]	49.05	115	105	52.5
H/C_{ax}		1.39	0.59	0.65	1.3
S/C_{ax}		0.924	0.498	0.768	
$\Delta\beta$	[°]	89.7	58	66	42.5
R_{LE}/C_{ax}		0.070	0.022	0.029	0.028
R_{TE}/C_{ax}		0.010	0.008	0.003	0.005
T_{max}/C_{ax}		0.282	0.228	0.21	0.22

As it seen on tables the aspect ratios are dramatically decreasing because of wide chord bodies, also S/C_{ax} the pitch to axial chord ratio is decreased to keep corrected mass flow rate like benchmark stator. Another important change is the blade turning angle $\Delta\beta$, to reduce boundary layer separation blade turning is also reduced. Finally, thickness values are moderately increased, but it seems to decrease because of wide chord effect. Typical trailing-edge thicknesses are kept, leading edge thicknesses are slightly increased, and maximum thickness is moderately increased.

**Figure 4.1:** Base flowpath comparison.

While designing base strut profiles the benchmark stator is used. The hub to tip ratio is kept constant for this study for sake of simplicity. The leading edge of splitter vane is located at the middle of passage and it's along to the end of passage with main strut.

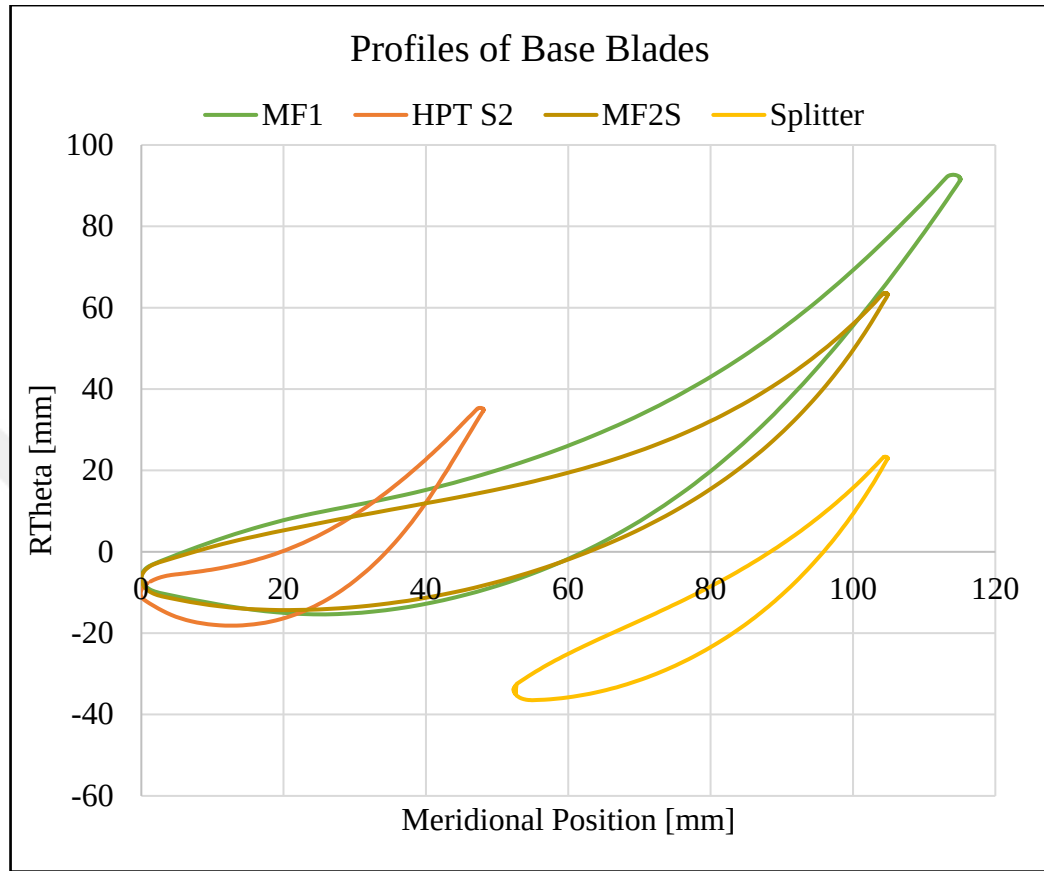


Figure 4.2: Profiles of base blades.

The strut and splitter profiles are generated with thickness and beta distribution control because Pritchard method is not robust for wide chord geometries and observed crashes during the geometry generation. The benefit of thickness and beta distribution control method is its capability to precisely manage the blockage distribution and local diffusion rates. Which is critical for controlling loading of blade and flow separations in low-aspect-ratio components. Strut blades are designed with aft loading with guidance to previous works on literature [31], [9].

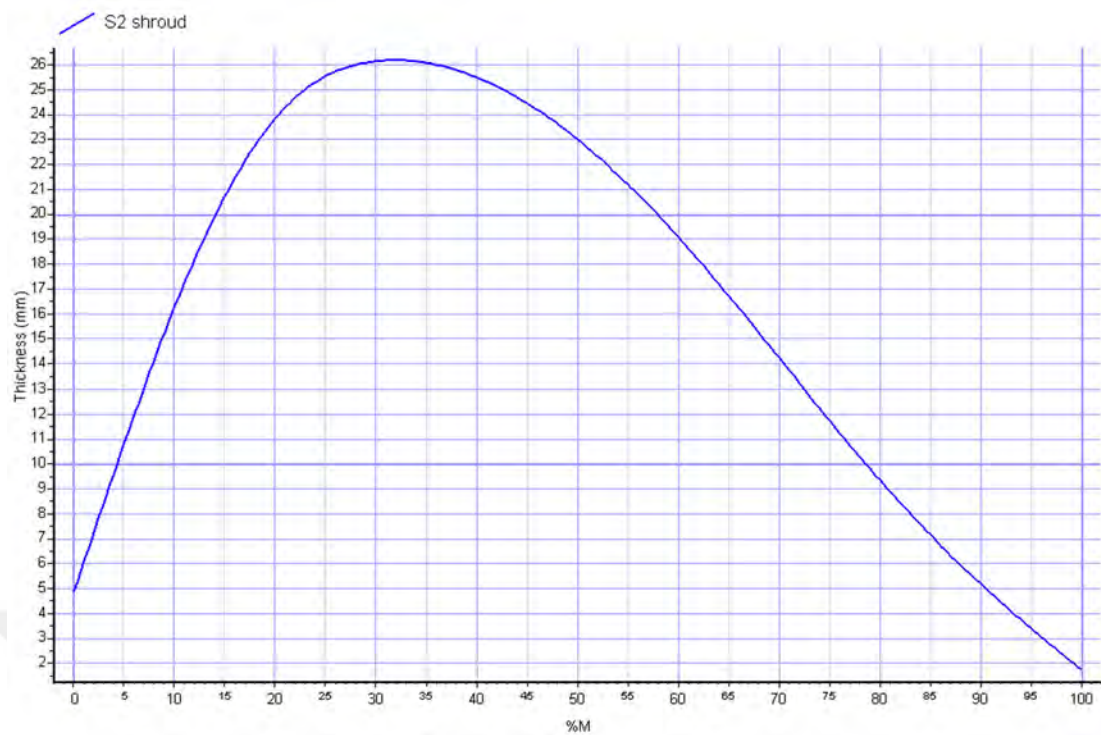


Figure 4.3: Thickness distribution of MF1.

The maximum thickness of MF1 before DoE is around 26 mm and the maximum thickness is positioned at 32% M. Leading-edge radius is 2.5 mm and trailing-edge radius is around 1 mm.

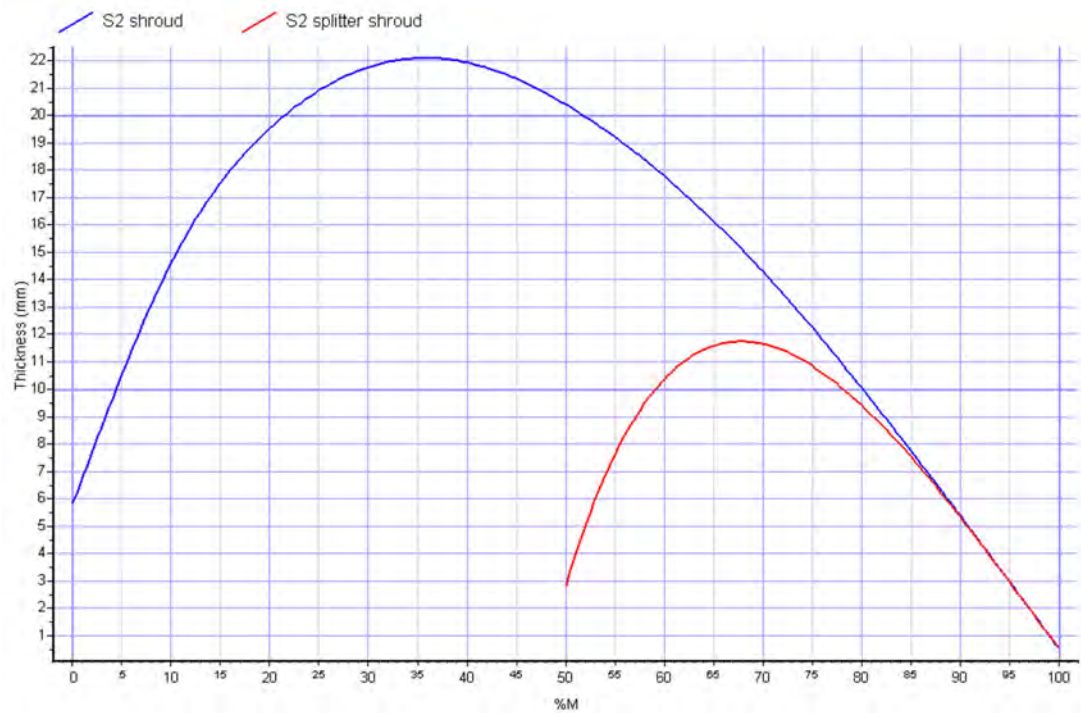


Figure 4.4: Thickness distribution of MF2S.

The maximum thickness of MF1 before DoE is around 22 mm. It is slightly reduced because of passage area distribution is changed with splitter and the maximum thickness is positioned at 35% M. Leading-edge radius is 3 mm while splitter has 1.5 mm leading-edge radius and maximum thickness is located at 37% of its relative meridional position. The unguided turning sections of both vanes have same thickness distributions in order to keep balanced passage area at the exit [32].

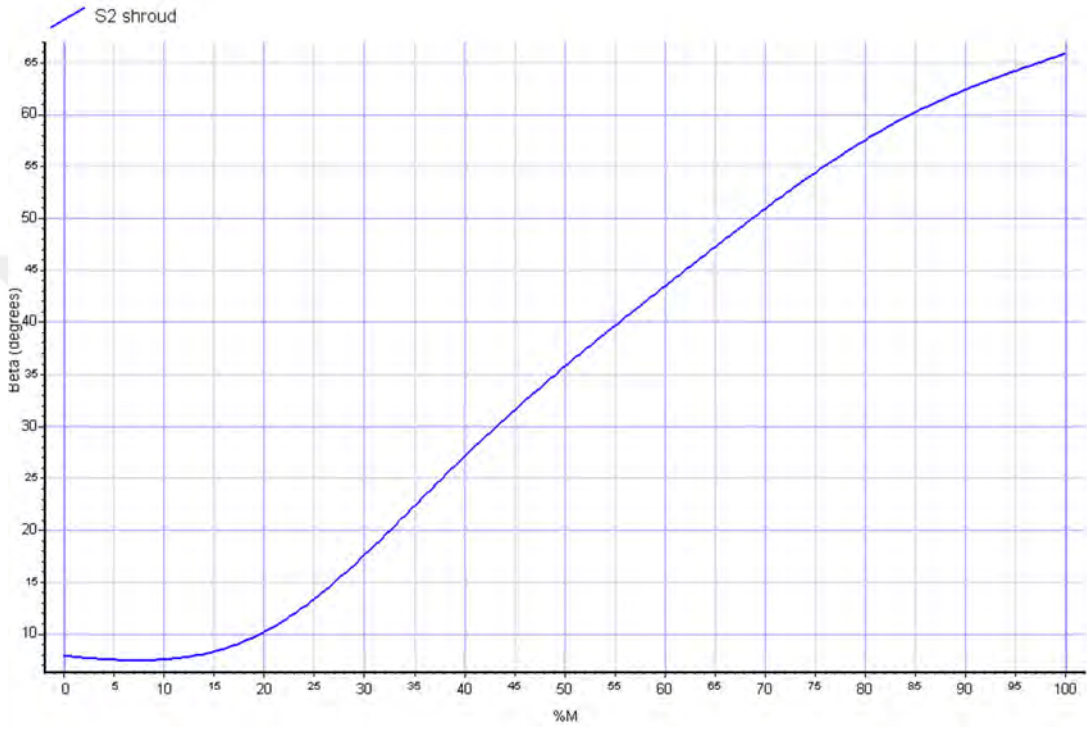


Figure 4.5: Beta distribution of MF1.

Here is shown the beta distribution of MF1 which indicates that most of the flow turning is achieved towards the mid and aft part of the passage, while the leading-edge region remains relatively mild. This is because it does not exceed loading limits as the rapid turning at front section might cause flow separation after maximum thickness location.



Figure 4.6: Beta distribution of MF2S.

Similar design intent has been shown for beta distribution of MF2S too. Loading of front region is relatively higher. In addition, the splitter modifies the local passage and therefore the beta trend becomes slightly different in the rear section. This redistribution helps to avoid excessive front loading, since a rapid turning close to the leading edge may increase the adverse pressure gradient and trigger separation even before the maximum thickness location.

4.2. Physical Model and Pre-Setups

While keeping the geometry realistic the design parameters are grouped into three sets to explore the design space efficiently: fixed parameters, independent design variables and dependent variables. This approach follows the adaptive geometry idea of Riebl et al [10] and helps to keep the optimization well-defined.

Fixed parameters define the constraints of analyses, such as the annular flow path boundaries and the structural limits such maximum thickness to axial chord ratio.

Independent design variables are the main degrees of freedom controlled by the optimization tool HEEDs. They represent the parameters that can be adjusted to observe characteristics of struts.

Dependent variables are computed automatically from the fixed constraints and the chosen independent variables. This is critical part of DoE study, to randomly generate feasible and logical designs that CFD can calculate a good set of logics for distribution of physical parameters. List of variables can be seen on **Table 4.2**.

Table 4.2: Fixed, independent and dependent variable list.

Fixed Parameters	Independent Variables	Dependent Variables
Hub & Shroud Contours	C_{ax}	Main Blade Beta Distribution
Mesh Density	Blade Count	Main Blade Thickness Distribution
Outlet Static Pressure	C_m	C_θ
	$\Delta\beta$	α_1
	Blade Outlet Angle	Splitter Beta Distribution
	P01	Splitter Thickness Distribution
	T01	
	Target Incidence	
	R_{LE}/C_{ax}	
	R_{TE}/C_{ax}	
	T_{max}/C_{ax}	
	$T_{max}\%M$	
	$R_{LE}/C_{ax,splitter}$	
	$T_{max}/C_{ax,splitter}$	
	$T_{max}\%M_{splitter}$	
	$\beta_{1,splitter}$	

In total there are more than 20 independent variables which are mostly controlling beta and thickness distributions for both main blade and splitter.

For both strut configurations runs are with the same inlet/outlet boundary conditions and identical numerical settings, which enables a direct comparison of loss trade-offs.

For parametric study, the operating conditions are derived from the validated baseline and re-parameterized using a small set of independent inlet parameters. The DoE includes controlled variations in:

- Stage pressure ratio (PR): Imposed through inlet total pressure and outlet static pressure,
- Target incidence: defined relative to the leading-edge metal angle.

- Inlet Mach Number: limited and calculated through the imposed inlet meridional velocity and calculated inlet tangential velocity.

To keep the inlet flow direction physically consistent, the tangential velocity component is not set independently. Instead, it is coupled to the meridional component so that the inlet velocity triangle satisfies the required incidence. Thus, the absolute inlet swirl angle α_1 is first determined by the inlet metal angle and the specified target incidence (i).

$$\alpha_1 = \beta_{k1} + i \quad (4.1)$$

Finally, the tangential velocity component C_θ is computed as a dependent variable derived from the independent meridional velocity C_m and the required flow angle:

$$C_\theta = C_m \tan(\alpha_1) \quad (4.2)$$

In this parametric setup, the incidence i and meridional velocity C_m serve as independent control factors, while C_θ is adjusted automatically. This formulation allows incidence and Mach Number related effects to be explored systematically without conflicting boundary-condition or redundantly increasing degrees of freedom.

For geometry parametrization, beta and thickness distributions are controlled by pre-defined functions. The beta distribution is defined by Bezier polynomial that ensures continuity and allows desired loading characteristics for the strut and splitter itself. Unlike, the splitter is highly dependent on the beta distribution of main blade to control uneven passage area distribution. Highly free splitter variables might cause an unequal passage area distribution in between two blades which can be beneficial to redistribute blade loadings or can be disadvantaged because of blockage effects which might cause shocks losses. Such an example can be seen at **Figure 4.7**.

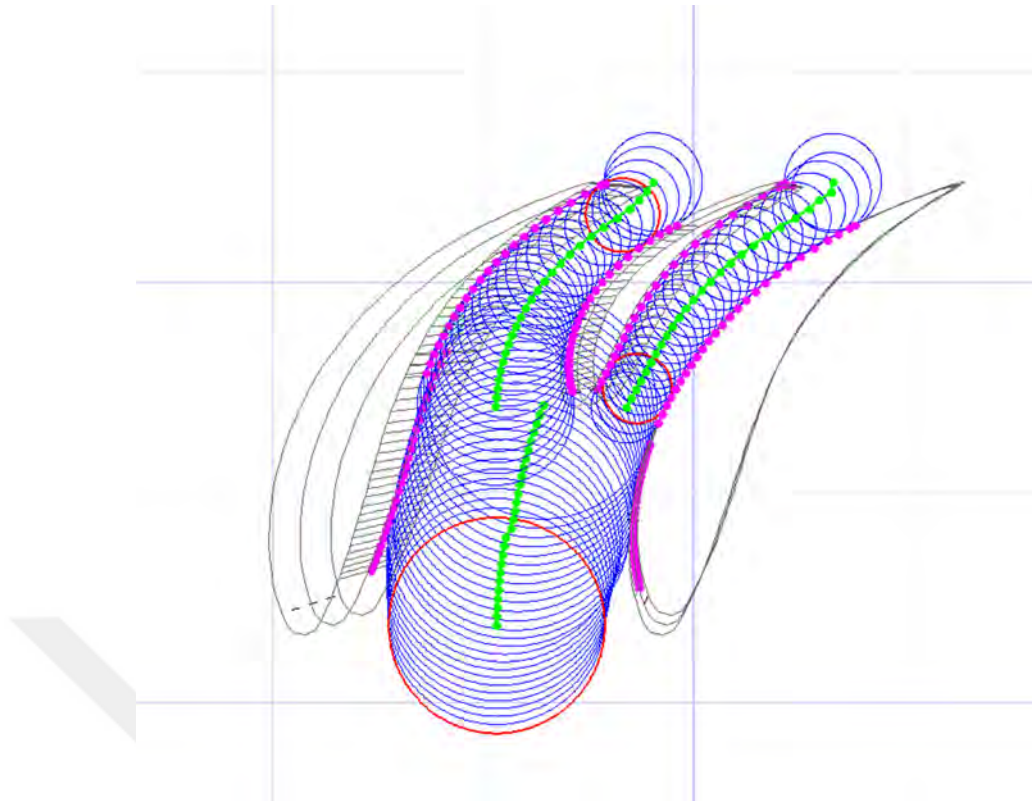


Figure 4.7: Uneven passage area distribution.

From **Figure 4.7** red circles indicate throat areas. Such aggressive channel designs would return with shock loss penalties thus front side of splitter is dependent on related main blade beta value.

The beta function requires the desired outlet metal angle and desired turning angle which modifies the inlet metal angle and coefficients of function as inputs. With help of systematic approach, “S” shaped camber or different types along the meridional position can be achieved.

On the other side the thickness distribution is controlled by asymmetric power law to characterize typical thickness monotonicity in between certain positions. Splitter thickness distribution is also controlled by same manner but with different modifiers to restrict any geometrical errors. The thickness function requires unidimensional leading edge and trail edge thicknesses, unidimensional maximum thickness and its position to build along the meridional position.

Final independent variable is blade count which directly affects the aerodynamic loadings of both strut and splitter. Pitch-to-chord ratio is derived from blade count input to observe its effect on aerodynamics of mid frame.

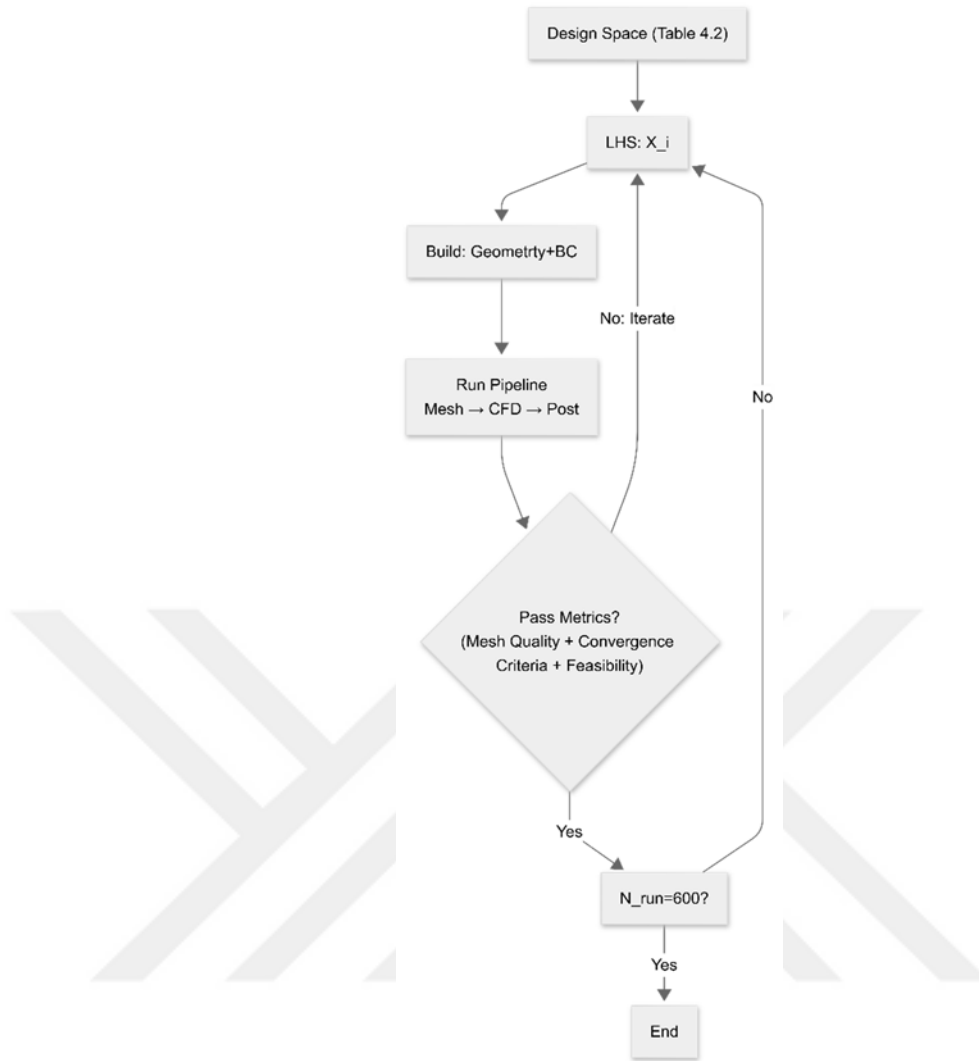


Figure 4.8: Flowchart of DoE study.

Following definition of parametric geometry, a Design of Experiments (DoE) study is formulated. Unlike, a direct optimization aimed at finding any local or global minimum, the primary purpose of this phase is to explore the design space, quantify the sensitivity of aerodynamics to geometrical variations and identify the trade-offs between metrics.

The characteristics of each candidate design is evaluated based on specific response variables. These outputs are recorded for each run in the DoE for post-process procedure.

The primary response variable is the Total Pressure Loss Coefficient Y_p , which shows how lossy the section is:

$$Y_p = \frac{P_{01} - P_{02}}{P_{01} - P_{s2}} \quad (4.3)$$

Where P_{01} and P_{02} are the mass-averaged total pressures at the strut inlet and outlet and P_{s2} is outlet static pressure.

In addition to the pressure loss coefficient, blading efficiency is monitored to assess profile performance, independent of the inlet boundary layer thickness.

$$\eta_b = \frac{V_1 - V_2}{V_1 - V_{is2}} \quad (4.4)$$

Other target responses are deviation angle (d) which represents the difference between outlet metal angle and outlet flow angle, turbine total-to-total efficiency and corrected mass flow rate.

$$d = \alpha_2 - \beta_{k2} \quad (4.5)$$

The workflows of DoE setup are same for both single strut and MF2S cases. Nondimensionalized input variables are changing within defined ranges by Latin Hypercube Sampling (LHS) strategy.

Because of the high dimensionality of the design space and the computational cost associated with CFD simulations, standard sampling techniques such Full Factorial designs are computationally unfavorable due to the curse of dimensionality. Similarly, Monte Carlo sample is prone to clustering, leaving large voids in the design space.

To overcome these limitations, the Latin Hypercube Sampling (LHS) method is employed. LHS is a stratified sampling technique that ensures a near-random but uniformly distributed coverage of each input variable.

The LHS algorithm divides the range of each design variable into N non-overlapping intervals of equal probability. Where N is the total number of design evaluations which is at least 500 for this study. The algorithm then selects exactly one sample from each interval. This process is repeated for all dimensions and the components.

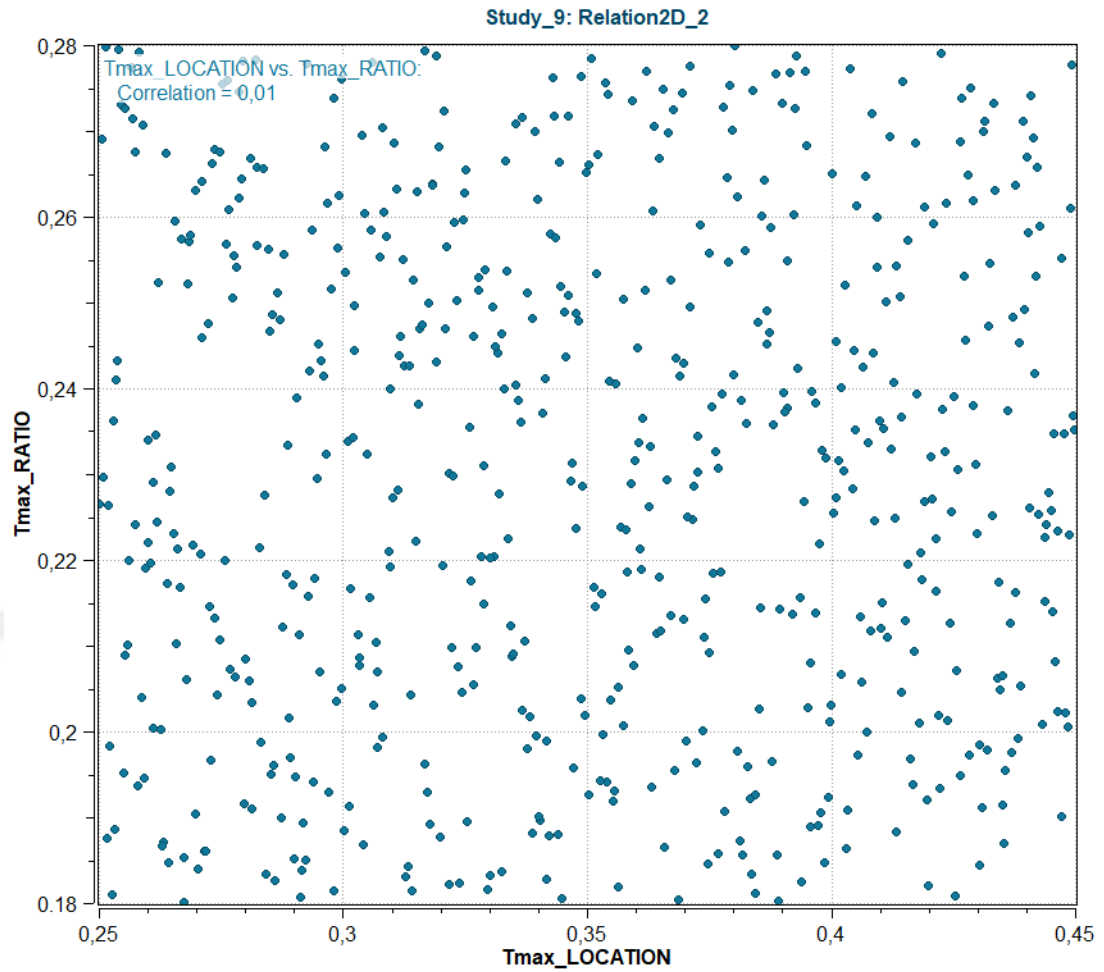


Figure 4.9: LHS variable distribution scatter plot.

Figure 4.9 shows how the LHS method works. The algorithm divides the design space into a hypercubic grid structure. The key rule is that there must be exactly one point in each row and each column. This prevents grouping of points, which is a common problem in simple random sampling. As a result, LHS ensures that the entire design space is covered evenly and efficiently.

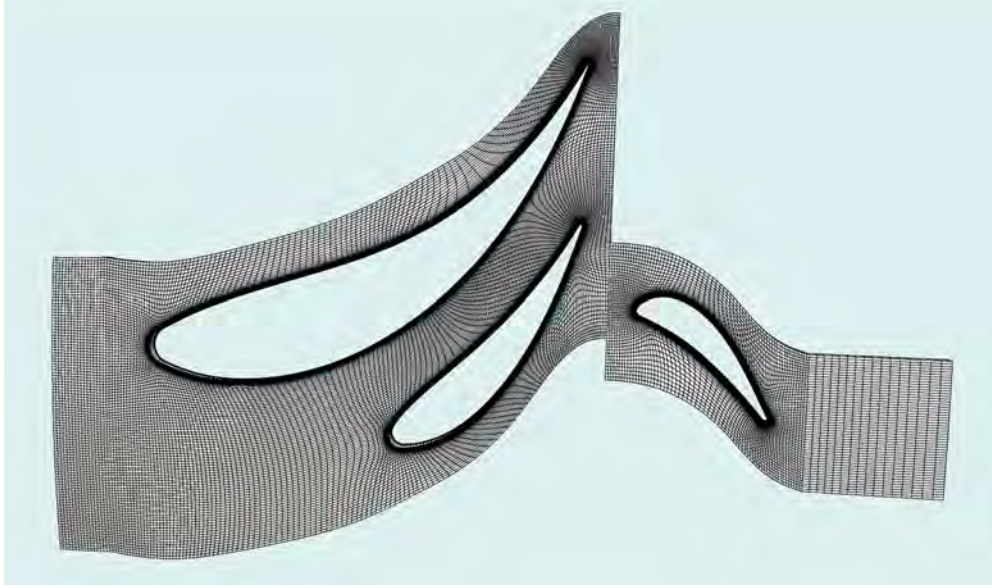


Figure 4.10: MF2S mesh.

For B2B solver both configurations are meshed with same R2 mesh density and differ at S1 circumferential nodes. The H grid is distributed evenly as 21 at first row and as 17 at the second row. The O grid number is 21 for row 1 and 17 for row 2 while the relative thickness is 0.11 and 0.15 for row 1 and row 2 respectively. The number of circumferential nodes for MF2S is 225 and for MF1 191, for row 2 it is 111.

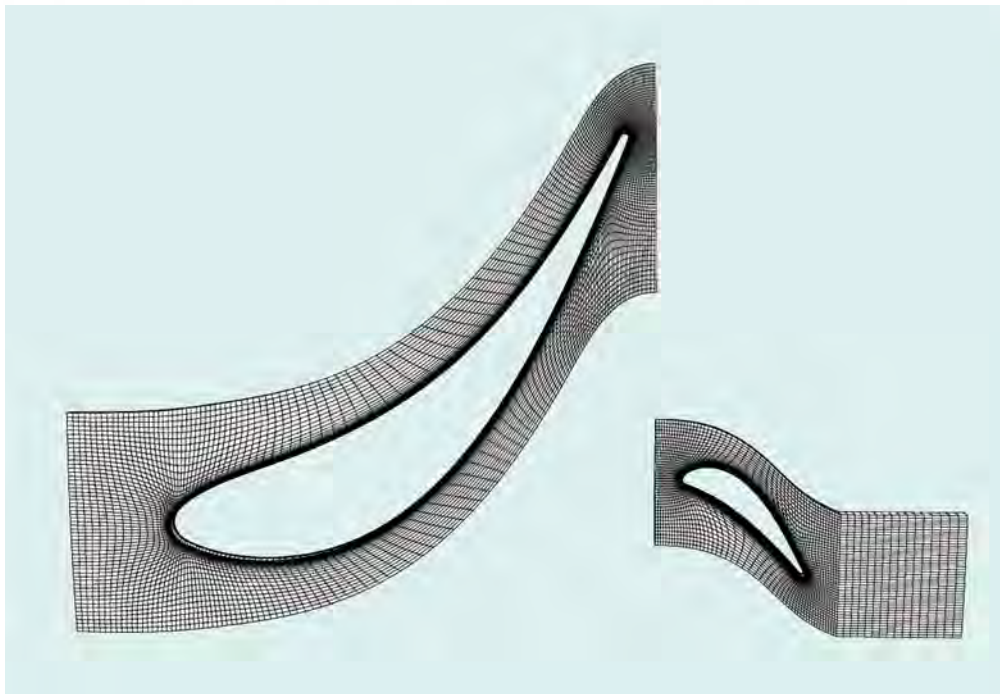


Figure 4.11: MF1 mesh.

4.3. Parametric Study (DoE)

For MF2S with splitter configuration 600 different iterations are ran. 511 iterations are successfully completed. The main reason for failed iterations is topology error during the meshing. After completing DoE runs infeasible results are outlined by constraints such minimum 50% efficiency limit and thickness limit. With that limit, the total number of designs is reduced to 500.

Same procedure is applied for configuration MF1 and no iteration failure has occurred with 500 runs. Also, during the post process none of designs are outlined by constraints.

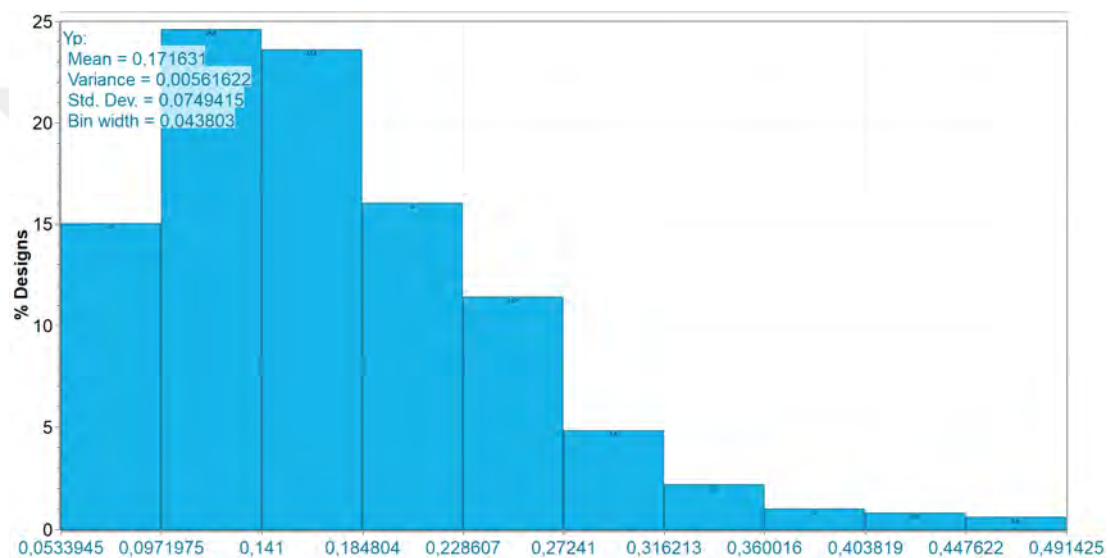


Figure 4.12: MF2S Y_p histogram.

At **Figure 4.12** is shown the histogram of Y_p for configuration MF2S. The mean loss is around 0.17 low standard deviation which is 0.075. The mean loss is relatively low due to nature of B2B calculation and variable ranges.

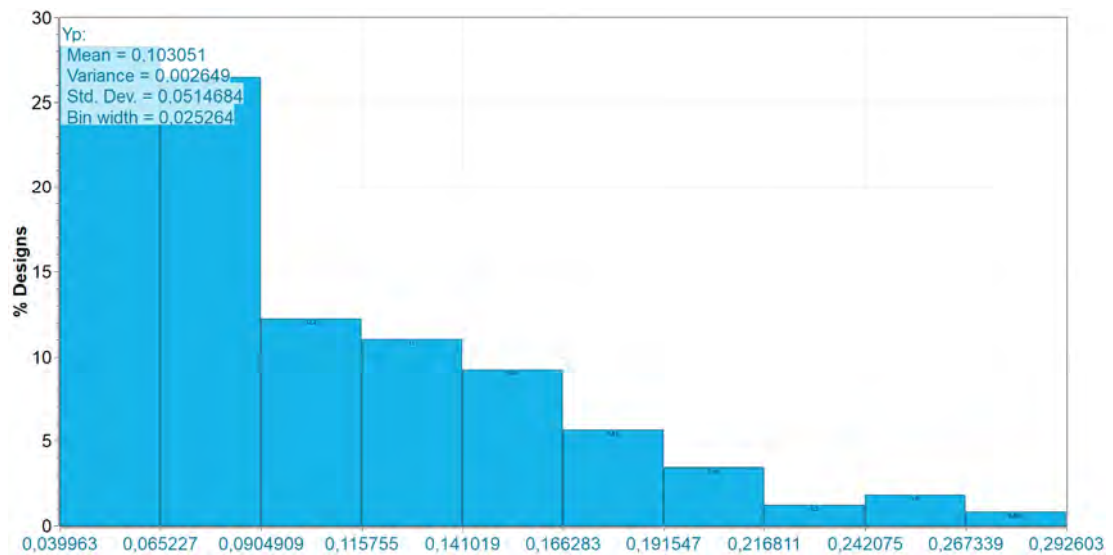


Figure 4.13: MF1 Y_p histogram.

Similar Y_p histogram for MF1 created at **Figure 4.13**. For this case, the mean Y_p is 0.1 with standard deviation of 0.05.

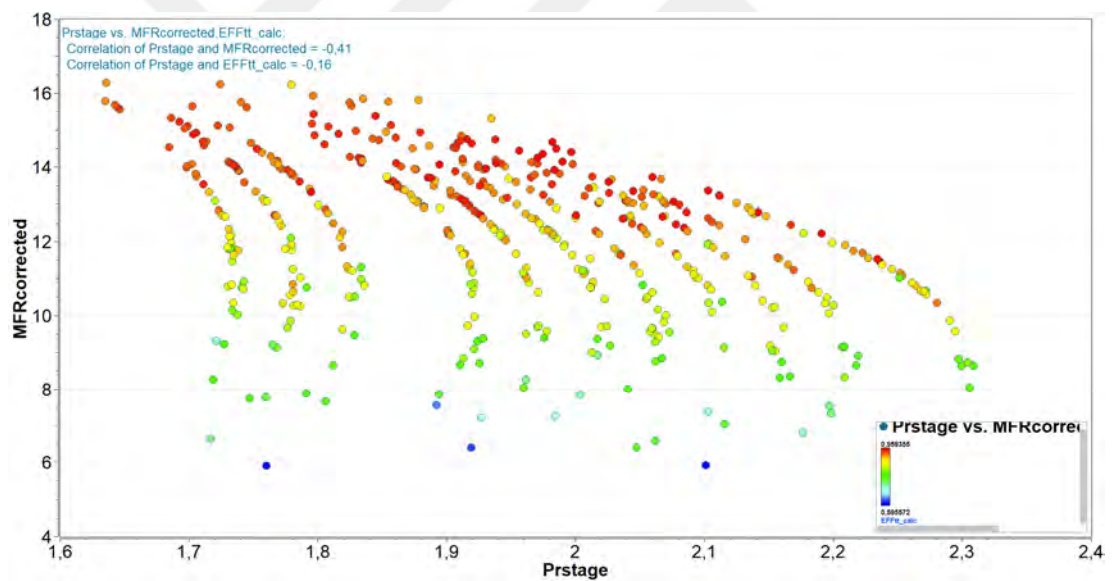


Figure 4.14: MF2S corrected mass flow rate versus pressure ratio.

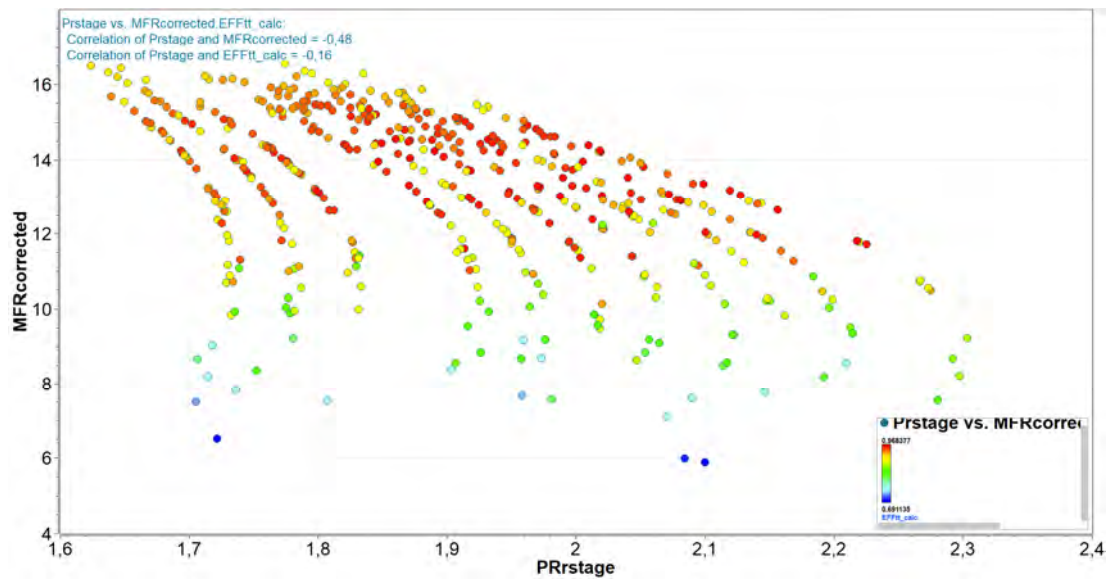


Figure 4.15: MF1 corrected mass flow rate versus pressure ratio.

At **Figure 4.14** and **Figure 4.15** stage pressure ratio against corrected mass flow rate is plotted and colored by stage total-to-total efficiency. Naturally, design points follow their work coefficient as seen on scatter plot. Observed that along the constants work coefficient line, as the pressure ratio decreases the corrected mass flow rate increases.

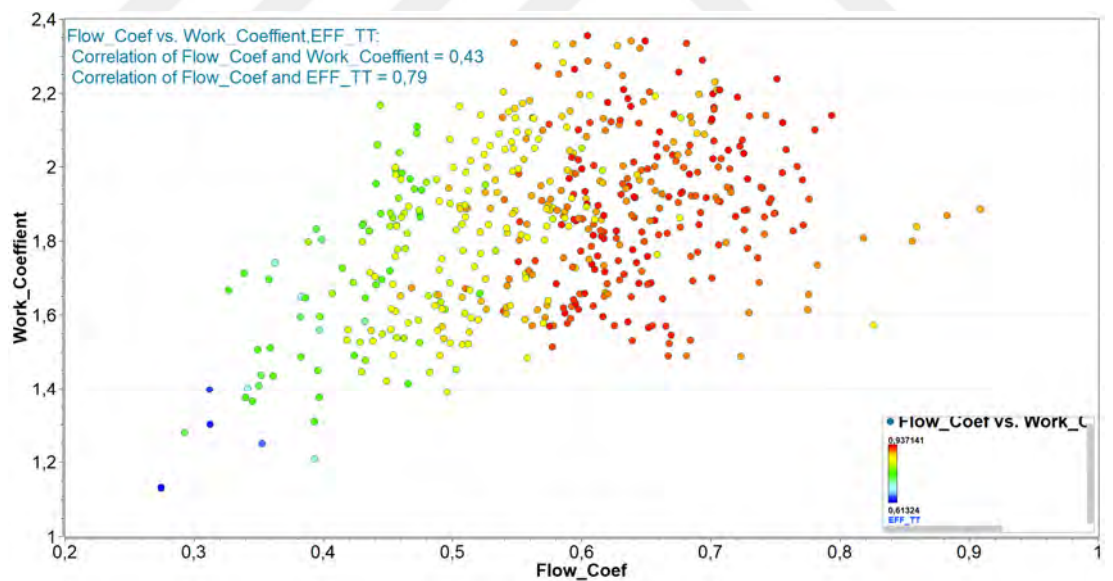


Figure 4.16: MF2S Smith Chart.

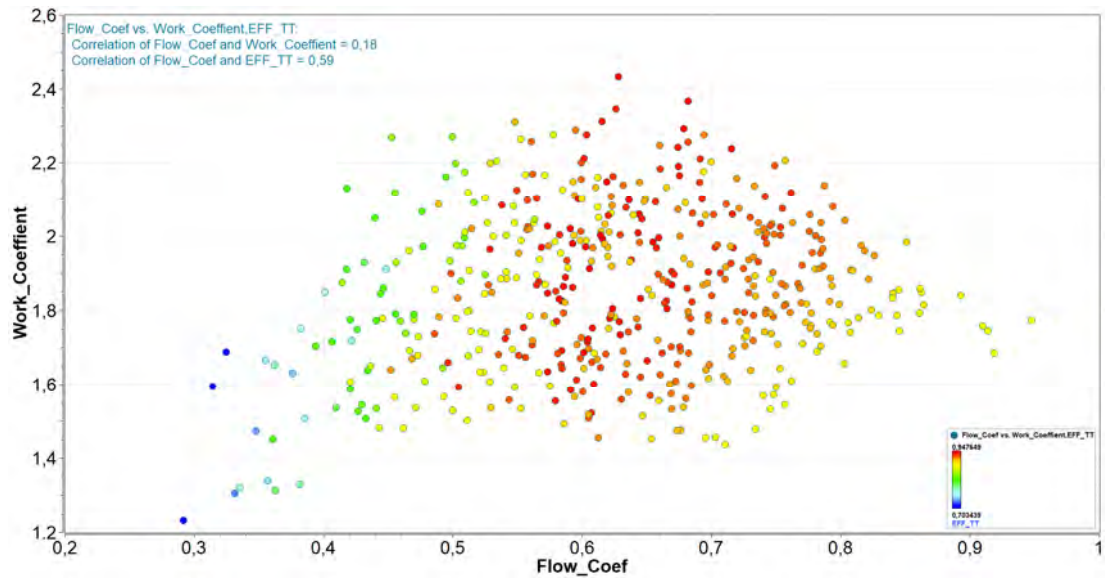


Figure 4.17: MF1 Smith Chart.

At **Figure 4.16** shown the scatter plot of MF2S configuration with smith chart. On X axis, flow coefficient is located with range of 0.28 to 0.91. On the Y axis work coefficient is located with range of 0.56 to 1.18 and total-to-total efficiency is colored from blue as lower and red as higher. The results are as expected as typical turbine stage range and Smith's Chart [23]. The design map is smaller, but the chart follows the trend from Smith's Chart. As the flow coefficient and work coefficient are getting lower, the total-to-total efficiency of the turbine is dramatically reducing.

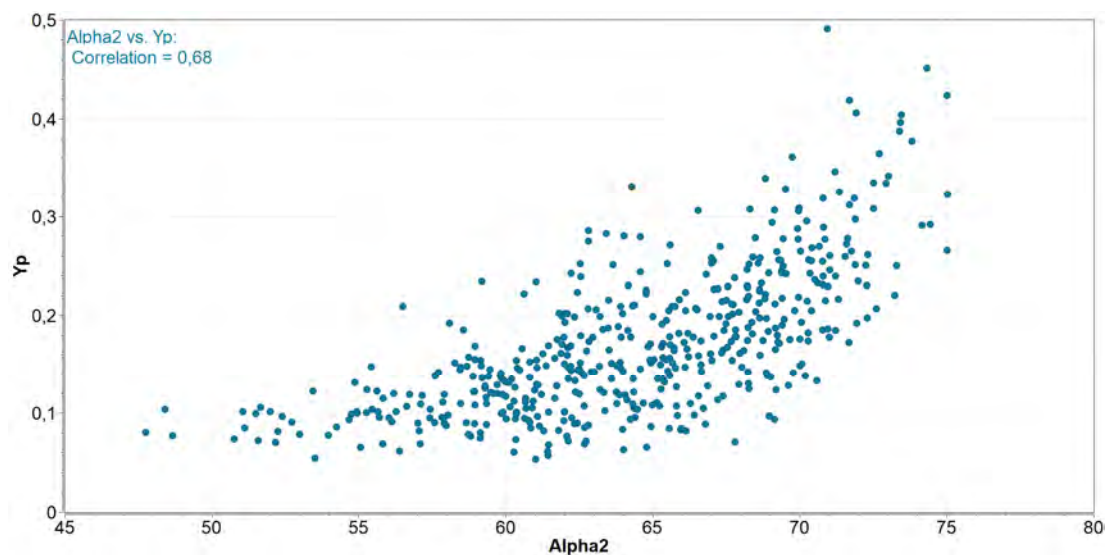


Figure 4.18: MF2S exit flow angle effect on Y_p .

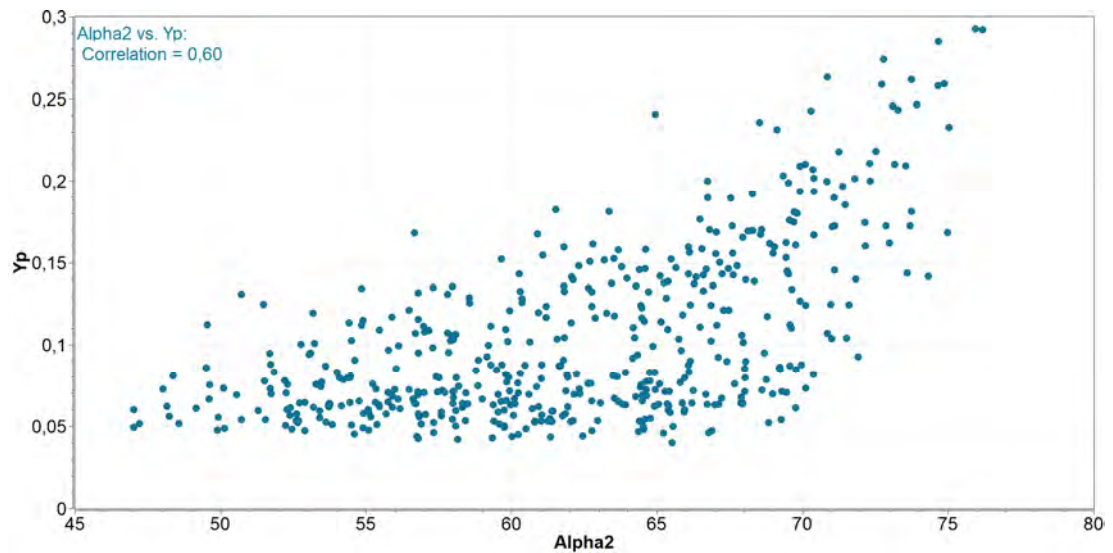


Figure 4.19: MF1 exit flow angle effect on Y_p .

At **Figure 4.18** effect of exit flow angle is scatter plotted against pressure loss Y_p . It is seen that as the exit flow angle increases the pressure loss is exponentially increased. This chart also shows the behavior of Ma_2 the frame exit Mach number as seen on **Figure 4.20** because as exit angle increases the exit Mach number increases too.

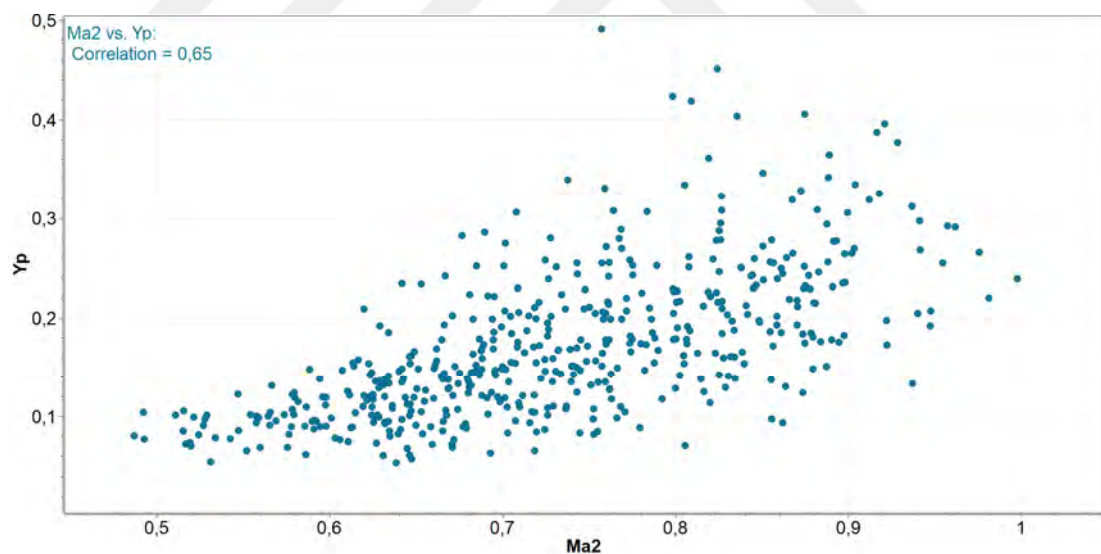


Figure 4.20: MF2S exit Mach number versus pressure loss plot.

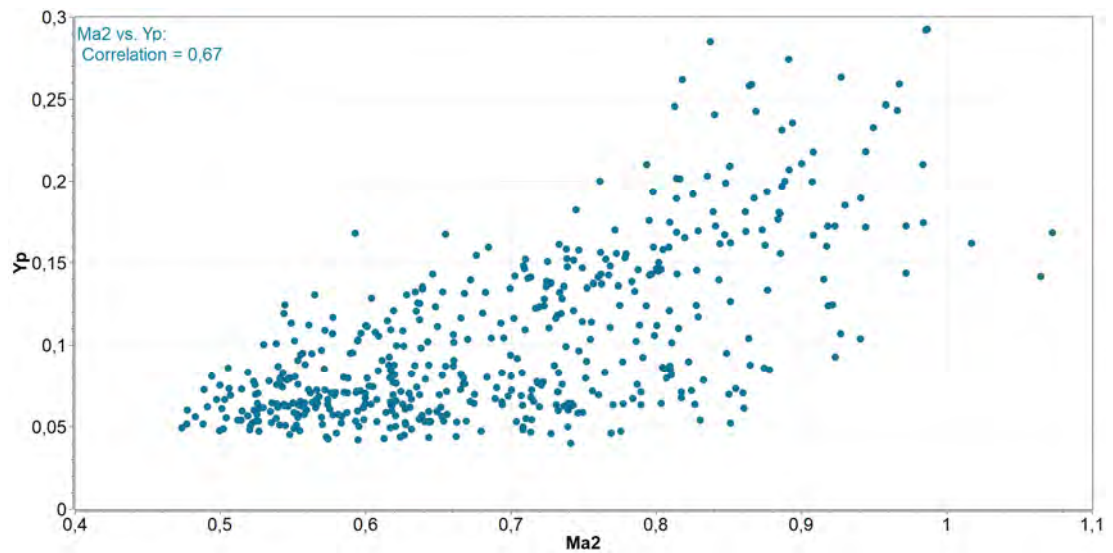


Figure 4.21: MF1 exit Mach number versus pressure loss plot.

As shown in the figures, the pressure loss is affected as the Mach number of the exit flow increases. But also, the correlation between Mach number and pressure loss seemed to be around 0.64. Thus, to improve the correlation, an additional parameter representing the blade loading is introduced, and its plots are shown in **Figure 4.22** and **Figure 4.23**.

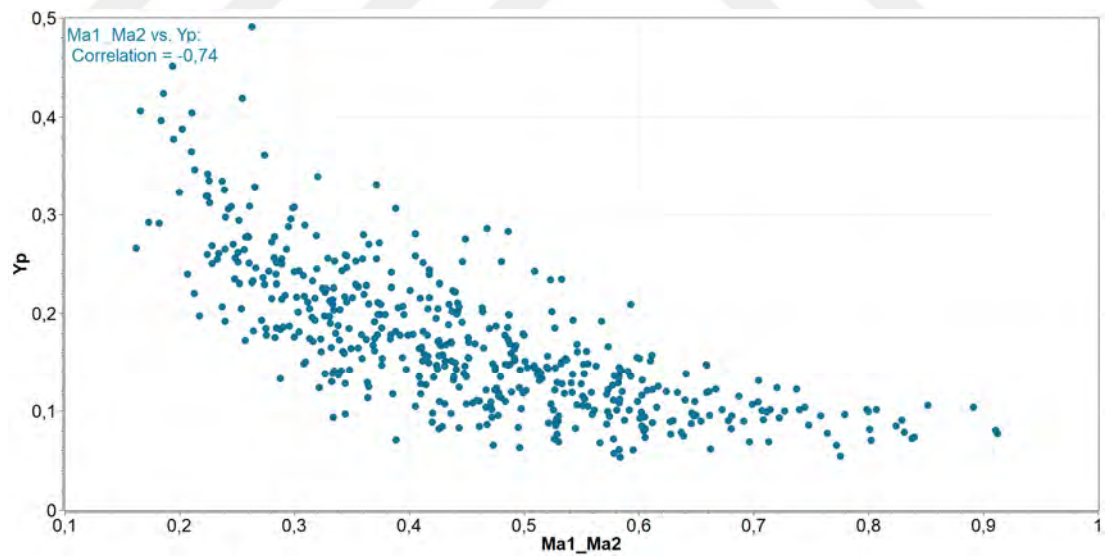


Figure 4.22: MF2S Mach number effect on Y_p .

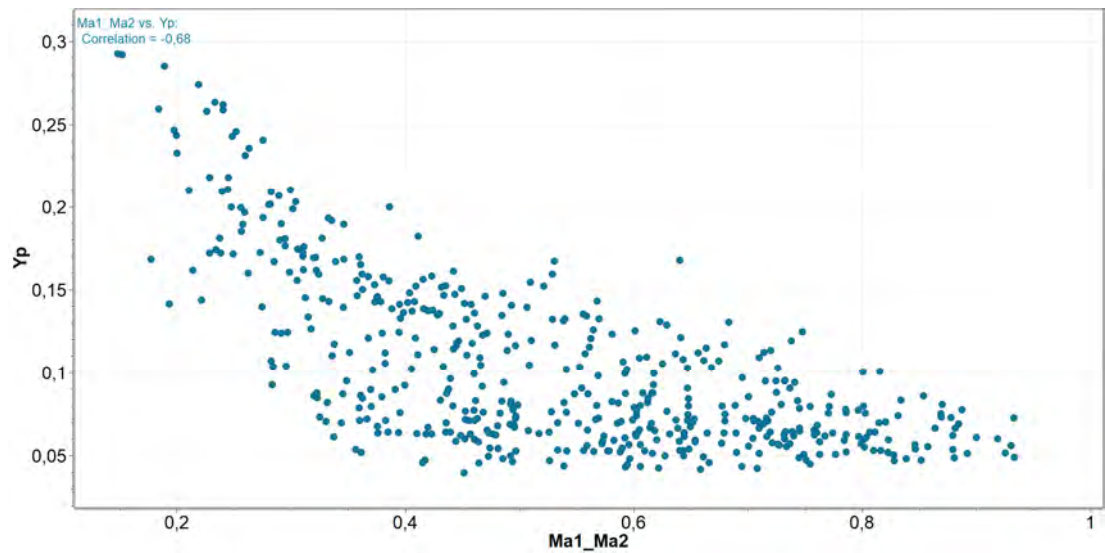


Figure 4.23: MF1 Mach number effect on Y_p .

The ratio of inlet over exit Mach Numbers at MF2S configuration is plotted at **Figure 4.22**. This is an important parameter as it is also defined at Kacker-Okapuu Loss Model [26]. As the Mach Number ratio is decreasing the loading of row is increasing. The loss coefficient is higher at especially transonic blades which are highly loaded blades than subsonic blades. The main reasons for it, shock losses which have occurred at transonic region also another reason is increase of boundary layer thickness at high curvature and high turning regions.

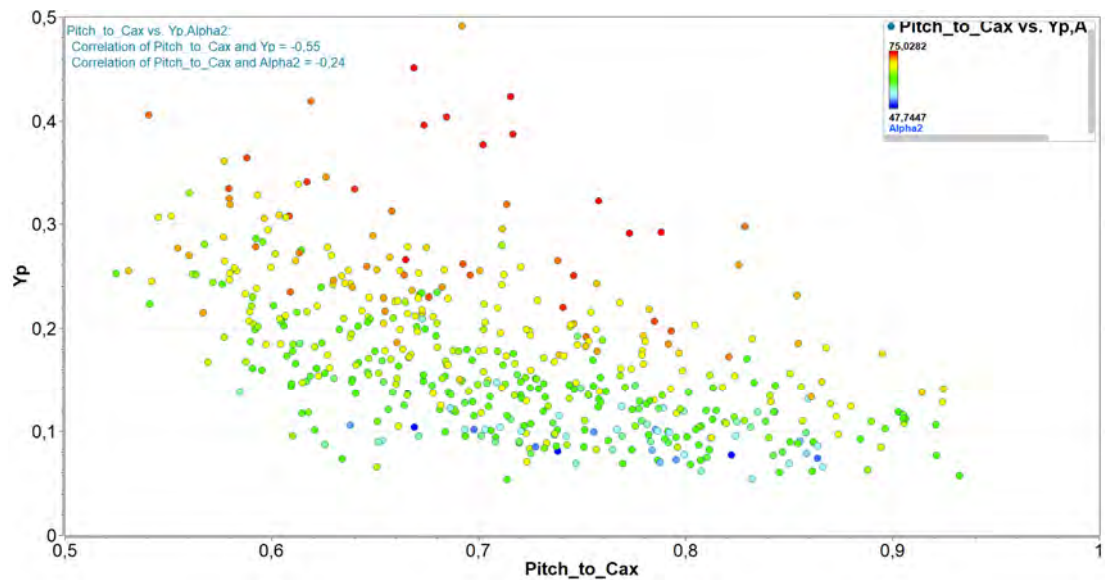


Figure 4.24: MF2S effect of pitch to axial chord on Y_p .

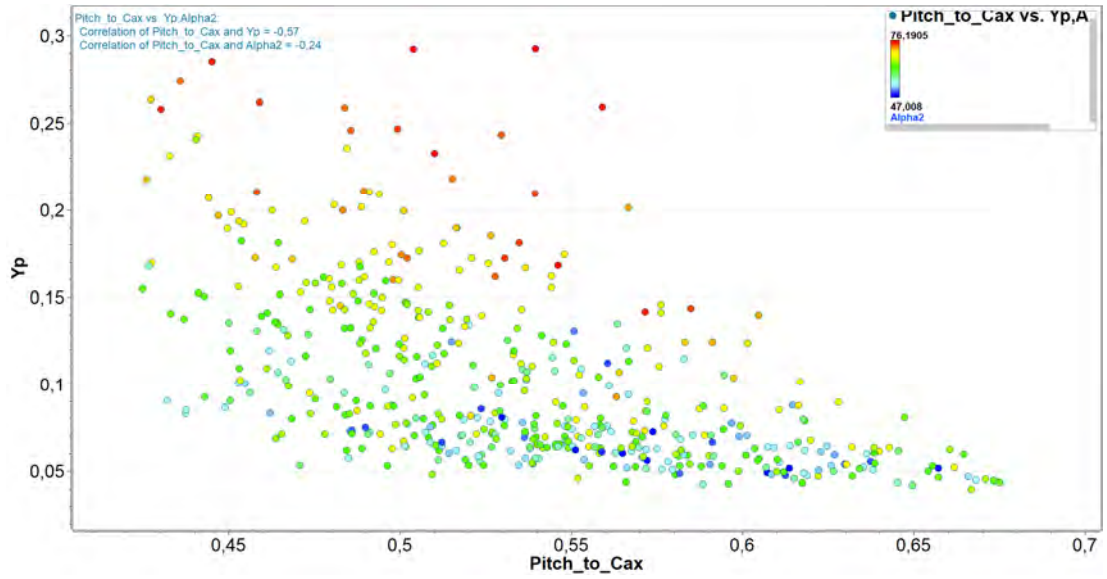


Figure 4.25: MF1 effect of pitch to axial chord on Y_p .

Figure 4.24 illustrates the effect of the pitch-to-axial-chord ratio (S/C_{ax}) on pressure loss, with color contours representing the exit flow angle. This parameter is combined with the blade exit angle in the calculation of profile losses using the Kacker–Okapuu model [25]. The S/C_{ax} ratio determines the number of blades and, consequently, the passage area and its distribution. It is observed that an increase in blade number leads to a significant rise in pressure loss. This behavior is attributed to the reduction in passage area, which promotes shock-related losses. Under these conditions, increasing the blade exit angle further narrows the passage area, making the occurrence of multiple shocks inevitable and resulting in higher pressure losses. Conversely, reducing the number of blades enhances blade efficiency up to a certain S/C_{ax} ratio, as predicted by the Kacker–Okapuu model. Beyond this optimum, pressure loss increases again because the blades become highly loaded, making them more susceptible to adverse pressure gradients, while aggressive exit angles further increase flow deviation.

The effects of certain parameters on the loss coefficient cannot be considered in isolation. As illustrated in **Figure 4.26**, **Figure 4.27** and **Figure 4.28**, scatter plots are presented showing the relationships between incidence angle and pressure loss, blade turning angle and pressure loss, and splitter inlet offset angle $\beta_{1s,offset}$ and pressure loss, respectively.

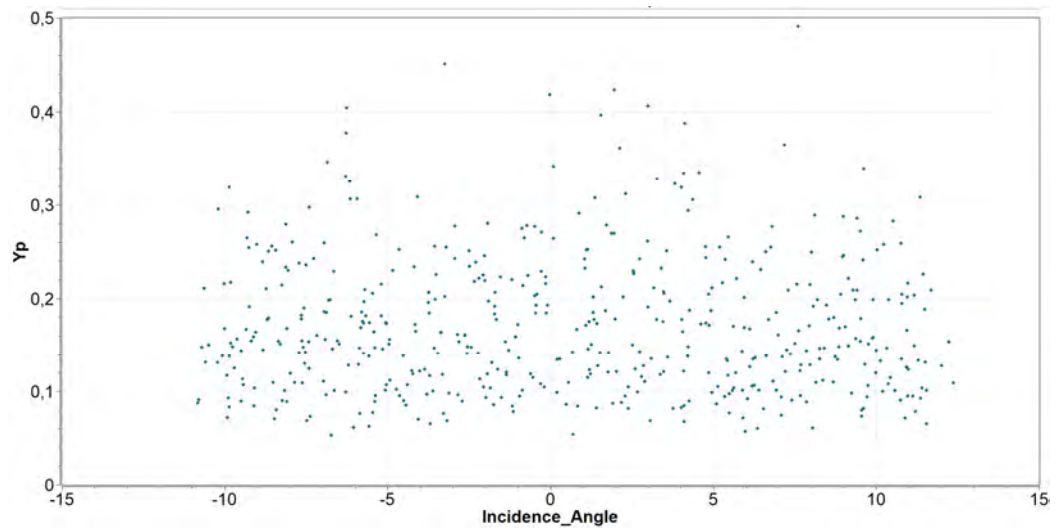


Figure 4.26: Scatter plot of incidence angle versus pressure loss.

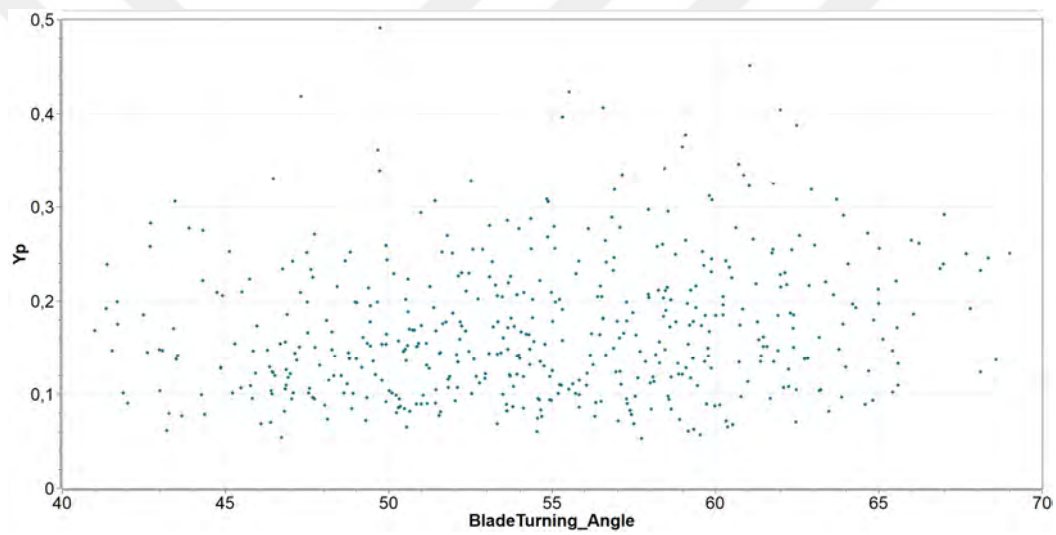


Figure 4.27: Scatter plot of blade turning angle versus pressure loss.

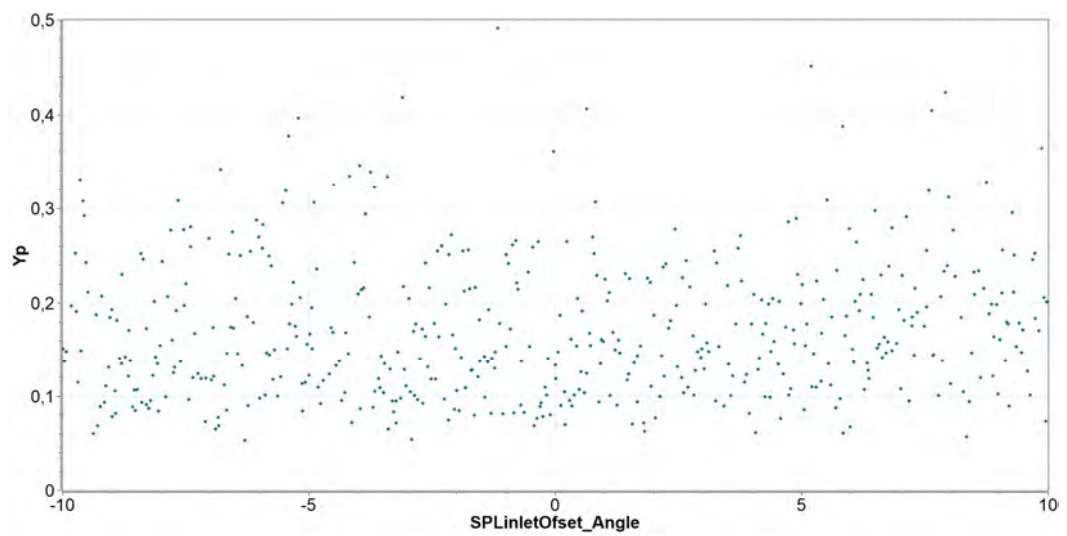


Figure 4.28: Scatter plot of splitter inlet offset angle versus pressure loss.

As observed, the effects of the incidence angle, blade turning angle, and splitter inlet offset angle cannot be adequately represented in isolation with respect to pressure loss. To capture the combined influence of these parameters, a reduced surrogate model is developed. The model is trained using the following input variables: the main blade turning angle $\Delta\beta$, splitter turning angle $\Delta\beta_{\text{splitter}}$ and splitter inlet angle offset $\beta_{1s,\text{offset}}$ as blade angle parameters; the leading-edge radius-to-axial-chord ratio $R_{\text{LE}}/C_{\text{ax}}$, trailing-edge radius-to-axial-chord ratio $R_{\text{TE}}/C_{\text{ax}}$, and splitter leading-edge radius-to-axial-chord ratio $R_{\text{LE},s}/C_{\text{ax}}$ as thickness parameters; the aspect ratio H/C_{ax} and pitch-to-axial-chord ratio S/C_{ax} as dimensionless blade parameters; and the incidence angle (i) and deviation angle (d) as flow parameters.

A quadratic least-squares method is employed to train the surrogate model. **Figure 4.29** presents a comparison between the actual pressure loss and the pressure loss predicted by the model.

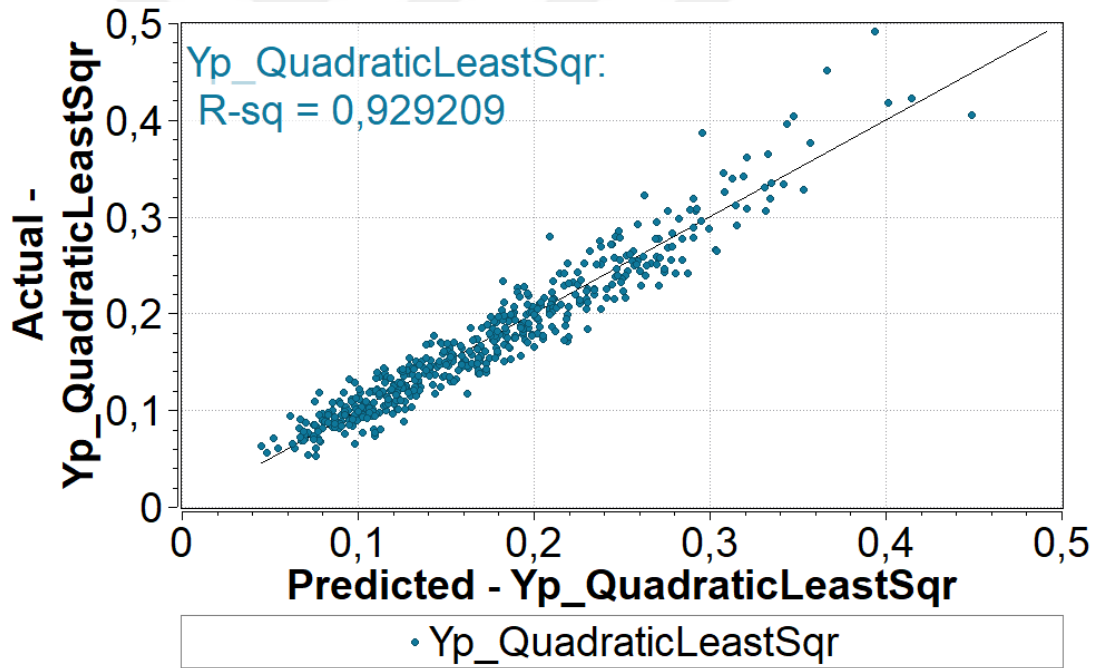


Figure 4.29: Comparison of actual and surrogate-predicted pressure loss.

The coefficient of determination (R^2) of the surrogate model is 0.929, while the predictive R^2 is 0.839, and the root mean square error (RMSE) of the model is 0.0197. Following model training, additional predictions are generated to cross-validate the previously presented scatter plots.

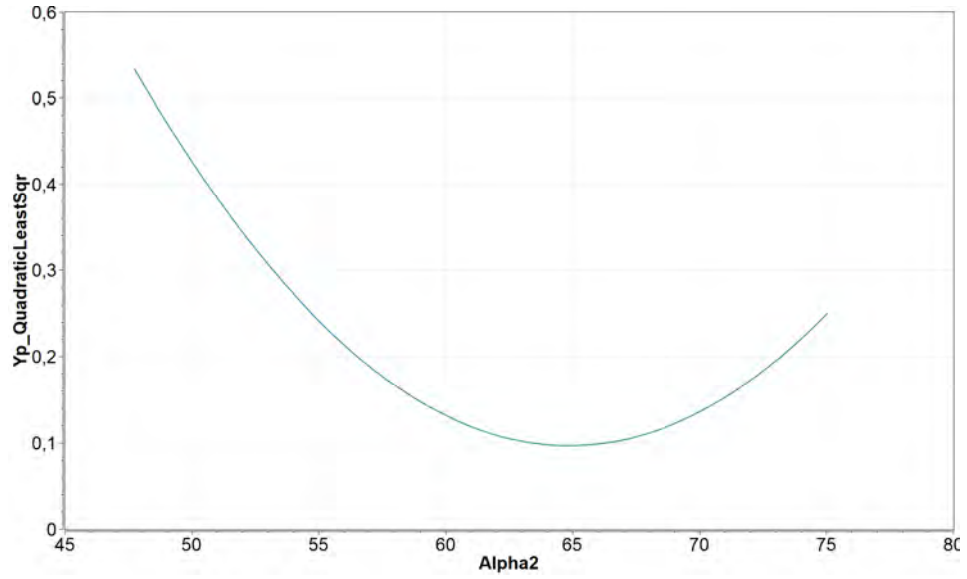


Figure 4.30: Surrogate model prediction of pressure loss versus mid-frame α_2 .

As shown in **Figure 4.30**, the effect of the exit flow angle on pressure loss is presented independently. The behavior of the plot is observed to be consistent with the Kacker–Okapuu loss model, as illustrated in **Figure 2.5** [25].

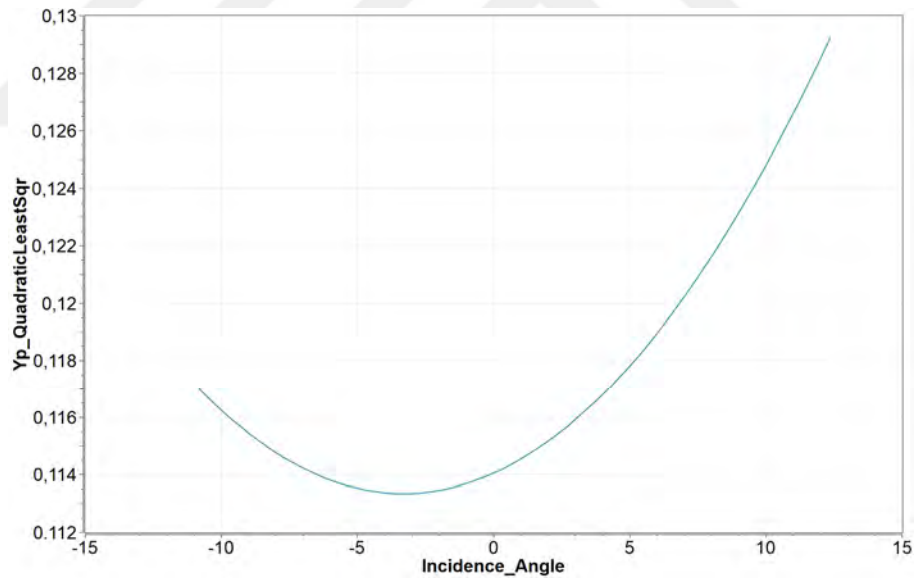


Figure 4.31: Scraped effect of incidence angle versus Y_P .

Figure 4.31 validates the effect of the surrogate model on pressure losses with respect to incidence angle, in agreement with the literature [20]. The minimum point is shifted from 0° to -3° , which is considered acceptable. The main trend is as follows: as the incidence angle decreases (negative incidence), the pressure loss increases slightly; however, as the incidence angle increases (positive incidence), the pressure loss rises sharply.

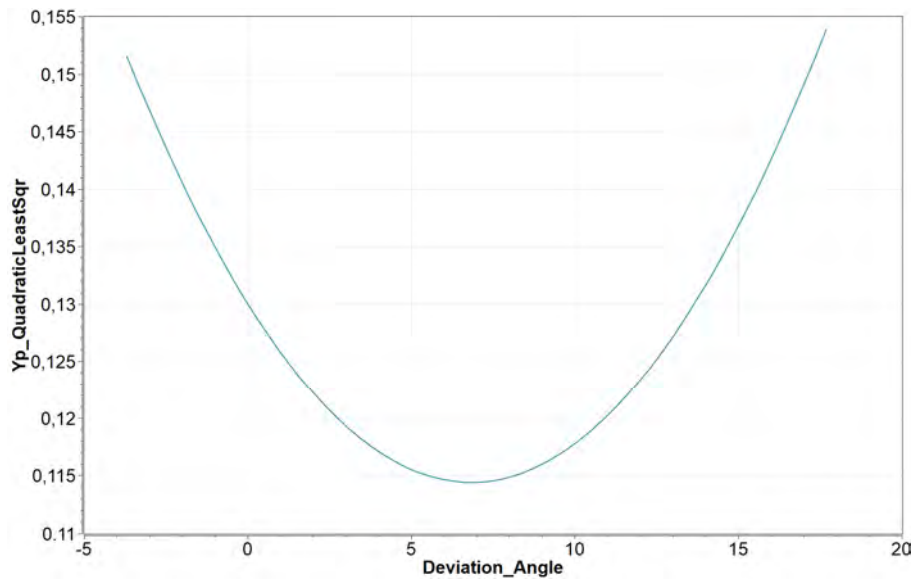


Figure 4.32: Model results on deviation angle versus pressure loss.

Furthermore, the effect of the deviation angle is represented. Although its minimum is shifted from the expected 0° , the overall trend confirms the influence of the deviation angle on pressure losses [20].

Additionally, splitter inlet angle offset value against incidence angle and against main blade turning angle are plotted on 2D response surface.

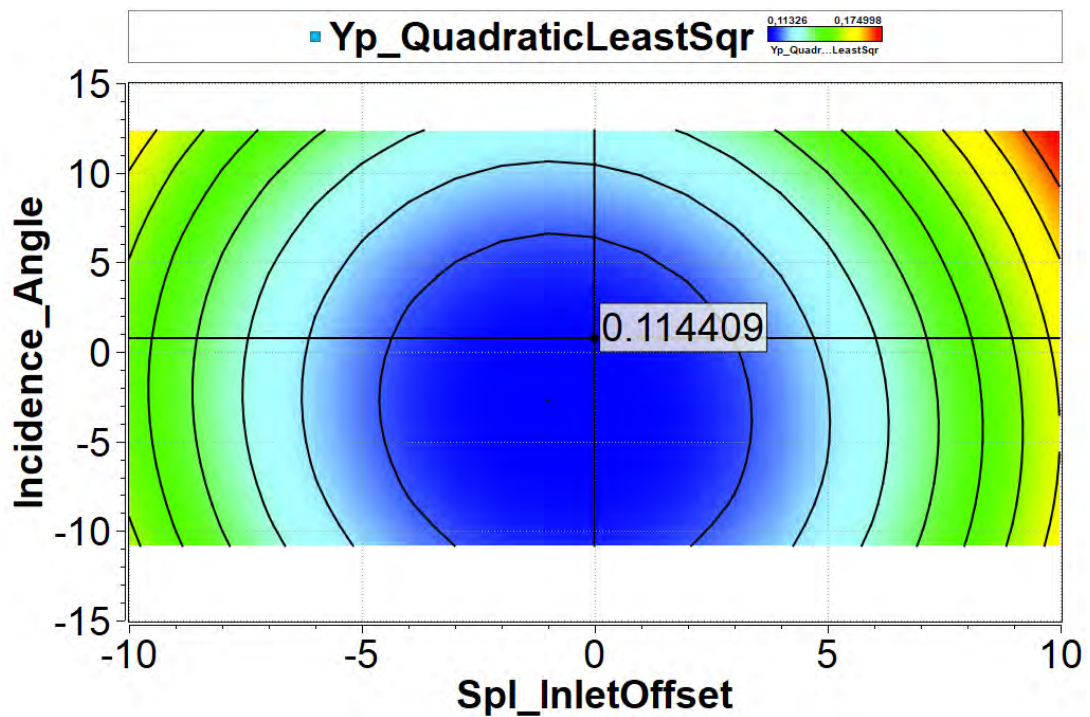


Figure 4.33: 2D response surface of Y_p as function of $\beta_{1s,offset}$ and frame (i).

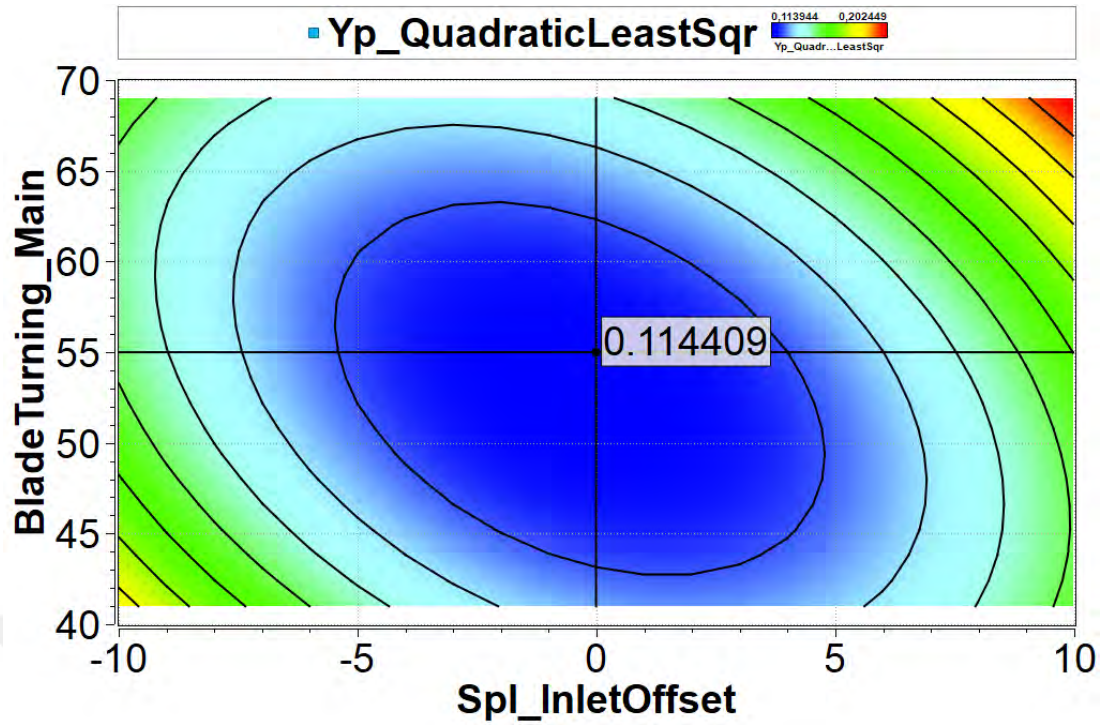


Figure 4.34: 2D response surface of Y_p as function of $\beta_{1s,offset}$ and $\Delta\beta_{main}$.

As the main frame is introduced with positive incidence angle, flow may separate on SS side due to the high curvature at leading-edge and adverse pressure gradient association. Thus, offsetting the inlet angle of splitter to the SS side of frame can slightly reduce pressure loss as the channel between SS side of frame and splitter is narrowed. Under the scenario of negative incidence of main frame, only minor difference has been found thus influence of splitter inlet angle offset is neglected on this side.

Furthermore, influence of main blade turning angle and splitter offset angle on pressure loss is more distinct. As the turning increases there is chance to observe boundary layer separation around the maximum thickness location of frame. Flow can be reattached as narrowing the channel in between SS side and splitter as reducing its inlet angle. Contrary, as blade turning angle decreases flow may separate on PS side of frame and narrowing the channel between PS side and splitter as increasing inlet angle of splitter may be beneficial.

5. COMPARISON OF RESULTS

In this section, NASA/GE E3 HPT turbine is compared as a benchmark turbine with MF1 and MF2S versions. To compare the variants, second stage stator is replaced by MF1 frame and MF2S frame respectively and aerodynamic characteristics are compared. The versions of MF1 and MF2S are selected by filtering DoE results. Selected designs are not the most efficient from study because intent of DoE wasn't the best optimum design, but it was finding global trend of characteristics and exploration. Thus, the best suitable candidates are selected with filtering pressure ratio, blade turning, corrected mass flow rate and total-to-total efficiency.

At first, pressure ratios from DoE studies are selected in between 2.05 to 2.15 for filter. Later, the selection criteria was a design with similar capacity of benchmark stator of second stage which is around 11.30 kg/s. Another criterion was finding a design around the stage efficiency which is 92% and final criteria was finding a high blade turning, minimum 65 degrees. The used scatter plot of both MF1 and MF2S designs during the filtering are as shown at **Figure 5.1** and **Figure 5.2**.

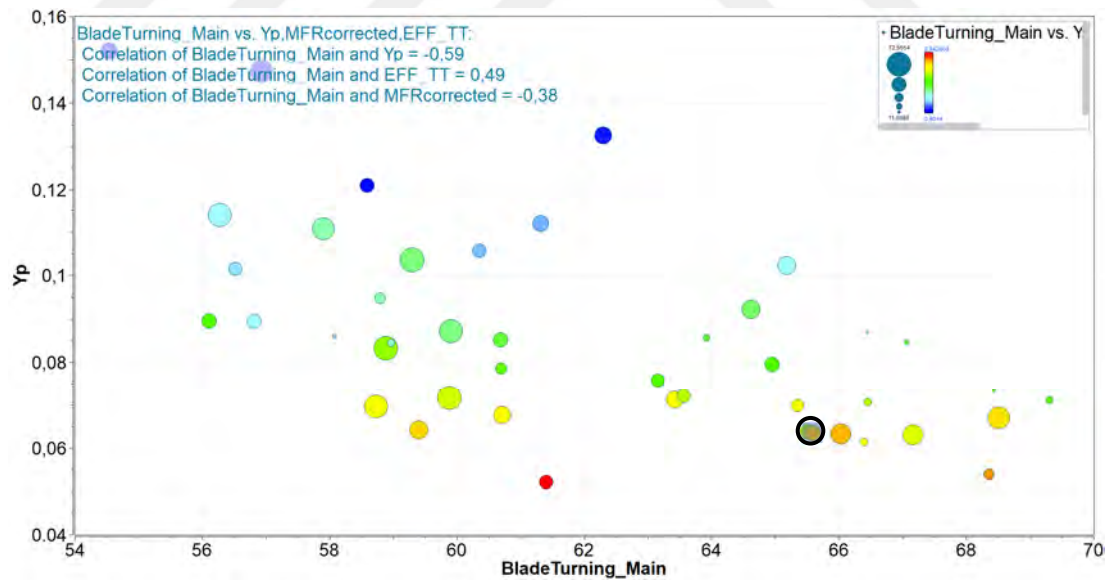


Figure 5.1: MF1 selection filter plot.

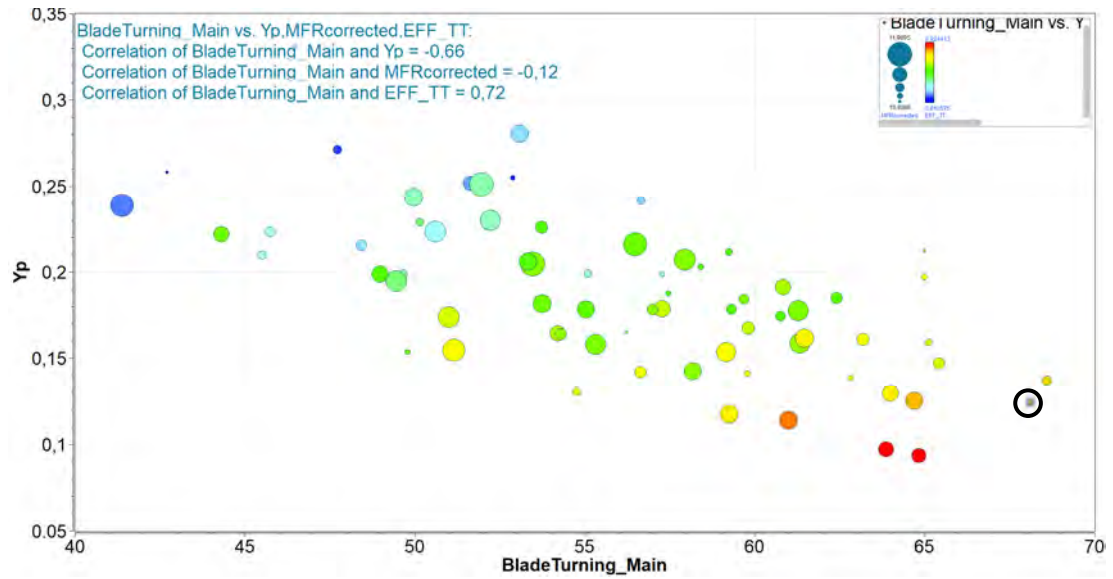


Figure 5.2: MF2S selection filter plot.

A balanced selection required a couple of trial-error even the filter parameters requirements have met. So, the blading efficiency parameter is introduced too. As the corrected mass flow rate of second stage is around 11.30 kg/s and pressure ratio is 2.11 these variants are selected for comparison [11]. The metrics of selected configurations as shown at **Table 5.1**.

Table 5.1: Selected MF1 and MF2S parameters.

Selection Properties	MF1	MF2S
$\dot{m}_{corr}$	11.57	11.06
PR	2.14	2.14
$\Delta\beta$	65.8	68.1
η_{tt}	0.92	0.90
BladingEFF	0.94	0.92

Mean section of selected profiles are as shown at **Figure 5.3**.

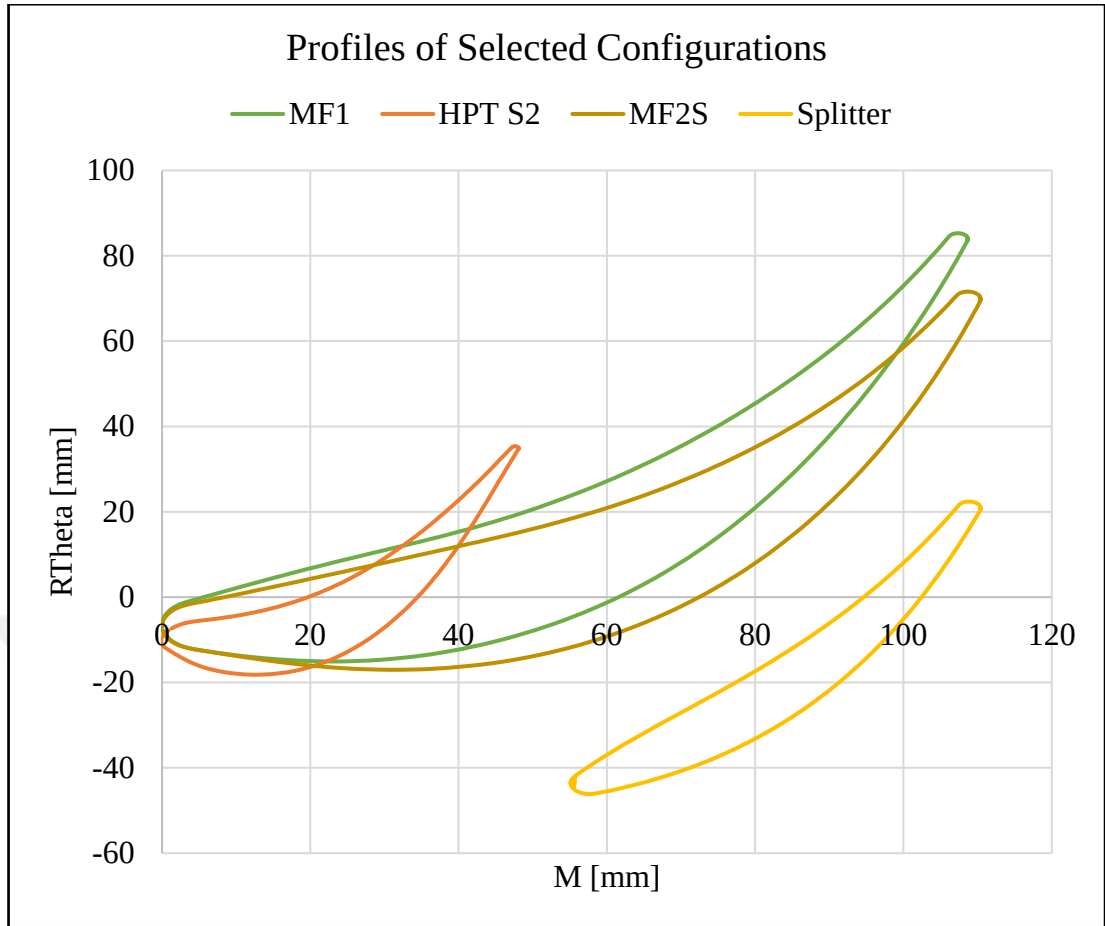


Figure 5.3: Profiles of selected configuration.

As compared to **Figure 4.2** the base profiles MF1 and MF2S blade inlet and exit angles almost same but the beta distribution has changed. Turning at front is slightly increased for MF1 on the contrary side aft loading is decreased.

On the other hand, for MF2S configuration front loading and aft loading decreased slightly. The most noticeable change has occurred at MF2S splitter. The profile thickness is increased dramatically, and around 8 degrees there is an inlet profile offset which opens the channel splitter between SS side and narrows the side between PS of another frame.

For B2B calculation mesh density has kept same for benchmark study and for MF1&MF2S configurations the first stage stator and rotor and second stage rotor also kept same. For MF1 and MF2S the density has kept same as DoE study. Boundary conditions for CFD are the EDP point of HPT as described at **Table 3.3**.

Geometries of all configurations with B2B CFD Mach Number contour are shown in **Figure 5.4**, **Figure 5.5** and **Figure 5.6**.

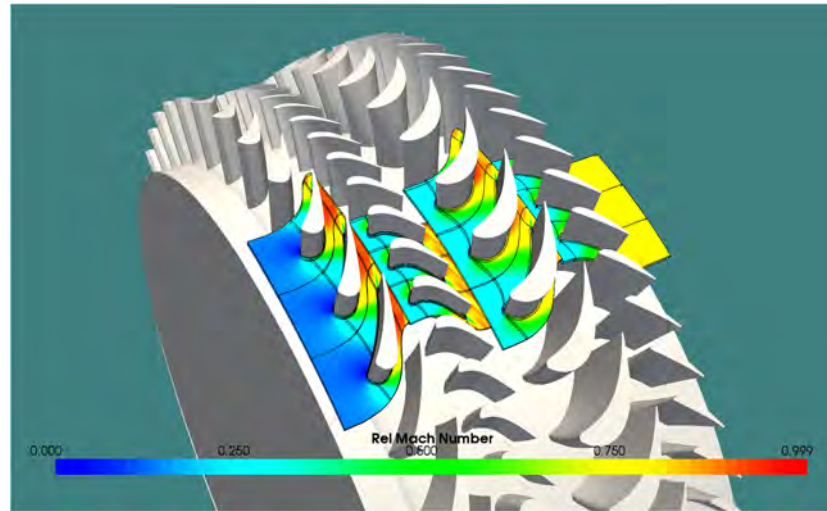


Figure 5.4: E3 3D geometry with mean section B2B CFD Ma contour plot.

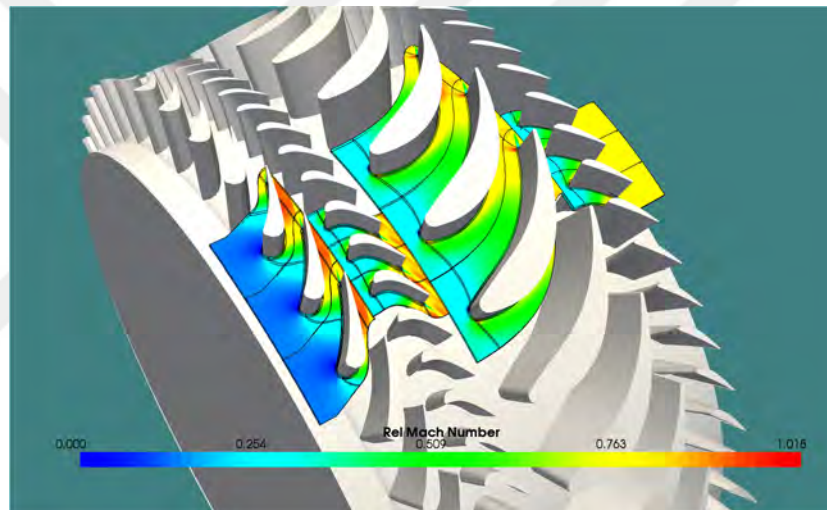


Figure 5.5: MF1 geometry with mean section B2B CFD Ma contour plot.

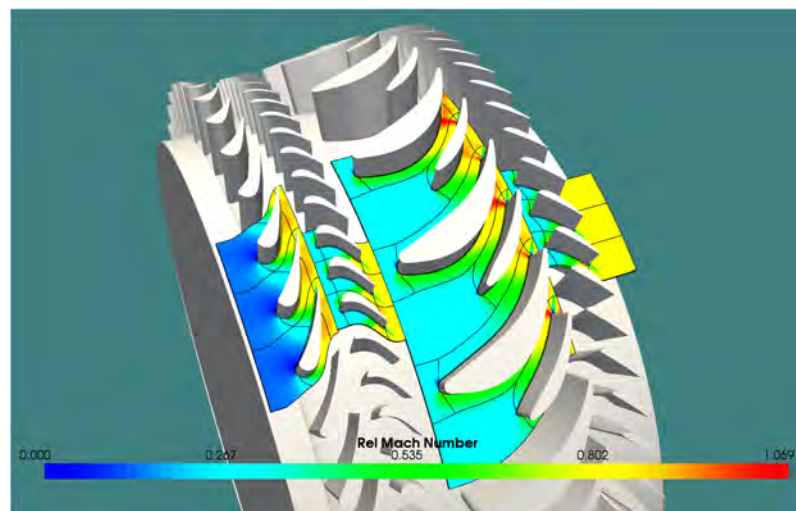


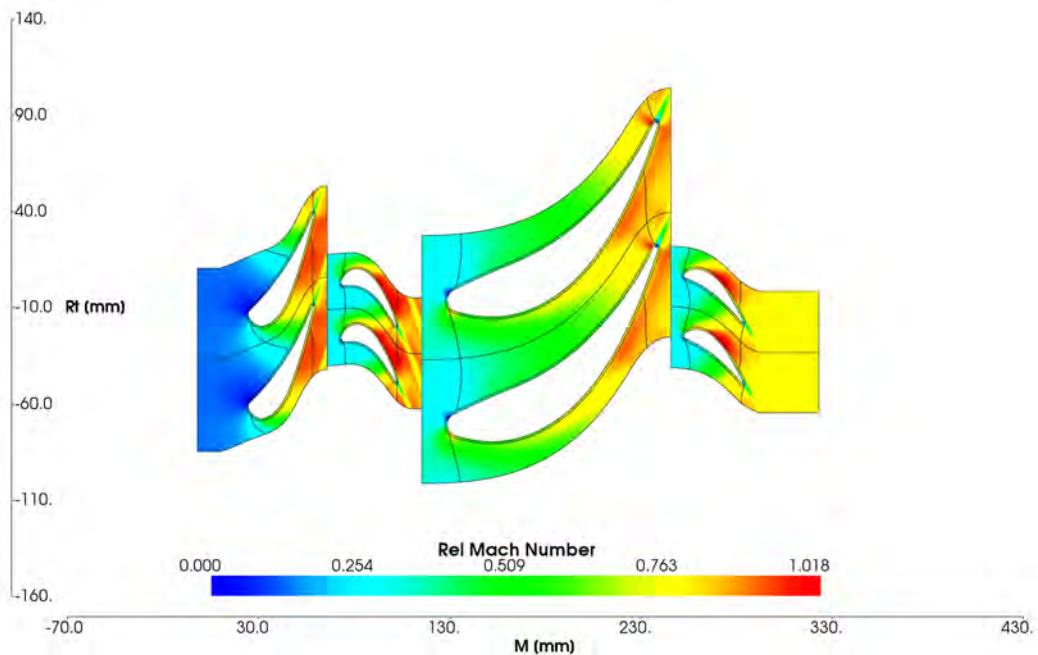
Figure 5.6: MF2S geometry with mean section B2B CFD Ma contour plot.

Table 5.2: Flow parameters of E3 - MF1 - MF2S at mean section.

	E3 S2	MF1	MF2S		E3 S2	MF1	MF2S
Parameters	Stage 1 Mean Section			Parameters	Stage 2 Mean Section		
S1 α_1 [°]	0	0	0	S2 α_1 [°]	-20.8	-22.2	-19.3
S1 α_2 [°]	73.7	73.8	73.8	S2 α_2 [°]	68.9	68.3	68.9
R1 β_2 [°]	36.3	36.9	36.5	R2 β_2 [°]	13.4	12.6	18.2
R1 β_3 [°]	-66.0	-66.0	-66.1	R2 β_3 [°]	-59.0	-58.9	-58.9
R1 α_3 [°]	-16.5	-17.5	-14.8	R2 α_3 [°]	5.1	7.1	8.2
S1 Ma ₁	0.13	0.13	0.13	S2 Ma ₁	0.326	0.331	0.31
S1 Ma ₂	0.86	0.87	0.86	S2 Ma ₂	0.829	0.828	0.86
R1 Mr ₂	0.32	0.33	0.32	R2 Mr ₂	0.307	0.309	0.32
R1 Mr ₃	0.86	0.87	0.84	R2 Mr ₃	0.813	0.795	0.79
R1 Ma ₃	0.39	0.40	0.37	R2 Ma ₃	0.430	0.421	0.42

Table 5.2 summarizes the mean-section flow angles and Mach numbers for the three configurations. Stage-1 quantities are close to each other, while the main differences are observed around the second-stage vane replacement and the downstream rotor inlet. MF2S shows a higher exit absolute Mach number at the row compared to MF1, respectively 0.86 and 0.829. In addition, MF2S results in a larger R2 inlet relative flow angle 18.2° than MF1 12.6° which indicates a stronger change in the downstream velocity triangle.

The Mach number contours are as shown in the **Figure 5.7** and **Figure 5.8**.

**Figure 5.7:** MF1 Mach number contour plot.

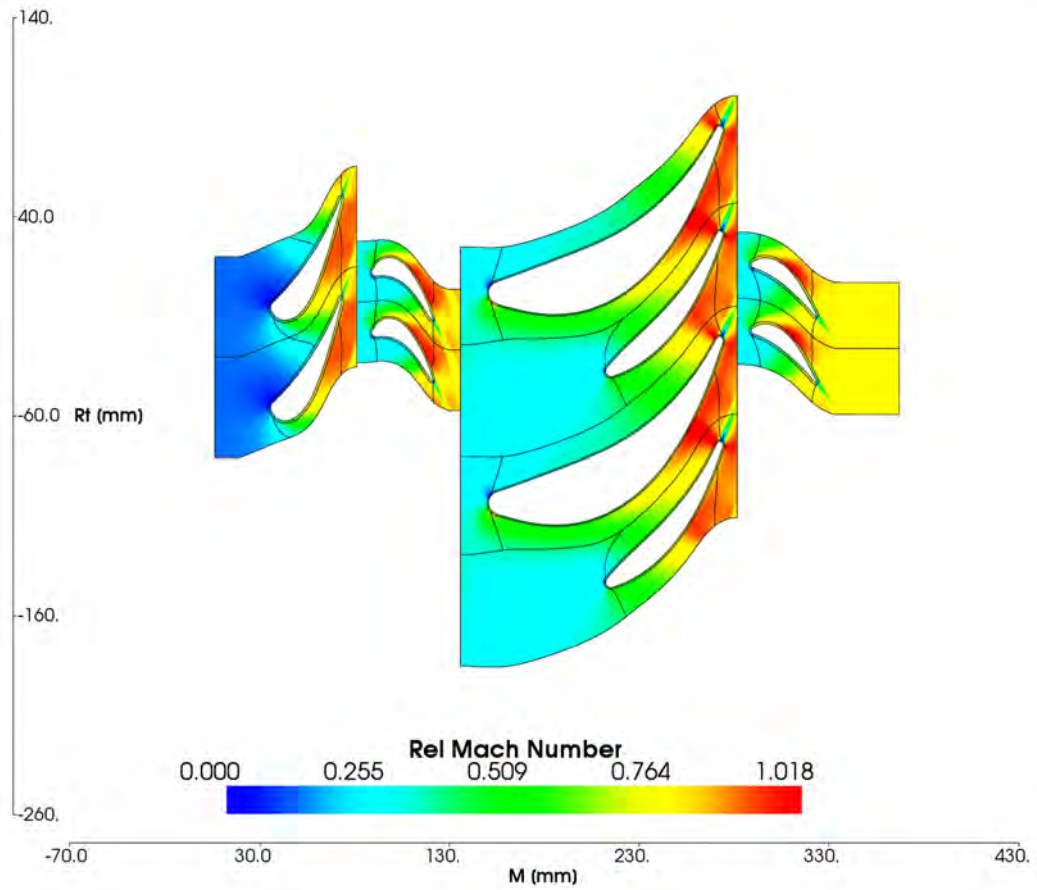


Figure 5.8: MF2S Mach number contour plot.

As visualized on **Figure 5.7** and **Figure 5.8**, upstream of both second stages are somewhat changed. R1 exit flow Mach number is 0.4 and R1 exit angle is -17.5° at MF1 configuration which is slightly higher than MF2S configuration. R1 exit conditions of MF2S are 0.37 exit Mach number and -14.8° exit flow angle. As the R1 exit flows leading to S2 inlet channel, the inlet Mach numbers are reduced to 0.33 and 0.31 and the inlet flow angles are increased to -22.2° and -19.3° for MF1 and MF2S configurations respectively. Because of higher inlet flow angles than inlet metal angles, the leading edges of mid frames are introduced with positive incidence angles. Nevertheless, there is no flow separation occurred at the front section of both configurations. The close-up visuals of LE regions are presented in **Figure 5.9** and **Figure 5.10**.

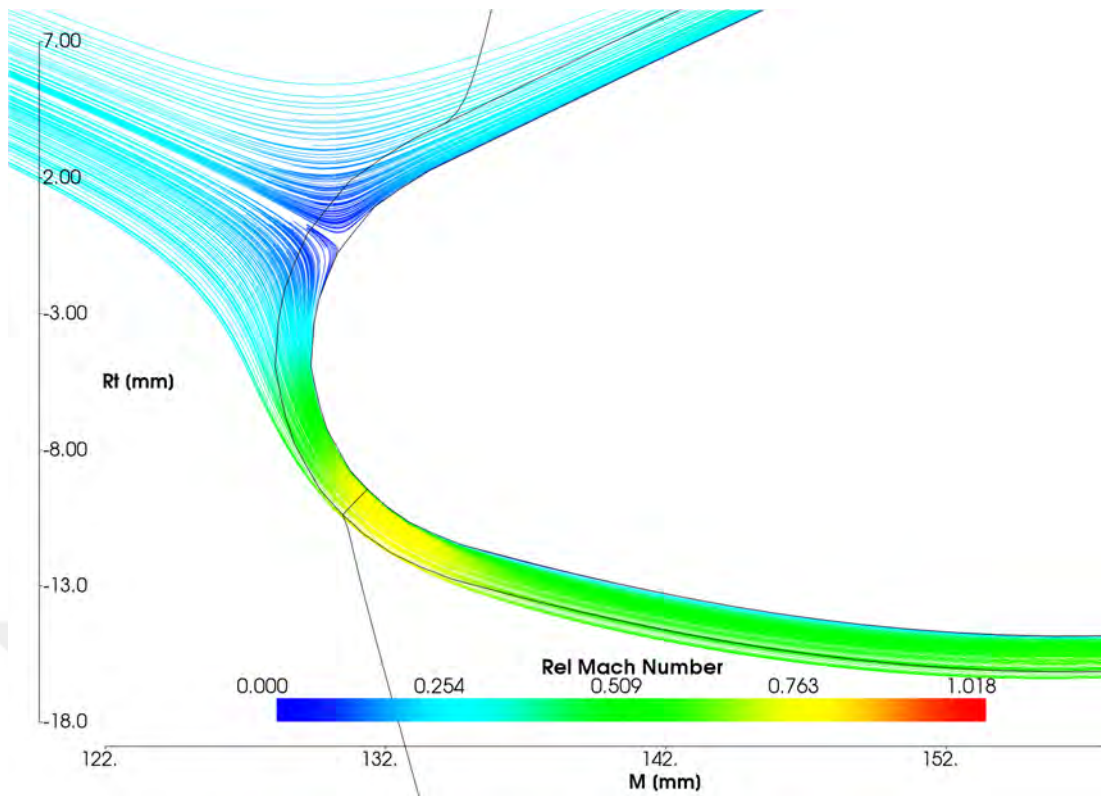


Figure 5.9: MF1 LE Streamlines.

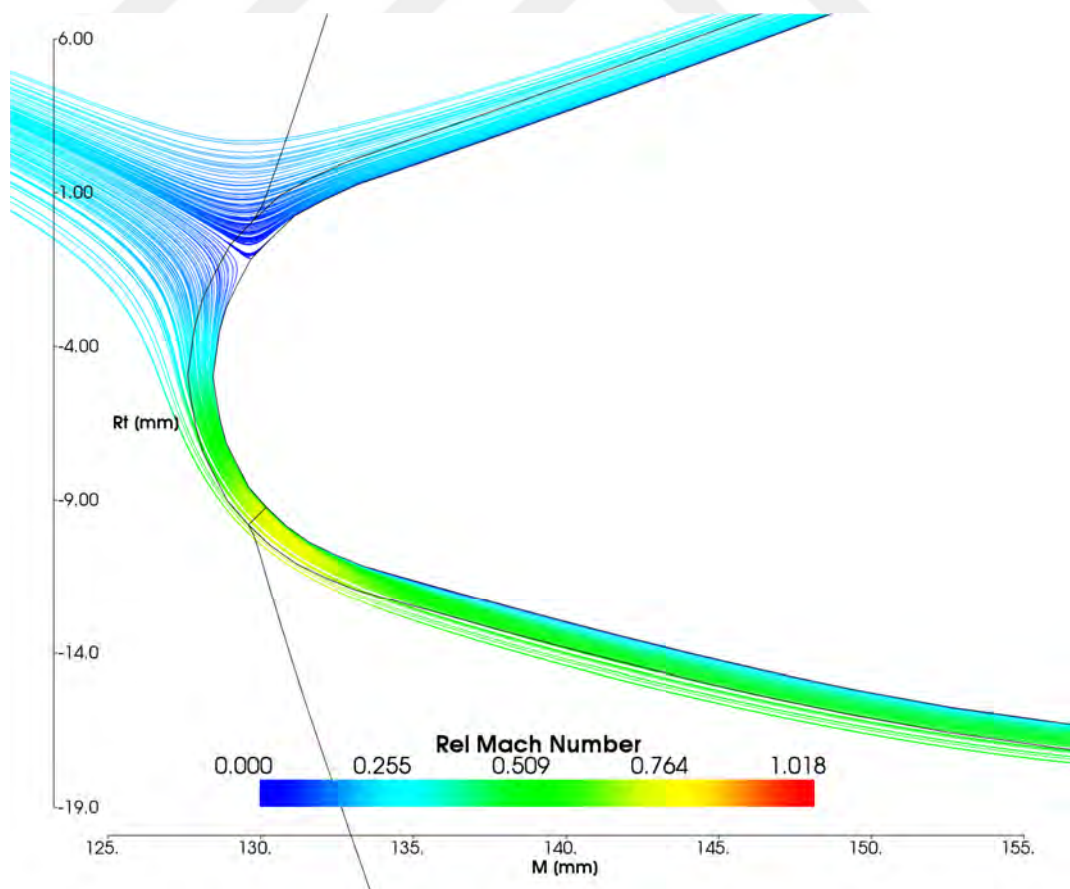


Figure 5.10: MF2S LE Streamlines.

As the flow and metal angles weren't met in this study incidence experienced at mid frames. From literature it is well known that positive incidence flows are prone to separate because of high curvature of leading-edge [20]. But for MF1 and MF2S cases flow separation is not visible but a small portion of adverse pressure gradient at suction side. Normally, it is expected to separate because of leading edge curvature but only small portion of adverse pressure gradient at the blade wall is experienced after sudden acceleration at the end of leading-edge. High radius of leading-edge and wedge angle can improve incidence tolerance, also the curvature of blade is almost minimum at the start of suction side. Thus, no separation has been experienced for both configurations.

As closing-up the Mach number contours of mid passage of MF1 and MF2S both frames are experiencing sudden shock in the throat section, which are visible at **Figure 5.11** and **Figure 5.12**. When the relatively high turning is introduced with wide axial chords it is inevitable to be choked at the throat section. After choking, flow diffuses depending on beta and thickness distribution of frames.

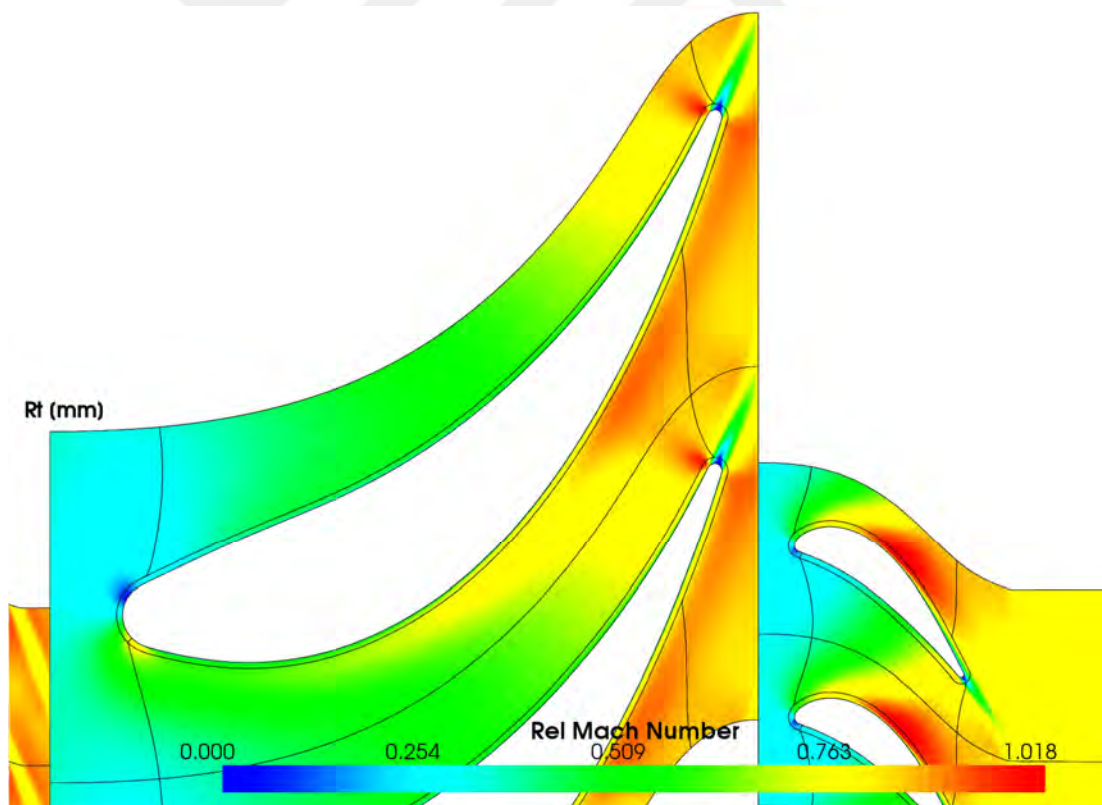


Figure 5.11: MF1 mid passage Mach number contour plot.

MF1 configuration experiences relatively small shock losses at the throat section compared to MF2S results. It is expected that the MF1 configuration has higher

corrected mass flow rate in other words flow capacity. Due to weak shock, the boundary layer thickness slightly increases at the frame walls. Finally, exit wakes can be observed at the trailing-edge section. Wake thickness is driven by trailing-edge thickness.

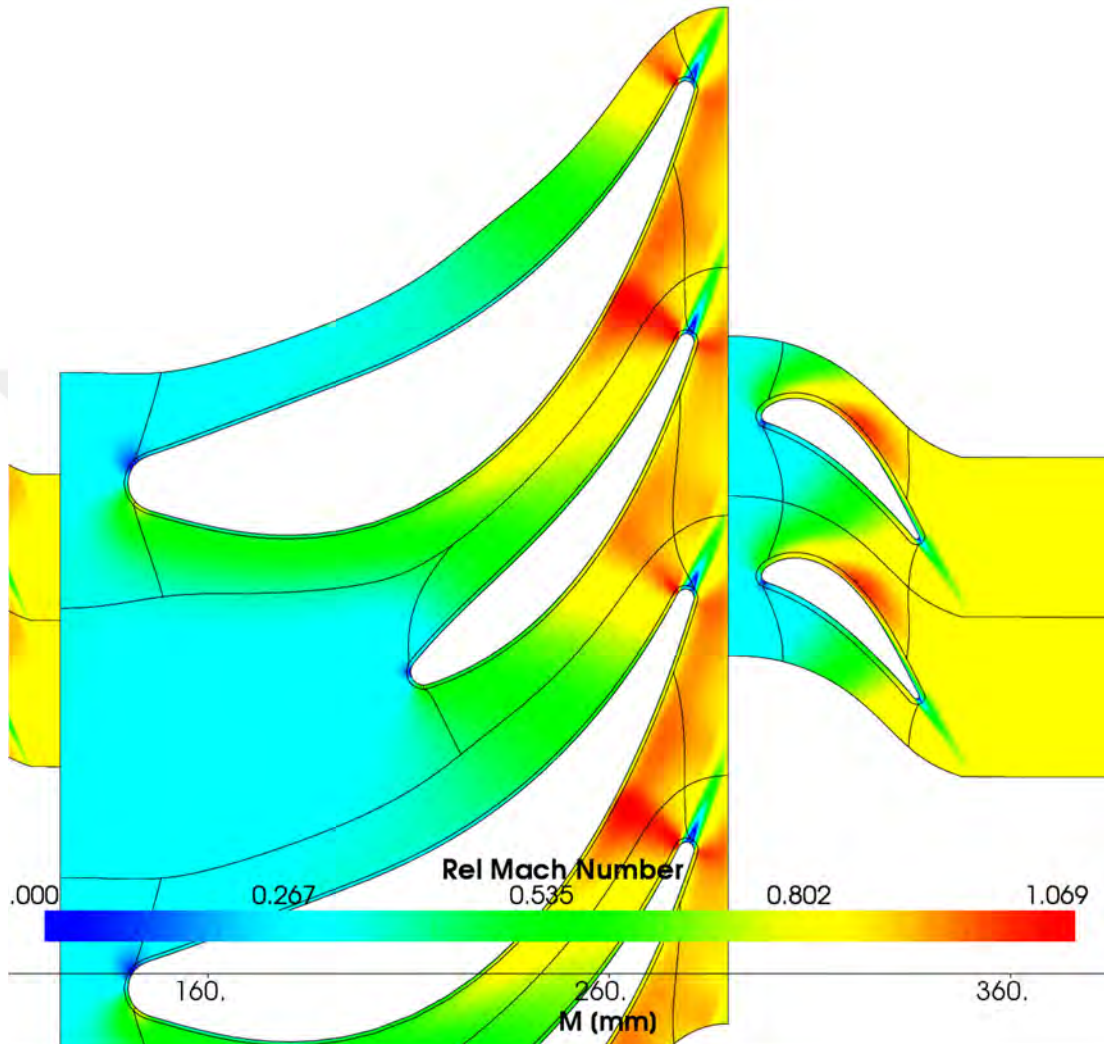


Figure 5.12: MF2S mid passage Mach number contour plot.

Mid passage of MF2S is highly differing than other configurations, MF1 and benchmark S2. Because splitter does not follow the profile of frame not as thickness but also as beta distribution especially at first 50% of its own axial chord. Due to leading-edge angle offset, the channel between the SS of frame and splitter is slightly wider than the channel between PS of frame and splitter. This is beneficial if there is a risk of the early shock at passage, such off-design conditions where the incidence angle changes depending on condition. When the channel between SS of frame and the splitter is opened acceleration rate of flow reduced but this action also introduces

risk of shock losses between PS side of frame and splitter if blading is not completed properly. In this DoE product, splitter is designed thinner at leading edge region relatively to the base MF2S.

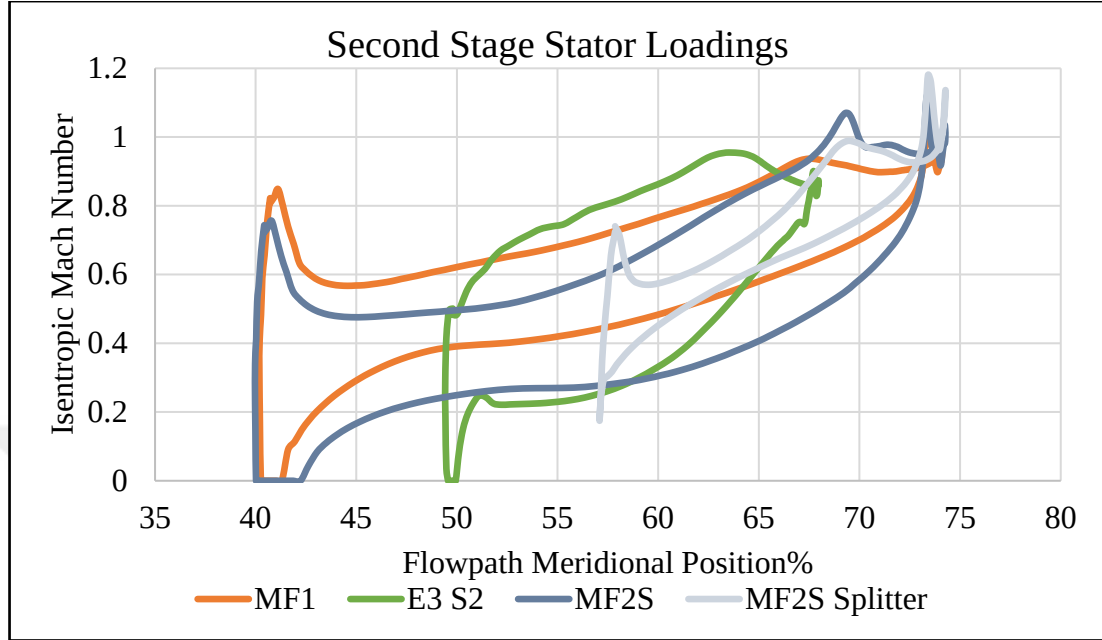


Figure 5.13: Second stage stator loadings.

The blade loading charts of all configurations are introduced at **Figure 5.13** along their meridional position respect to flow path. As stated before, incidence occurrence is visible at the leading-edge of both MF1 and MF2S also the splitter. The magnitude of incidence is higher at MF1 than MF2S which is visible from the relatively high peak of isentropic Mach number at the inlet.

Immediately at the downstream of incidence region, flow diffuses locally with the ratio of 0.626 at MF2S and 0.656 at MF1 because of weaker adverse pressure gradient.

In the mid-chord region, the difference between the suction and pressure side Mach numbers indicates the distribution of aerodynamic loading.

A smoother and more gradual variation suggests a better-conditioned pressure recovery, whereas oscillations may hint at localized acceleration–diffusion effects. The lowest loading is observed at MF1 as the pressure side isentropic Mach number is higher than any other configuration. Looking at the benchmark S2 loading, it is required to load the S2 row for efficient operation. Thus, MF2S is favorable rather than the MF1 configuration so far.

At the 57.5 percent meridional position splitter meets the flow. It is seen that due to diffusion introduced by incidence the aerodynamic loading of splitter is relatively low compared to other frames. The splitter is also experiencing low loading due to its low blade turning.

Between the throat and trailing-edge section the most loss influential shock loss occurs at MF2S configuration. It is explained by sudden diffusion, while other configurations have moderate diffusion at this region.



6. DISCUSSION AND FUTURE WORKS

In this study two stage axial turbine of E3 is referenced as a benchmark and validated with 0.14 point efficiency difference compared to literature and 0.68 points with test data at the design point RDG 12 with weighted averaged B2B CFD calculation. Calculations and the methodology have a good agreement compared with open test data and open CFD data as examining the validation results. The highest difference exists in a case with the high pressure ratio and low speed condition named as RDG 59. The thermodynamic efficiency difference is 2.88 points between test and the B2B calculation for this condition. This difference can be explained by CFD mixing plane method, the difference between C_p function and real gas behavior, uncertain cooling flow locations and distributions and natural behavior of B2B CFD calculation method on secondary flow effects. It can be investigated on 3D calculations deeply and the reason behind this difference can be completed as future work.

To design mid frame, parametric beta and thickness distributions are formulated. Corresponding shape coefficients are controlled by Latin Hypercubic Sampling method. Thanks to robustness of B2B CFD, method of formulation can be manipulated quickly with help of comparatively fast solution time than 3D CFD calculations. The final DoE study ended with 1100 runs in total. After feasibility filtering 1000 of them were examined at post-process. As seen from scatter plots in Chapter 4, loss characteristic of MF1 and MF2S configurations are extracted and they follow the global trends of Kacker-Okapuu's model on profile losses [26]. It is seen that both of frames are sensitive to high blade turning, high exit flow angle combined with pitch to chord or axial chord and low ratio of inlet and exit of Mach numbers. Also, reduced surrogate model is developed with quadratic least square method to investigate loss trends further. It is found that the splitter inlet angle relative to the corresponding metal angle of frame is affecting the pressure loss depending on cases. The detailed effects Reynolds Number on losses alone are not isolated as they're showing random distribution versus Y_p . Effect of Reynolds Number can be isolated with enriched surrogate model for further study.

A direct MF1-MF2S comparison is performed by integrating these designs into the benchmark turbine. Selected configurations for comparison represent similar behavior generally but locally their physics is different as seen on contour plots and loading

chart. The incidence tolerances of both configurations are improved as they're having relatively large leading-edge radius compared to E3 S2 row. Also, the low curvature at the front side of both frames helps the flow keep attached as it is seen from a low adverse pressure gradient on positive incidence. Both configurations are experiencing choking condition and shock losses at the throat section because of high turning at aft region and wide axial chord. Due to formulated set it is found that the MF2S configuration is sensitive to splitter position because of inlet angle offset and non-matching beta and thickness distributions which can open one side channel and introduce shock-loss risk. Aerodynamically, selected MF1 configuration has shown better characteristics as compared to the MF2S because its pressure loss coefficient is lesser. Nevertheless, both configurations have found promising designs and optimization study can improve aerodynamical characteristics of MF1 and MF2S which can be a follow up study. Additionally, mid frame designs can be integrated into the S shaped duct and can be cascade tested as future work.

Finally, this thesis shows that splitter integration is not directly beneficial in 2D design space. As designing frame with splitter, it naturally creates additional degrees of freedom which must be carefully controlled by designers. Splitters can improve characteristics and re-arrange loading distribution on frames or cause local shock and mixing penalties, so the turbine performance degrades.

REFERENCES

- [1] Dixon S. L., Hall C. A., (2014), “Fluid Mechanics and Thermodynamics of Turbomachinery”, 7th Edition, Butterworth-Heinemann.
- [2] Santner C., Göttlich E., Wallin F., Hoeger M., (2011), “Experimental investigation of turning mid turbine frame designs”, 20th International Symposium on Air Breathing Engines, (Accessed Date: 01/12/2025), <https://graz.elsevierpure.com/en/publications/experimental-investigation-of-turning-mid-turbine-frame-designs-3>.
- [3] Paradiso B., Santner C., Hubinka J., Göttlich E., Hoeger M., (2011), “Turning Mid Turbine Frame Behavior for Different HP Turbine Outflow Conditions”, Volume 7: Turbomachinery, Parts A, B, and C, 1761–1771, Vancouver, British Columbia, Canada, January.
- [4] Krajnc N., et al., (2025), “Experimental and Numerical Comparison of Two Turbine Vane Frame Designs”, Journal of Turbomachinery, 147 (8), 081005.
- [5] Yasa T., Lavagnoli S., Paniagua G., (2011), “Impact of a multi-splitter vane configuration on the losses in a 1.5 turbine stage”, Proceedings of the Institution of Mechanical Engineers, Part J: Journal of Power and Energy, 225 (7), 964–974.
- [6] Liu J., Liu H., Liu G., Du Q., Wang P., Chang S., (2021), “Investigation on the flow-control strategy for an aggressive turbine transition ducts”, Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, 235 (5), 589–599.
- [7] Xu Q., Wang P., Du Q., Liu J., Liu G., (2019), “Effects of axial length and integrated design on the aggressive intermediate turbine duct”, Proceedings of the Institution of Mechanical Engineers, Part J: Journal of Power and Energy, 233 (4), 443–456.
- [8] Orsenigo S., (2018), “Gradient-Based Aerodynamic Optimization of a Splittered Turbine Vane Frame Through the Adjoint Method”, M.Sc. Thesis, Politecnico di Milano.
- [9] Russo V., Orsenigo S., Mueller L., Verstraete T., Lavagnoli S., (2019), “Adjoint Based Aerodynamic Optimization of a Multi-Splitter Turbine Vane Frame”, Volume 2C: Turbomachinery, Phoenix, Arizona, USA, American Society of Mechanical Engineers, June.
- [10] Riebl S. F., Wakelam C., Niehuis R., (2021), “Adaptive Geometry Representation of Turbine Vane Frames for Use in Optimization”, Volume 2C: Turbomachinery, Virtual, Online, American Society of Mechanical Engineers, June.

- [11] Timko L. P., (1984), “Energy Efficient Engine high pressure turbine component test performance report”, Technical Report No: NASA-CR-168289, (Accessed Date: 02/12/2025), <https://ntrs.nasa.gov/citations/19900019237>.
- [12] Claus R. W., Beach T., Turner M., Siddappaji K., Hendricks E. S., (2015), “Geometry and Simulation Results for a Gas Turbine Representative of the Energy Efficient Engine (EEE)”, Technical Report No: E-18986, (Accessed Date: 02/12/2025), <https://ntrs.nasa.gov/citations/20150002712>.
- [13] Halila E. E., Lenahan D. T., Thomas T. T., (1982), “Energy efficient engine high pressure turbine test hardware detailed design report”, Technical Report No: NASA-CR-167955, (Accessed Date: 02/12/2025), <https://ntrs.nasa.gov/citations/19850002687>.
- [14] Sridhar M., Sunnam S., Goswami S., Liu J. S., (2011), “CFD Aerodynamic Performance Validation of a Two-Stage High Pressure Turbine”, Volume 7: Turbomachinery, Parts A, B, and C, 1175–1184, Vancouver, British Columbia, Canada, January.
- [15] Murari S., Sunnam S., Liu J. S., (2012), “Steady State and Transient CFD Studies on Aerodynamic Performance Validation of a High Pressure Turbine”, Volume 8: Turbomachinery, Parts A, B, and C, 2067–2077, Copenhagen, Denmark, June.
- [16] Miki K., Turner M., Wey T., Moder J., (2023), “LES Simulation of Cooling Airflow of High-Pressure Turbine using the Source Term Approach”, ASME-JSME-KSME Joint Fluids Engineering Conference 2023, Osaka, Japan, 9-13 July.
- [17] Wehrel P., Carvalho F., (2024), “Cooling Model Calibration in a Collaborative Turbine Preliminary Design Process Using the NASA Energy Efficient Engine - Part I: 0D Performance Modeling”, International Journal of Gas Turbine, Propulsion and Power Systems, 15 (3), 18–30.
- [18] Yang X., Cai H., Kang J., Liu W., Li P., (2023), “Influence of Hot Streak and Swirl Clocking Position on Aerothermal Performance of High-Pressure Turbine”, Aerospace, 10 (11), 934.
- [19] Zhang S., Ding S., Liu P., Qiu T., (2023), “Effects of swirl and hot streak on thermal performances of a high-pressure turbine”, Chinese Journal of Aeronautics, 36 (5), 250–267.
- [20] Farokhi S., (2014), “Aircraft Propulsion”, 1st Edition, John Wiley & Sons.
- [21] Aungier R. H., (2003), “Axial-Flow and Radial-Inflow Turbine Design and Analysis”, ASME Press.
- [22] Chen N., (2010), “Aerothermodynamics of Turbomachinery: Analysis and Design”, 1st Edition, Wiley.
- [23] Smith S. F., (1965), “A Simple Correlation of Turbine Efficiency”, Aeronautical Journal, 69 (655), 467–470.

- [24] Ainley D. G., Mathieson G. C. R., (1951), “A method of performance estimation for axial-flow turbines”, (Accessed Date: 02/12/2025), <https://reports.aerade.cranfield.ac.uk/handle/1826.2/3538>.
- [25] Craig H. R. M., Cox H. J. A., (1970), “Performance Estimation of Axial Flow Turbines”, *Proceedings of the Institution of Mechanical Engineers*, 185 (1), 407–424.
- [26] Kacker S. C., Okapuu U., (1982), “A Mean Line Prediction Method for Axial Flow Turbine Efficiency”, *Journal of Engineering for Power*, 104 (1), 111–119.
- [27] Zou Z., Wang S., Liu H., Zhang W., (2018), “Axial Turbine Aerodynamics for Aero-engines”, Springer Singapore.
- [28] Pritchard L. J., (1985), “An Eleven Parameter Axial Turbine Airfoil Geometry Model”, Volume 1: Aircraft Engine; Marine; Turbomachinery, Houston, Texas, USA, March.
- [29] Versteeg H. K., Malalasekera W., (2007), “An Introduction to Computational Fluid Dynamics: The Finite Volume Method”, Pearson Education Limited.
- [30] Wu C. H., (1952), “A General Theory of Three-Dimensional Flow in Subsonic and Supersonic Turbomachines of Axial, Radial, and Mixed-Flow Types”, *Journal of Fluids Engineering*, 74 (8), 1363–1380.
- [31] Jiang R., et al., (2025), “The application of bowing blades in a low aspect ratio integrated inter-turbine duct using splitters”, *Energy*, 321, 135430.
- [32] Bader P., Sanz W., Spataro R., Göttlich E., (2017), “Flow Evolution Through a Turning Midturbine Frame with Embedded Design”, *Journal of Propulsion and Power*, 33 (6), 1478–1488.

BIOGRAPHY

Eren Işık is graduated with Bachelor's Degree from Department of Mechanical Engineering at Kocaeli University. Eren is employed at TRMOTOR Power Systems as an Aerothermal Design Engineer since October 2021.



PUBLICATIONS AND PRESENTATIONS FROM THE THESIS

E. Işık, and S.Ö. Ünverdi, “Aerodynamic Design and Validation of a High Pressure Turbine Using 1D Methods and Blade-to-Blade CFD,” 9th Graduate Research Symposium of Gebze Technical University, 2025



APPENDICES

APPENDIX-A: Flow Angle and Mach Number Comparison

Table A.1: Stage 1 hub comparison.

Stage 1	E3	1D	2D	Check E3-2D
	Stage 1 Hub			Diff
S1 A1 [°]	0	0	0	0
S1 A2 [°]	73.1	73.71	73	0.1
R1 B2[°]	38.6	46.73	41.1	-2.5
R1 B3[°]	-65.6	-66.76	-64.2	-1.4
R1 A3[°]	-9.7	-17.53	-10.8	1.1
S1 Ma1	0.114	0.109	0.12	-0.01
S1 Ma2	0.91	0.897	0.93	-0.02
R1 Mr2	0.343	0.367	0.37	-0.03
R1 Mr3	0.745	0.77	0.79	-0.04
R1 Ma3	0.314	0.317	0.34	-0.03

Table A.2: Stage 1 mean comparison.

Stage 1	E3	1D	2D	Check E3-2D
	Stage 1 Mean			Diff
S1 A1 [°]	0	0	0	0
S1 A2 [°]	74.2	74.23	73.55	0.65
R1 B2[°]	43.2	39.31	39.36	3.84
R1 B3[°]	-66.9	-67.34	-65.5	-1.4
R1 A3[°]	-17.7	-14.35	-14.66	-3.04
S1 Ma1	0.109	0.109	0.12	-0.01
S1 Ma2	0.878	0.8504	0.89	-0.01
R1 Mr2	0.324	0.298	0.35	-0.02
R1 Mr3	0.819	0.786	0.82	0
R1 Ma3	0.338	0.314	0.35	-0.01

Table A.3: Stage 1 tip comparison.

Stage 1	E3	1D	2D	Check E3-2D
	Stage 1 Tip			Diff
S1 A1 [°]	0	0	0	0
S1 A2 [°]	75.4	74.6	73.25	2.15
R1 B2[°]	29.7	28.31	33.47	-3.77
R1 B3[°]	-64.4	-65.25	-65.22	0.82
R1 A3[°]	-5.2	-8.45	-7.98	2.78
S1 Ma1	0.084	0.109	0.12	-0.04
S1 Ma2	0.815	0.806	0.87	-0.06
R1 Mr2	0.22	0.243	0.32	-0.1
R1 Mr3	0.822	0.819	0.81	0.01
R1 Ma3	0.357	0.345	0.36	0

Table A.4: Stage 2 hub comparison.

Stage 2	E3	1D	2D	Check E3-2D
	Stage 2 Hub			Diff
S1 A1 [°]	-11.5	0	-13.2	1.7
S1 A2 [°]	69	71.03	68.75	0.25
R1 B2[°]	31.5	40.55	29.2	2.3
R1 B3[°]	-59.9	-61.61	-58.08	-1.82
R1 A3[°]	-4.6	-3.06	-5.64	1.04
S1 Ma1	0.249	0.317	0.28	-0.03
S1 Ma2	0.728	0.924	0.8	-0.07
R1 Mr2	0.286	0.395	0.3	-0.02
R1 Mr3	0.934	0.77	0.87	0.07
R1 Ma3	0.469	0.353	0.46	0.01

Table A.5: Stage 2 mean comparison.

Stage 2	E3	1D	2D	Check E3-2D
	Stage 2 Mean			Diff
S1 A1 [°]	-21.8	0	-17.16	-4.64
S1 A2 [°]	69	70.25	69.08	-0.08
R1 B2[°]	17	21.48	17.5	-0.5
R1 B3[°]	-59.8	-61.45	-59.1	-0.7
R1 A3[°]	1.5	2.15	2.22	-0.72
S1 Ma1	0.276	0.314	0.3	-0.03
S1 Ma2	0.828	0.845	0.84	-0.01
R1 Mr2	0.306	0.306	0.32	-0.01
R1 Mr3	0.836	0.786	0.82	0.01
R1 Ma3	0.421	0.378	0.43	-0.01

Table A.6: Stage 2 tip comparison.

Stage 2	E3	1D	2D	Check E3-2D
	Stage 2 Tip			Diff
S1 A1 [°]	-7.4	0	-10.47	3.07
S1 A2 [°]	69	72.03	69.27	-0.27
R1 B2[°]	-18.2	1.77	-12.3	-5.9
R1 B3[°]	-59.8	-61.03	-59.02	-0.78
R1 A3[°]	-1	5.84	1.8	-2.8
S1 Ma1	0.269	0.345	0.29	-0.03
S1 Ma2	0.892	0.788	0.84	0.05
R1 Mr2	0.339	0.243	0.31	0.03
R1 Mr3	0.724	0.819	0.83	-0.1
R1 Ma3	0.365	0.409	0.38	-0.02