

CRANFIELD UNIVERSITY

YIMIN YANG

THE IMPACT OF HEAT TRANSFER EFFECTS ON CIVIL
AIRCRAFT ENGINE TRANSIENT PERFORMANCE

SCHOOL OF AEROSPACE, TRANSPORT AND
MANUFACTURING

PhD in Aerospace
Academic Year: 2020 - 2024

Supervisor: Dr Theoklis Nikolaidis
Associate Supervisor: Prof Pericles Pilidis
Apr 2024

CRANFIELD UNIVERSITY

SCHOOL OF AEROSPACE, TRANSPORT AND
MANUFACTURING

PhD in Aerospace

Academic Year 2020 - 2024

YIMIN YANG

The Impact of Heat Transfer Effects on Civil Aircraft Engine
Transient Performance

Supervisor: Dr Theoklis Nikolaidis
Associate Supervisor: Prof Pericles Pilidis
Apr 2024

This thesis is submitted in partial fulfilment of the requirements for
the degree of PhD

© Cranfield University 2024. All rights reserved. No part of this
publication may be reproduced without the written permission of the
copyright owner.

ABSTRACT

During gas turbine transient manoeuvre, heat transfer occurs between the fluid and metal. This results in various heat transfer effects, including heat soakage, tip clearance, and change in component performance map. These will cause heat loss and change in component flow characteristic and efficiency, which further affect gas turbine transient performance. In this work, a comprehensive heat transfer model including heat soakage, tip clearance and compressor map modification has been developed. The proposed heat soakage model improves the current state-of-the-art model by establishing a comprehensive thermal network with the consideration of the combustor temperature distribution and cooling technologies including film cooling, internal cooling, and thermal barrier coatings. Additionally, the proposed novel compressor map modification model can derive numerical correlations for compressor maps based on movement of compressor speed line and map scaling, enabling the modification of adiabatic maps to non-adiabatic maps during transient simulations. It improves the current compressor map modification models by ensuring both flexibility and accuracy. The developed heat transfer model has been integrated into Cranfield gas turbine simulation platform Turbomatch, enhancing the realism of transient simulations. The accuracy of the proposed model has been validated against data from public sources, simulation platforms, and experimental results. A sensitivity analysis has also been conducted to assess the impact of various assumptions on heat flow rate estimation. The impact of heat transfer to overall engine's performance has been demonstrated by simulating transient operation of a turbojet and two turbofan engines to demonstrate the effects of heat transfer on gas turbine transient performance. Comparing with the conventional heat soakage method, the application of the improved models can capture a delay on engine's response beyond the one simulated by the existing methods. This is a result of considering the combustor temperature distribution and cooling technologies, not included in the conventional heat soakage models. For the impact of heat transfer effect on compressor characteristic and performance, a 4% reduction in compressor surge margin is observed during a hot reslam transient manoeuvre, as a result of movement in compressor speed line due to heat transfer effect.

Keywords:

Gas Turbine Transient Performance, Heat Transfer Effect, Heat Soakage, Tip Clearance, Compressor Map Modification

ACKNOWLEDGEMENTS

The completion of this work marks a significant milestone in my academic journey. I am deeply grateful to the individuals whose contributions and support have paved the way for the successful conclusion of this research.

First and foremost, I extend my heartfelt appreciation to my supervisors, Dr. Theoklis Nikolaidis and Prof. Pericles Pilidis, whose guidance and inspiration have been invaluable throughout my years of study. Without their technical expertise and unwavering encouragement, completing this project would have been exceedingly challenging.

I am indebted to my family for their unwavering love and steadfast support. Their belief in me has been a constant source of strength, instilling in me the confidence and determination needed to complete the project.

Special gratitude is extended to Mrs. Tong Wang and Dr. Hasani Aguirre, whose insights were instrumental in the development of the compressor mean line model.

I am also indebted to Cranfield University for providing the essential academic resources and support that facilitated the successful completion of this research.

Lastly, I extend my sincere appreciation to my friends: Mr. Shang Lyu, Dr. Quan Peng, Dr. Lichao Yang, Miss Jingru Xing, Miss Yue Chen, and Mr. Rang Zhao. Their kindness, support, and camaraderie have been a constant source of joy and inspiration throughout my life. Their presence has not only enriched my academic experience but also infused it with a sense of fulfilment and purpose.

To all those mentioned and countless others who have contributed in various ways, I extend my heartfelt gratitude. This achievement would not have been possible without your unwavering support and encouragement.

TABLE OF CONTENTS

ABSTRACT	i
ACKNOWLEDGEMENTS.....	iii
LIST OF FIGURES	viii
LIST OF TABLES	xiii
LIST OF EQUATIONS.....	xv
LIST OF ABBREVIATIONS	xxiii
1 INTRODUCTION	1
1.1 Background.....	1
1.2 Aim and Objectives	2
1.3 Contribution	3
1.4 Thesis Structure.....	5
2 Literature Review	9
2.1 Gas Turbine Transient Performance Simulation	9
2.1.1 White Box Approach.....	10
2.1.1.1 Linear Method.....	10
2.1.1.2 Non-Linear Method	11
2.1.2 Black Box Approach	19
2.1.2.1 Static Network (Feedforward)	20
2.1.2.2 Dynamic Network (Feedback).....	23
2.1.3 Summary.....	26
2.2 Heat Transfer Effect on Transient Performance	29
2.2.1 Heat Soakage Effect	30
2.2.2 Tip Clearance Effect.....	36
2.2.3 Component Characteristics Change Effect	39
2.3 Transient Simulation Platform.....	44
2.4 Research Gap Identification.....	51
2.5 Author's Publication	53
3 Methodology.....	55
3.1 Introduction	55
3.2 Component Geometry Identification.....	60
3.2.1 Compressor Geometry Identification	60
3.2.2 Turbine Geometry Identification	67
3.2.3 Combustor Geometry Identification	69
3.3 Heat Soakage Model	72
3.3.1 Component Heat Flow Rate Estimation	75
3.3.1.1 Temperature Distribution Identification	75
3.3.1.2 Heat Transfer Model for Low Biot Number Parts	77
3.3.1.3 Heat Transfer Model for High Biot Number Parts	82
3.3.2 Establishment of Quasi-2D Thermal Network	94
3.3.3 Component Performance Modification	96

3.3.4 Component Initial Temperature Identification	96
3.4 Tip Clearance	97
3.4.1 Thermal and Mechanical Deflections	99
3.4.1.1 Blades.....	99
3.4.1.2 Disc.....	100
3.4.1.3 Casing	103
3.4.2 Component Performance Modification	104
3.4.2.1 Total deflection	104
3.4.2.2 Efficiency Prediction	104
3.5 Compressor Map Modification	105
3.5.1 Compressor Mean Line Model Establishment.....	107
3.5.1.1 Input data for the compressor mean line model.....	107
3.5.1.2 Performance calculation	110
3.5.1.2.1 Calculation of Velocity Triangles	110
3.5.1.2.2 Calculation of Blade Pressure Loss	118
3.5.1.2.3 Calculation of Deviation Angle	126
3.5.1.3 Speed line Identification.....	128
3.5.2 Heat Transfer Effect on Mean Line Model.....	129
3.5.3 Map Modification Model	130
3.6 Model Implementation to Turbomatch.....	131
4 Validation.....	133
4.1 Turbomatch off-design performance validation	133
4.2 Turbomatch transient performance validation.....	139
4.2.1 Engine Model Identification	139
4.2.2 Component Map Identification.....	140
4.2.3 Map Scaling Point Selection.....	146
4.2.4 Nozzle Outlet Area Calculation	147
4.2.5 Transient Performance Validation	148
4.2.5.1 Transient Fuel Schedule Setting.....	148
4.2.5.2 Burner Time Constant Setting.....	149
4.2.5.3 Transient Outcome Validation.....	149
4.3 Geometry Identification Model Validation.....	151
4.4 Heat Soakage Model Validation.....	161
4.4.1 Heat Flow Rate Estimation Model Validation	161
4.4.2 Heat Soakage Effect on Transient Performance Model Validation	163
4.5 Tip Clearance Model Validation	167
4.6 Mean Line Model Validation.....	171
4.6.1 Adiabatic Mean Line Model Validation	171
4.6.2 Non-Adiabatic Mean Line Model Validation.....	172
5 Case Study.....	175
5.1 Quantifying Heat Flow Rate Sensitivity to Model Inputs.....	175

5.2 Quantifying Combustor Temperature Distribution and Cooling Technologies Effect on Heat Flow Rate and Transient Performance	183
5.2.1 Effect of combustor temperature profile and cooling technologies on combustor heat flow rate	183
5.2.2 Heat flow rate variation effect on gas turbine transient performance	185
5.3 Heat Soakage and Tip Clearance Effect on Gas Turbine Transient Performance	189
5.3.1 Gas Turbine Engine Model and Design Point Parameters	189
5.3.2 Engine Transient manoeuvre Parameter.....	191
5.3.3 Geometry Identification of the Aircraft Engine Model	192
5.3.4 Heat Flow Rate and Tip Clearance Change.....	195
5.3.5 Heat Soakage and Tip Clearance Change Effect on Transient Performance.....	198
5.4 Map Modification Case Study	200
5.4.1 Mean Line Model Outcome	200
5.4.2 NASA E3 HPC Map Midification.....	202
5.4.3 Transient Performance Comparison.....	206
5.4.3.1 Engine Structure and Design Point Parameters	206
5.4.3.2 Engine Transient manoeuvre Parameters	208
5.4.3.3 Engine Transient Performance with and without map modification	209
6 Conclusion.....	213
7 Recommendation for Future Work	218
7.1 Burner Time Constant.....	218
7.2 Model Validation	218
7.3 Geometry Identification Model	219
7.4 Combustor System Architecture	221
7.5 Heat Soakage Model Sensitivity Analysis	222
7.6 Heat Soakage Model	223
7.7 Tip Clearance Movement Model.....	224
7.8 Compressor Map Modification Model.....	225
REFERENCES.....	227
APPENDICES	249

LIST OF FIGURES

Figure 2-1 Flow chart of the CMF method	12
Figure 2-2 ICV method illustration	15
Figure 2-3 The comparison of operating trajectories for CMF and ICV method	18
Figure 2-4 A simple structure of an MLP NN with input, hidden, and output layers	22
Figure 2-5 A simple structure of an RBF NN with input, hidden, radial basis function and output layers.....	23
Figure 2-6 A schematic of a typical NARX network	24
Figure 2-7 Heat transfer effects on the performance of the components	30
Figure 2-8 Change in boundary layer development due to heat transfer [109].	40
Figure 2-9 Compressor speed line movement due to boundary layer effect and density change [106]	41
Figure 2-10 Example schematic diagram of a composite compressor map used to represent the unsteady properties of velocity and stall lines during transient with heat transfer [118]	43
Figure 2-11 Gas turbine model based on NPSS software [133]	46
Figure 2-12 TURBOMATCH interface	47
Figure 2-13 Types of Engines in GasTurb (GasTurb 12) [140].....	48
Figure 2-14 GSP window with a turbojet engine model [142]	49
Figure 2-15 Multidisciplinary aircraft system simulation with PROOSIS [145] ..	50
Figure 3-1 Workflow of the Overall Project	57
Figure 3-2 Cross Sectional Diagram of Gas Turbine Engine [152].....	60
Figure 3-3 Equivalent compressor geometry model	61
Figure 3-4 Different Compressor Geometries.....	62
Figure 3-5 Calculation Example of the Annulus Dimension of Compressor	65
Figure 3-6 Equivalent turbine geometry model.....	67
Figure 3-7 Combustor Equivalent Configuration.....	69
Figure 3-8 Structure illustration of annular combustor	71
Figure 3-9 Heat soakage estimation logic during engine transient	74
Figure 3-10 Temperature and Pressure Profile of Compressor	76

Figure 3-11 Flow chart for combustor zone temperature calculation.....	77
Figure 3-12 Heat Quantity Estimation Using Fixed Temperature [69]	79
Figure 3-13 Heat Quantity Estimation Using Exponential Decay [69].....	80
Figure 3-14 CVM calculation process (turbine disc as an example).....	82
Figure 3-15 Component geometry feature capture and mesh	87
Figure 3-16 Mesh outcome (turbine disc as an example).....	87
Figure 3-17 Mesh density analysis	88
Figure 3-18 Quasi-2D thermal network illustration	95
Figure 3-19 Tip clearance estimation logic during engine transient.....	98
Figure 3-20 Blade centrifugal force calculation.....	100
Figure 3-21 Compressor map modification model calculation logic.....	106
Figure 3-22 compressor flow passage geometry.....	108
Figure 3-23 Compressor cascade notation [1].....	108
Figure 3-24 Axial compressor stage nomenclature	111
Figure 3-25 Losses obtained in an axial flow compressor [155].....	119
Figure 3-26 Profile Loss Parameter with Variation in Mach number [159].....	123
Figure 4-1 Energy Efficiency Engine Configuration	133
Figure 4-2 Energy efficiency engine Turbomatch model	134
Figure 4-3 Schematic of the one-spool turbojet engine	140
Figure 4-4 99 NASA TM 101433 example compressor map	141
Figure 4-5 99 AGARD two-stage turbine map	142
Figure 4-6 Interpolated compressor pressure ratio map.....	143
Figure 4-7 Interpolated compressor efficiency map.....	143
Figure 4-8 Interpolated turbine mass flow map	145
Figure 4-9 Interpolated turbine efficiency map	145
Figure 4-10 Nozzle discharge coefficient map [140].....	147
Figure 4-11 Transient Fuel Schedule for Gas Turbine Transient Performance	149
Figure 4-12 Transient Performance Comparison between Turbomatch and GasTurb.....	151

Figure 4-13 Comparison between E3 compressor and simulation outcome ..	153
Figure 4-14 Difference between actual compressor geometry and simulation outcome.....	155
Figure 4-15 Geometry comparison between Turbomatch and GasTurb	159
Figure 4-16 Heat flow rate comparison between TM and GasTurb	163
Figure 4-17 Gas turbine transient performance comparison between TM and GasTurb with and without heat soakage effect.....	166
Figure 4-18 Fuel schedule for tip clearance validation against GasTurb	168
Figure 4-19 Turbine tip clearance change comparison between GasTurb and Turbomatch	168
Figure 4-20 The geometry of the turbine rotor in Kumar's research [163]	169
Figure 4-21 Tip clearance variation comparison.....	170
Figure 4-22 Flow passage for NASA stage 35 [164]	171
Figure 4-23 Comparison between the developed mean line model and NASA experiment data.....	172
Figure 4-24 3D CFD model of NASA rotor 35 developed by Shah [165].....	173
Figure 4-25 Comparison of the heat transfer effect on compressor map	174
Figure 5-1 Sensitivity analysis of dynamic viscosity on heat flow rate.....	177
Figure 5-2 Sensitivity analysis of film cooling efficiency on heat flow rate	179
Figure 5-3 Sensitivity analysis of TBC thickness on heat flow rate.....	180
Figure 5-4 Sensitivity analysis of luminosity factor on heat flow rate	181
Figure 5-5 Combustor temperature profile distribution and film cooling effect on heat transfer process	185
Figure 5-6 Heat flow rate change effect on combustor outlet temperature	186
Figure 5-7 Heat flow change effect on gas turbine transient performance	188
Figure 5-8 Cross section of CFM 56-7B turbofan engine [166]	190
Figure 5-9 Fuel flow schedule for the two spool turbofan engine	192
Figure 5-10 Component flow path geometry outcome.....	192
Figure 5-11 Heat flow rate in several components	195
Figure 5-12 Fan tip clearance change and efficiency change	196
Figure 5-13 Booster tip clearance change and efficiency change	197

Figure 5-14 HPC tip clearance change and efficiency change	197
Figure 5-15 HPT and LPT tip clearance change and efficiency change	198
Figure 5-16 Spool corrected rotational speed change	199
Figure 5-17 Engine thrust and SFC change	200
Figure 5-18 E3 Engine HPC IGV and Stator Schedule	202
Figure 5-19 E3 Engine HPC Mean Line Model Outcome	202
Figure 5-20 Normalized heat transfer rate distribution in the compressor mean line model	203
Figure 5-21 Heat Transfer Effect on Speed Line	203
Figure 5-22 Scaling factor for mass flow and pressure ratio under various q^*	205
Figure 5-23 Relation between slope of scaling factor lines and rotational speed	206
Figure 5-24 Schematic of the two-spool turbofan engine	207
Figure 5-25 Transient Fuel Schedule for Two Spool Turbofan Engine Transient Performance	209
Figure 5-26 Non-dimensional heat transfer rate during transient manoeuvre	210
Figure 5-27 High pressure compressor working line comparison.....	210
Figure 7-2 Logic flow of Falck's compressor geometry identification method.	220
Figure 7-3 Parameterized disc model.....	221
Figure 7-4 Temperature distribution of the uniform and non-uniform disc	223
Figure 7-5 Temperature distribution along the radius direction	224
Figure 7-6 Heat flow rate comparison between uniform and non-uniform disc	224
Figure C-1 Example of .hhs title section.....	251
Figure C-2 Turbomatch .ttr file original input sequence	256
Figure C-3 Flow chart for .hhs file reading and storing process	258
Figure C-4 Flow chart for heat transfer effect calculation process.....	260
Figure D-1 Burner time constant effect on gas turbine transient performance	263
Figure E-1 Fan tip clearance movement	265
Figure E-2 Booster tip clearance movement	266
Figure E-3 HPC tip clearance movement	268

Figure E-4 HPT tip clearance movement 268

Figure E-5 LPT tip clearance movement 269

LIST OF TABLES

Table 2-1 Comparison between different transient simulation methodologies .	27
Table 2-2 Comparison between different heat soakage model	35
Table 2-3 Comparison between different component map modification model	43
Table 2-4 Comparison between different gas turbine simulation platform	50
Table 3-1 Comparison between proposed model and other models	58
Table 3-2 Input data during the compressor geometry identification	61
Table 3-3 Input data during the turbine geometry identification.....	68
Table 3-4 Input data during the combustor geometry identification	69
Table 3-5 Summary of the quasi-2D thermal network	95
Table 3-6 Geometry Data for Mean Line Model	108
Table 3-7 Working Condition Data for Compressor Mean Line Model	109
Table 4-1 Energy efficiency engine parameters	134
Table 4-2 Design point performance comparison.....	136
Table 4-3 Off design (Maximum Cruise) performance comparison	137
Table 4-4 Off design (Take Off) performance comparison	138
Table 4-5 One-spool turbojet engine parameters (Sea Level, Static, ISA condition)	140
Table 4-6 Scaling Factor in Both Software	146
Table 4-7 Energy efficiency engine compressor parameter	152
Table 4-8 Geometry Data.....	156
Table 4-9 Wetted area and mass of component for Turbomatch and GasTurb	159
Table 5-1 Sensitivity analysis: impact of assumed variables on estimation outcome.....	182
Table 5-2 Design point data for the two-spool turbofan engine	190
Table 5-3 Rotor moment of inertia and the internal component volume parameter	191
Table 5-4 Component mass and wetted area outcome	193
Table 5-5 Parameters of NASA E3 Engine High Pressure Compressor	201

Table 5-6 two spool turbofan engine parameters (ISA+10, 10668m, 0.8 Mach)	207
Table 7-2 Specific Parameters for Falck's Method	219
Table C-1 Compressor heat transfer analysis data	252
Table C-2 Combustor heat transfer analysis data	254
Table C-3 Turbine heat transfer analysis data.....	254
Table C-4 Added transient .ttr file input sequence.....	257
Table D-1 Burner time constant sensitivity analysis outcome.....	264

LIST OF EQUATIONS

(3-1)..... 63

(3-2)..... 63

(3-3)..... 63

(3-4)..... 64

(3-5)..... 64

(3-6)..... 64

(3-7)..... 64

(3-8)..... 64

(3-9)..... 64

(3-10)..... 65

(3-11)..... 65

(3-12)..... 65

(3-13)..... 66

(3-14)..... 66

(3-15)..... 66

(3-16)..... 66

(3-17)..... 66

(3-18)..... 66

(3-19)..... 66

(3-20)..... 66

(3-21)..... 66

(3-22)..... 70

(3-23)..... 70

(3-24)..... 70

(3-25)..... 71

(3-26)..... 71

(3-27)..... 71

(3-28)..... 71

(3-29).....	72
(3-30).....	72
(3-31).....	72
(3-32).....	72
(3-33).....	76
(3-34).....	76
(3-35).....	78
(3-36).....	79
(3-37).....	80
(3-38).....	80
(3-39).....	81
(3-40).....	81
(3-41).....	81
(3-42).....	81
(3-43).....	81
(3-44).....	83
(3-45).....	83
(3-46).....	83
(3-47).....	83
(3-48).....	84
(3-49).....	84
(3-50).....	84
(3-51).....	84
(3-52).....	85
(3-53).....	85
(3-54).....	88
(3-55).....	88
(3-56).....	89
(3-57).....	89

(3-58).....	90
(3-59).....	90
(3-60).....	90
(3-61).....	90
(3-62).....	91
(3-63).....	91
(3-64).....	91
(3-65).....	91
(3-66).....	92
(3-67).....	92
(3-68).....	93
(3-69).....	93
(3-70).....	93
(3-71).....	93
(3-72).....	94
(3-73).....	94
(3-74).....	96
(3-75).....	96
(3-76).....	96
(3-77).....	99
(3-78).....	100
(3-79).....	100
(3-80).....	101
(3-81).....	101
(3-82).....	101
(3-83).....	101
(3-84).....	101
(3-85).....	102
(3-86).....	102

(3-87).....	102
(3-88).....	102
(3-89).....	102
(3-90).....	102
(3-91).....	103
(3-92).....	103
(3-93).....	103
(3-94).....	104
(3-95).....	104
(3-96).....	104
(3-97).....	104
(3-98).....	105
(3-99).....	105
(3-100).....	105
(3-101).....	108
(3-102).....	110
(3-103).....	110
(3-104).....	112
(3-105).....	112
(3-106).....	112
(3-107).....	112
(3-108).....	112
(3-109).....	112
(3-110).....	112
(3-111).....	112
(3-112).....	113
(3-113).....	113
(3-114).....	113
(3-115).....	113

(3-116).....	113
(3-117).....	113
(3-118).....	113
(3-119).....	114
(3-120).....	114
(3-121).....	114
(3-122).....	114
(3-123).....	114
(3-124).....	114
(3-125).....	114
(3-126).....	114
(3-127).....	114
(3-128).....	115
(3-129).....	115
(3-130).....	115
(3-131).....	116
(3-132).....	116
(3-133).....	116
(3-134).....	116
(3-135).....	117
(3-136).....	117
(3-137).....	117
(3-138).....	117
(3-139).....	117
(3-140).....	117
(3-141).....	117
(3-142).....	117
(3-143).....	117
(3-144).....	117

(3-145).....	118
(3-146).....	118
(3-147).....	118
(3-148).....	118
(3-149).....	118
(3-150).....	118
(3-151).....	119
(3-152).....	120
(3-153).....	120
(3-154).....	120
(3-155).....	121
(3-156).....	121
(3-157).....	121
(3-158).....	121
(3-159).....	121
(3-160).....	122
(3-161).....	122
(3-162).....	122
(3-163).....	123
(3-164).....	123
(3-165).....	124
(3-166).....	125
(3-167).....	125
(3-168).....	126
(3-169).....	126
(3-170).....	127
(3-171).....	127
(3-172).....	127
(3-173).....	127

(3-174).....	127
(3-175).....	127
(3-176).....	128
(3-177).....	128
(3-178).....	129
(3-179).....	129
(3-180).....	129
(3-181).....	130
(3-182).....	130
(3-183).....	130
(3-184).....	130
(3-185).....	130
(4-1).....	144
(4-2).....	147
(4-3).....	147
(5-1).....	206
(5-2).....	206
(7-1).....	221
(A-1).....	249
(A-2).....	249
(A-3).....	249
(A-4).....	249
(A-5).....	249
(A-6).....	250
(A-7).....	250
(A-8).....	250

LIST OF ABBREVIATIONS

A	Area (m^2)
A^*	Minimum loss area contraction ratio
a	Speed of Sound (m/s)
ANN	Artificial Neural Networks
APR	Aspect Ratio
ATEST	Advanced Turbine Engine Simulation Technique
C	Constant
c	Chord (m)
CMF	Constant Mass Flow
CNN	Convolutional Neural Networks
C_p	Specific Heat ($\text{J/kg}\cdot\text{K}$)
CVM	Control Volume Method
D	Diameter (m)
E	Young's Modulus (Pa)
FAR	Fuel to Air Ratio
FEM	Finite Element Method
GSP	Gas turbine Simulation Program
H	Enthalpy (J)
h	Blade Height (m) / Heat Transfer Coefficient ($\text{W/m}^2/\text{K}$)
HPC	High Pressure Compressor
HPT	High Pressure Turbine
HTC	Heat Transfer Coefficient
i	Time Step / Incidence angle ($^\circ$)
ICV	Inter-Component Volume
IMF	Imbalanced Mass Flow
IRM	Impulse Response Model
L	Length (m)
LHV	Lower Heating Value
LPM	Lumped Parameter Method
mGT	Micro Gas Turbine
M	Mach Number
m	Mass (kg)

MLP	Multi-Layer Perceptron
N	Rotational Speed (rpm) / Number of items
NARX	Non-linear Autoregressive Exogenous
NN	Neural Networks
NPSS	Numerical Propulsion System Simulation
P	Total Pressure (Pa)
p	Static Pressure (Pa)
PCN	Rotational Speed
PCR	Pitch Chord Ratio
PDF	Pressure Drop Factor
PW	Pressure Wave
PWV	Pressure Wave in Volume
Q	Flow Function / Quantity of Heat
q^*	Non-dimensional Heat Transfer Rate
R	Gas Constant (J/kg K)
r	Radius (m)
RBF	Radial Basis Function Neural Networks
RMSE	Root Mean Square Relative Error
RNN	Recurrent Neural Networks
RSM	Replacement Structure Model
s	Blade Spacing (m)
SF	Scaling Factor
SL	Stage Length
SSM	State-Space Model
T	Total Temperature (K)
TM	Turbomatch
TDM	Temperature Difference Model
Th	Thickness (m)
t	Time (s) / Static Temperature (K)
t_{max}	Maximum tip clearance (m)
U	Blade peripheral speed (m/s)
u	Component Thermal Displacement (m)
V	Velocity (m/s)

W	Mass Flow (kg/s) / Velocity in Relative Frame (m/s)
w	Axial Width of the Node (m)

Greek letters

α	The Coefficient of Thermal Expansion / Flow Absolute Angle (°)
α'	Blade Angle (°)
β	Flow Relative Angle (°)
β_m	Mean air angle (°)
η	Efficiency
τ	Time Constant
γ	Ratio of Specific Heats
Φ	Heat Flow Rate (kW)
ϕ	Flow coefficient
λ	Conductive Coefficient (W/(m K))
σ	Stefan-Boltzmann Constant (W/(m ² K ⁴)) / Thermal Stress / Blade Solidity
ε	Emissivity
ε^*	Flow deflection angle (°)
ω	Rotational Speed (rpm) / Pressure Loss Coefficient
ν	Component Elastic Displacement (m)
μ	Poisson's ratio
ψ	Blade loading coefficient
δ	Deviation Angle (°)
δ^*	Minimum-loss deviation angle
Γ	Minimum loss circulation parameter
θ	Blade camber angle (°)

1 INTRODUCTION

1.1 Background

Gas turbine engines are designed to operate in a wide range of operating conditions, especially for aircraft propulsion units. In the steady state, fixing the fuel flow will fix the rotational speed and all other gas turbine parameters. This is primarily due to the power balance between the compressor and turbine. However, this power balance will be broken when the gas turbine is accelerating or decelerating to its targeted condition. And all other engine parameters will be different from the steady-state condition. Therefore, the concept of transient performance is introduced, which deals with the working state of engine performance parameters as a function of time.

There are always several phenomena accompanying transient operation, among them, one important phenomenon is the heat transfer effect. The heat transfer effect refers to the effect caused by the heat transfer between the working fluid and the engine metal. It will affect the component's performance in three ways: the heat soakage effect, the tip clearance change effect and the component map change effect.

The current state-of-the-art heat transfer model primarily relies on a 0D (zero-dimensional) approach. It simplified the combustor heat transfer model by neglecting the temperature distribution and does not incorporate cooling technologies such as film cooling internal cooling and thermal barrier coating, as highlighted in Table 2-2. Additionally, research on component map modification is limited in the public domain. As discussed in Section 2.2.3, there is a clear need for a more accurate and convenient map modification model. Furthermore, as reviewed in Section 2.3, existing commercial simulation platforms like GasTurb and GSP, only include simplified 0D heat soakage and tip clearance models. None of the current commercial software considers the heat transfer effects in the component maps.

Therefore, in this project, the research area focuses on the development of a comprehensive heat transfer effect model that can be used to modify gas turbine

transient performance. The heat transfer model includes the heat soakage model, tip clearance model and the compressor map modification model. The heat soakage model can estimate the heat transfer rate between the working fluid and the engine metal during gas turbine transient simulation, with consideration of the combustor temperature distribution and advanced cooling technologies. The tip clearance model can estimate the compressor and turbine tip clearance change during transient simulation and modifying the component's performance. The compressor map modification model can modify the compressor pressure ratio and corrected mass flow during transient simulation.

1.2 Aim and Objectives

The aim of this project is to investigate the impact of heat transfer effect on gas turbine transient performance.

To achieve the aim, several major objectives are defined:

1. Development of a comprehensive heat transfer effect model.

As described above, the heat transfer will affect gas turbine transient performance in three major ways: the heat soakage effect, the tip clearance change effect and the component map change effect. The developed model needs to be capable of estimating all three effects during gas turbine transient manoeuvre. It also required to provide great flexibility so it can be easily used by different gas turbine engine model. A quasi 2-dimensional model based on the cross-section area of gas turbine components is selected during this process because it can have a better prediction on temperature distribution along gas turbine axial and radius direction compared with 0D model and required less simulation time and necessary data compared with full 3D model during transient simulation. The quasi-2D model is a combination of 0D and 1D model. For each compressor blade and casing, they are modelled in 0D. For each compressor disc, flame tube, and turbine blade, casing and disc, they are modelled in 1D. So, it is defined as a quasi-2D model instead of a full 2D model.

2. The validation of the proposed heat transfer model.

With the developed heat transfer model, the effect of heat transfer effect on gas turbine transient performance can be estimated. However, it is very important to ensure the accuracy of the proposed model. In this process, data from open literature, commercial simulation platform and experiment are used to compare with the simulation outcome of the heat transfer effect model. As long as the difference between the outcome from the proposed model and the validation data is within tolerance, the model accuracy is guaranteed.

3. The investigation of heat transfer effect on gas turbine transient performance

Based on the validated heat transfer model, the heat transfer effect on gas turbine transient performance can be investigated. The effect of considering combustor temperature profile and cooling technologies on heat soakage estimation and gas turbine transient performance is investigated. the comparison between heat soakage effect and tip clearance effect is investigated. The effect of component map change effect on compressor working line is investigated.

1.3 Contribution

The major contribution of this project to knowledge can be concluded in two main points.

Firstly, a comprehensive novel high flexibility quasi-2D thermal network based heat soakage model that can be used during gas turbine transient simulation is developed. The project improves the existing heat soakage model available in public domain. Firstly, temperature difference model is developed, which can be used for compressor blade and casing. This model considers the fluid temperature change during heat transfer process, which reflects the physics better than conventional lumped parameter method. Secondly, control volume method that can be used for compressor disc, combustor flame tube, and turbine is developed. This model considers film cooling, internal cooling and thermal barrier coating into consideration, which are neglected in other state-of-art researchers' heat soakage model and commercial simulation software. Thirdly, combustor temperature distribution model that is suitable for conventional rich burn combustor is developed. Compared with the current state-of-art combustor

heat soakage model that based on mean combustor temperature, the proposed model offers a more realistic heat transfer process prediction by considering combustor temperature distribution into consideration.

Secondly, a novel simulation model is developed for compressor map modification. Leveraging the mean line model and heat transfer effect correlation, this model enables the creation of non-adiabatic compressor performance maps for gas turbine transient simulation. By using map scaling, the difference between adiabatic and non-adiabatic compressor map can be represented by the numerical correlations based on corrected rotational speed, pressure ratio, corrected mass flow, and heat flow rate. The numerical correlations can be applied to different transient manoeuvre (when compressor configuration kept same) instead of repetitively develop non-adiabatic performance map for different transient manoeuvre. Compared with the state-of-art compressor heat transfer effect analysis, the developed model ensures the accuracy and flexibility, which solves the weak point of MacCallum and Kiss's work.

A minor contribution has been made to tip clearance estimation. A novel thermal displacement model has been developed for the casing and disc, suitable for the proposed control volume method. This model accounts for the uneven temperature distribution in the casing and disc, producing a more realistic thermal displacement compared to conventional OD models. However, since the model's dimensionality is lower than that of state-of-the-art thermal displacement models (2D), it is considered a minor contribution rather than a major one.

Besides the contribution to knowledge, this project also makes contributions to simulation tool Turbomatch. The proposed model offers a systematically insight on the heat transfer effect on gas turbine transient performance by considering heat soakage effect, tip clearance changes effect and component map change effect. The comprehensive heat transfer model has been integrated into Cranfield software Turbomatch and enhances Turbomatch's capability for heat soakage, tip clearance and component map modification simulation. The developed model can be applied to different gas turbine model and is applicable across various gas turbine transient simulations. Compared with the state of art gas turbine

simulation tools, Turbomatch is the only software that includes heat soakage analysis, tip clearance changes analysis and compressor map change analysis.

1.4 Thesis Structure

This manuscript is divided into 7 Chapters, each one subdivided into several Sections and Subsections.

Chapter 2 - Literature Review

This chapter presents a comprehensive literature review conducted within the scope of this investigation. It focuses on three parts closely aligned with the objectives of this research. The first part discusses the development of gas turbine transient performance simulation methods. While the project may not directly involve the development of transient simulation methods, it relies on transient simulation. Therefore, understanding how gas turbine transient performance is simulated is crucial. The second part concerns the development of heat transfer effect simulation methods during gas turbine transient simulation. This includes three main aspects: heat soakage effect, tip clearance change effect, and component map change effect. The third part revolves around gas turbine performance simulation software. This section provides a comparative analysis among different commercial software solutions. The final part involves the identification of the research gap in heat transfer effects on gas turbine transient simulation field.

Chapter 3 - Methodology

This chapter encompasses the development methodology of the proposed heat transfer effect model during gas turbine transient simulation. The aim of this chapter is to provide a detailed explanation of the theoretical background of the heat transfer effect on gas turbine transient performance. Four main models are included in the methodology development process. Firstly, the geometry identification methods for different gas turbine components are discussed in Section 3.2. Secondly, the proposed heat soakage models for both low Biot number and high Biot number components are discussed in Section 3.3. This section also covers the establishment of a quasi-2D thermal network and the

method for modifying component performance. Thirdly, the development of the tip clearance models for rotating parts such as the compressor and turbine are discussed in Section 3.4. It addresses both elastic and thermal displacements for blades, casing, and discs. Finally, the determination of the map modification method is discussed in Section 3.5. This involves two steps: establishing the compressor off-design performance map based on the mean line model and considering the heat transfer effect on the compressor off-design performance. Using the mean line model, the variation in compressor pressure ratio and mass flow due to heat transfer effect is analyzed, and a numerical correlation between rotational speed, pressure ratio, mass flow, and heat flow rate is established.

Chapter 4 – Validation

In this chapter, the proposed models described in Chapter 3 are validated against experimental and open literature data. All proposed models undergo validation. Section 4.2 focuses on validating the steady-state and transient simulation accuracy of Turbomatch. The steady-state performance is validated against the real gas turbine data. For the transient performance, it is validated against the commercial software GasTurb due to the lack of experiment validation material in public domain. In Section 4.3, the proposed geometry identification model is firstly compared with the real compressor design data. Because the real component mass and wetted area data are not offered, the model is further validated against the outcomes from GasTurb. Section 4.4 validates the proposed heat soakage model against GasTurb data. Because the heat soakage model used by GasTurb is different from the proposed model, the boundary conditions of the proposed heat soakage model is controlled to keep the consistency of the heat transfer process. Section 4.5 validates the proposed tip clearance model against data from open literature. In Section 4.6, the accuracy of the proposed compressor mean line model is validated against NASA experimental data and open literature data.

Chapter 5 – Case Study

In this chapter, a sensitivity analysis is conducted to investigate the impact of small changes in model assumptions on the magnitude of heat flow predictions.

Then, several case studies based on the proposed heat soakage model, tip clearance model, and map modification model are presented. In section 5.2, the comparison between the traditional LPM and the proposed quasi-2D thermal network is discussed. The significance of considering the combustor temperature distribution and cooling technologies is emphasized. In Section 5.3, the effect of heat soakage and tip clearance change on an advanced two-spool civil turbofan engine is illustrated. In Section 5.4, the map modification model based on the NASA energy efficiency engine high-pressure compressor is detailed. The correlations between rotational speed, pressure ratio, mass flow, and heat flow rate are investigated. Besides, the transient performance of a two-spool turbofan engine with the same structure as the E3 engine is investigated using the map modification model.

Chapter 6 – Conclusion

In this Chapter, the conclusions and achievements of this study is discussed.

Chapter 7 – Future Work

In this Chapter, the limitation of the proposed model is discussed, including geometry identification model, heat soakage model, tip clearance movement model, and compressor map modification model. Based on the discussed limitation, the potential improvement is also discussed.

2 Literature Review

In this section, a comprehensive literature review related to gas turbine transient performance, the heat transfer effect on transient performance, and the transient simulation platform is illustrated. While the development of transient simulation methods is not within the scope of this project, it is crucial to note that the heat transfer effect relies on transient simulation. Therefore, understanding the gas turbine transient simulation method remains essential. Consequently, this literature review will encompass the current state of transient simulation methodologies.

2.1 Gas Turbine Transient Performance Simulation

The change in engine performance as a result of altering the fuel flow is referred to as transient operation. This process can last anywhere from a few seconds in the case of aero engines to several hours for large industrial gas turbines [1]. Since civil aero engines need to meet the certification requirement and the starting time of a large industry turbine is a critical economic indicator, the transient performance plays a crucial role in gas turbine analysis.

Over the past 70 years, extensive research has been conducted on the development and improvement of gas turbine transient performance prediction. Many comprehensive surveys of this research have been published, including those by Fawke and Saravanamuttoo, Szuch, Evans and Sanghi [2–5]. These surveys provide a broad overview of the development of transient simulation from 1950 to 2000. Based on the comprehensive review, the gas turbine overall transient methodology can be broadly classified into two categories: the white box and the black box approach [6]. The fundamental ideas of these two approaches are different. The white box approach involves using physics-based models, where the dynamic equations, thermodynamic relationships, and energy balance are considered. On the other hand, the black box approach involves the use of the ANN, machine learning, or deep learning techniques when direct access to the complex dynamic equations of the system is not feasible or when

working with them is challenging [7]. In this section, the development of the white box, black box and numerical methodologies will be discussed.

2.1.1 White Box Approach

White box models usually represent the physics-based models or mathematical models that deal with coupled and complicated dynamic equations, thermodynamic relationships, energy balance, and linearization methods [7]. According to the published resources, the white box method can be broken down into linear and non-linear methods. In this section, they will be discussed in chronological order, tracing the progression of its methodology development, as well as related transient investigation work.

2.1.1.1 Linear Method

The analysis and experimental study of the dynamic characteristics of gas turbines began in the early 1950s [8] and aimed at meeting engine control needs. Researchers from National Advisory Committee for Aeronautics, Otto and Taylor, were the first to investigate the rotor speed response of a single-spool turbojet engine with a step change in fuel flow using a first-order system [9]. Because the accelerating torque can be represented as a function of fuel flow and spool rotational speed, they used a linear differential equation obtained from the steady state to represent the acceleration behavior with a speed change range of less than 600 rpm.

Then it was realized that the engine dynamic performance could be predicted from the steady state data, and the calculation of the rotor-speed time constant became the focal point of the research. Gold and Rosenzweig developed a method for estimating the rotor-speed time constant for both turbojet and turbine-propeller engines [10]. For turbojet engines, the method required the variation of steady-state air flow and steady-state compressor temperature rise with engine speed and the polar moment of inertia of an engine. For turbine-propeller engines, additional propeller torque data was needed. The results showed that the calculated values deviated from the mean experimental values by only slightly more than the spread of the experimental data. Similarly, Lawrence and Powell

developed a good approximate method that assumed the turbine torque changes when fuel flow changes but zero change in compressor torque [11]. All parameters used in the time constant could be readily obtained from steady-state data, making it easy and convenient to establish the time constant.

The next major contribution was made by Novik, who extended the current linear single-spool turbojet engine into a two-spool engine by applying a second-order linear system [12]. The relation between spool speed and turbine inlet temperature was estimated experimentally, and the method was able to specify different spool speed responses.

The linear system is capable of representing the gas turbine transient performance. However, it could only simulate engine transient operating points with minor changes [9]. Therefore, it was impossible to obtain the compressor's operating trajectories. The development and implementation of a non-linear system were necessary to obtain more accurate results.

2.1.1.2 Non-Linear Method

The non-linear approach was initially introduced by Dugan and Fillipi [13,14]. Their research focused on investigating the acceleration performance of a twin-spool turbojet engine, by determining a set of operating points that satisfied flow compatibility.

The constant Mass Flow (CMF) method, also known as the iterative method, was developed by Larowe et al. using an analogue computer [15]. This method assumed that flow compatibility was always maintained, even during the transient condition. The simulation process was carried out in three steps. Firstly, the initial engine component outlet pressure and shaft rotational speed were obtained based on the starting condition of the transient process. Next, the engine parameters, such as the compressor pressure ratios and turbine non-dimensional flows, were estimated and iterated while keeping the rotational speed constant and ensuring that flow compatibility was satisfied after a step change in fuel flow. Finally, with the new component working condition, the compressor power and turbine power could be calculated, and the rotor angular acceleration could be

determined based on the surplus power. The new rotational speed after a one-time step could then be obtained. The complete engine computer assembly was composed of engine components assemblies. Due to the equations concluded in each component assembly, the flow compatibility was satisfied automatically. Figure 2-1 shows the process of CMF calculation, where W represents the mass flow, P represents the total pressure, T represents the total temperature, N represents the rotational speed, t represents the time step, and 1-4 subscripts mean compressor inlet, compressor outlet, combustor outlet and turbine outlet.

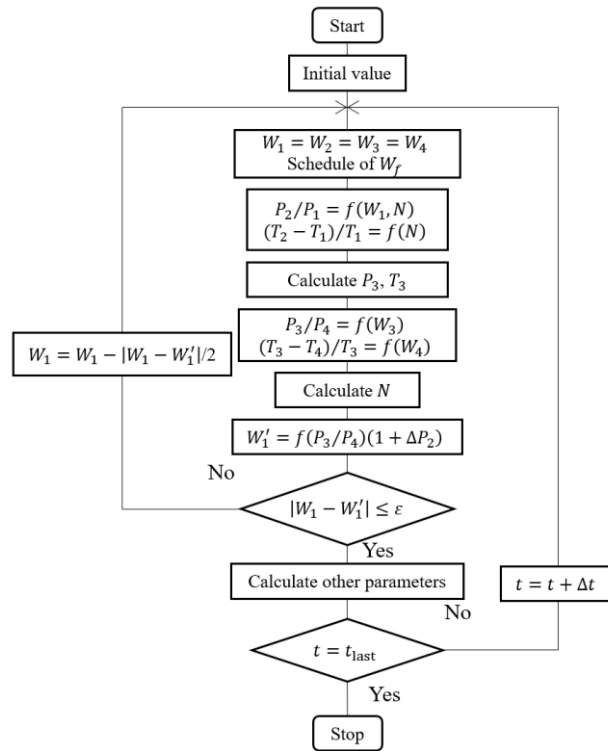


Figure 2-1 Flow chart of the CMF method

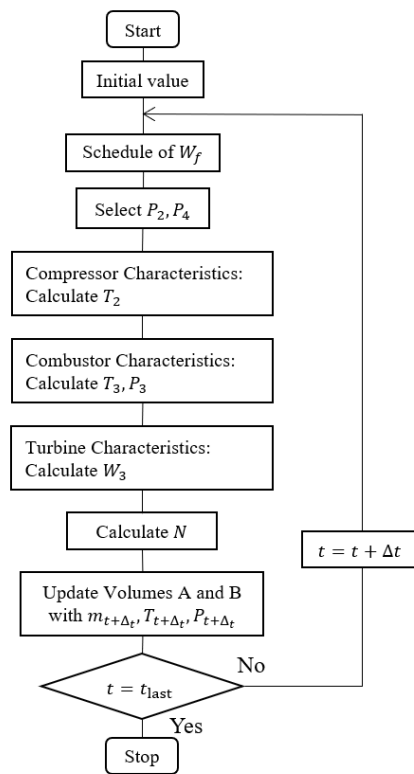
Several gas turbine studies have been investigated based on the utilization of the CMF method. Kong et al. analyzed the steady state and transient performance of a 200 kW class small turboshaft engine with a free power turbine [16]. It was found that the turbine inlet temperature exceeded the limit temperature by 0.3 seconds with the step change in fuel flow. Compared with experimental data, the maximum range of error was within 5.56%. The transient performance of a geared turbofan engine was investigated by Wanjun and Shijian by using the CMF method [17]. The outcome was compared with the result calculated from

GasTurb 10, and the maximum difference was within 1.5%. However, they didn't fully explain the benefit coming from the application of gearbox in their paper. The transient performance of hybrid propulsion systems was investigated by Wortmann et al. [18]. Two types of gas turbine engines had been examined with the design parameters from GasTurb 12. The outcome showed that the acceleration time of the hybrid gas turbine can be reduced by 50% when the motor shares 25% of the shaft power. The transient performance of the T100 micro gas turbine (mGT) in the wet mode for critical surge conditions was investigated by Carrero et al. [19]. This study simulated the engine performance under certain conditions and assessed the potential effect on power output. It was found that the wet mode will decrease the surge margin during the transient process, and the acceleration and deceleration speed should be limited. Further investigation on the effect of fuel composition change on T100 mGT transient performance was done by Raggio et al. [20]. It was found that sudden injection of hydrogen led to an excessive temperature peak at turbine entry section, while in the case of ammonia, high temperatures were not reached. An in-house program, especially for the mGT transient simulation, was developed by Kim et al. [21]. The CMF method was used during the transient simulation. Besides, a quasi-2D recuperator model was established to offer a realistic counter-flow compact heat exchanger. The transient performance of a solid oxide fuel cell and mGT hybrid system has been investigated by Mueller et al. by CMF method [22]. It was showed that SOFC/mGT hybrid systems could have a rapid transient load following capability that is faster than that of recuperated mGTs. Singh et al. [23] investigated the transient behaviour of an industrial gas turbine operating on low-calorific fuels based on the CMF method. Engine transient behaviour during a load shed and engine trip with varying fuel lower heating value (LHV) was investigated. It was found that lower LHV fuels tend to result in higher rotor over-speeds in case of a load shed and require a shorter valve closing time.

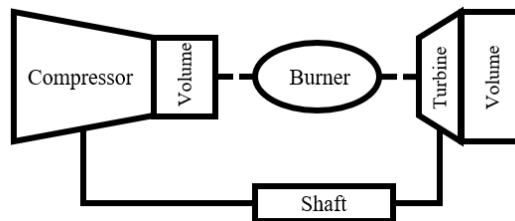
The next major advancement in gas turbine simulation was the Inter-Component Volume (ICV) method developed by Saravanamuttoo and Fawke, which considered the volume dynamics in its calculation [24]. Figure 2-2 (a) shows the process of ICV calculation. In this method, the flow was treated as discontinuous.

The gas turbine model was composed of a series of discrete components, each consisting of two parts: an actuator disk and a lumped volume, as shown in Figure 2-2 (b). The calculation process is shown in Figure 2-2 (c). The component thermal dynamic block was used for component performance calculation without considering volume dynamics, resulting in the same mass flow at positions A and B1 ($W_{B1} = W_A$). The volume dynamics block only considered the effect of volume, leading to a difference in the mass flow at positions B1 and B ($W_B \neq W_{B1}$). At each time interval, the component flow mismatch was used to calculate the change rate of component inlet/outlet parameters. The new mass flow was then used for component work calculation and rotor acceleration rate calculation.

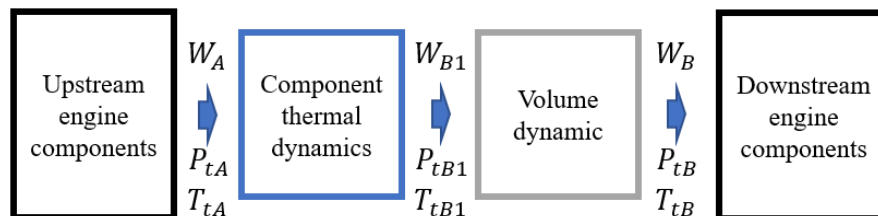
The volume packing effect had been extended from the main component volume to additional secondary air systems that include rotating pump components (such as rotating cavities between compressor or turbine disks), pressure loss components (such as pipes, labyrinth finned seals, orifices), and volume packing components (such as ducts and cavities) by Chuankai et al. [25]. Similar to the ICV method, the method modified the component outlet mass flow and the enthalpy based on the change in volume parameters. After the simulation on a turbofan engine during the acceleration and deceleration process, it is found that the disk cavities will significantly reduce the coolant flow for the turbines and might lead to potential adverse transient thermal load on the engine.



(a) ICV method flow chart



(b) Component volume description



(c) Component volume calculation logic. A represents the inlet condition, B represents the outlet condition, and B_1 represents the intermediate condition

Figure 2-2 ICV method illustration

Several studies have examined the choice of the ICV method. Chung et al. developed a dynamic model based on governing equations to investigate the compressor stall effect on aero-engine transient performance and the effectiveness of stall recovery control strategies [26]. The model consisted of various components represented by 1-D and 2-D control volumes. According to the authors, the developed model was capable of simulating in-stall transients induced by inlet temperature ramping, main burner fuel pulse, and inflow bleed. However, the authors also noted that the heat transfer effect, transonic effect, and start stall effect should be included in further analysis. The immediate effects of water ingestion on the transient performance were investigated by Haykin and Murthy by establishing a generic, high bypass ratio engine model that was based on the ICV method [27]. The mass fraction of the water varied from 0% to 8%, and effects on the air compression system and burner were considered. A non-linear relation between the change in compression subsystem performance and the amount of water is found, and this change in compression subsystem performance will have an unmanageable effect on engine transient performance, especially during engine deceleration when water mass fraction is high. The steady-state and transient performance of a twin-spool mixed-flow turbofan engine was investigated by Yadav et al. [28]. The method used was a modified version of the ICV method. Different from the other researchers who validated their data with experimental data, the validation was against the result from the Gas turbine Simulation Program (GSP). The transient performance of externally fired mGTs was investigated by Traverso et al. based on the TRANSEO platform [29,30]. The ICV method was implemented with the consideration of interconnecting volume between two consequent components. The simulation outcome was compared with experimental data and showed good accuracy. Wang et al. investigated the gas turbine fuel system based on the ICV method [31]. The simulation outcome showed that the fuel system would delay the transient performance for 0.5 seconds. Shamekhi et al investigated the transient performance of a concentrated solar power mGT during the start-up phase based on the ICV method [32]. Validation was provided for both sub-element (solar

receiver and recuperator) performance and overall transient performance, and a good agreement with errors less than 0.6% was achieved.

CMF and ICV methods have their advantages and disadvantages and will lead to a different component transient working line. Saravanamuttoo and Fawke noted a significant difference between these two methods on the compressor working line when there was a rapid change in fuel flow. In Figure 2-3, an instantaneous move from 0 to 0' then up the line 0'1 (line A) can be found when using the iterative method. However, the working line by using the ICV method would follow line B [33]. It was also mentioned that the CMF method required more calculation time per point due to the iterative process but could take a larger time step, while the ICV method required less calculation time per point, but a smaller time step was necessary. The CMF method was suitable for long-term transient simulation due to its larger time step tolerance, while the ICV method was more suitable for short-term transient simulation. Nishi and Sawada mentioned that when the complexity of the engine increases, the convergence in the CMF method would become more difficult to meet and would lead to a longer calculation time [34]. Pilidis also highlighted the strengths and weaknesses of each method [35]. He mentioned that convergence in the ICV method was occasionally difficult and depended on the choice of time step length. With an unsuitable time-step length, the convergence time of the model was longer and may even slightly oscillate [17]. Naylor pointed out that the ICV method was more suitable for fast transient simulation, while the CMF method was more suitable for slow transient simulation, where the assumption of zero mass flow accumulation within components was more justifiable [36]. Additionally, Rahman and Whidborne reported that the CMF method was slightly more accurate than the ICV method [37]. However, the calculation speed and suitability for complex gas turbine models made the ICV method more appropriate for real-time transient simulation [17].

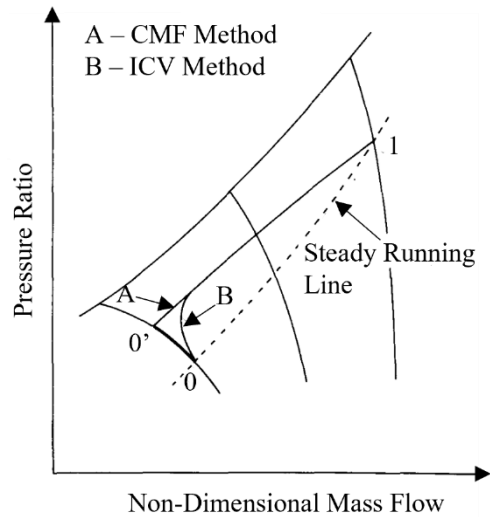


Figure 2-3 The comparison of operating trajectories for CMF and ICV method

The ICV method, while considering volume dynamics, assumed that pressure propagates immediately throughout the volume, which could lead to inaccuracies when the volume is large. To address this issue, the Pressure Wave in Volume (PWV) method was developed by Li et al in 2001 [38]. The PWV method considered four types of pressure waves (PW): PW from the turbine, PW reflection from the nozzle, PW reflection from the turbine, and PW passing through one another. A comparison between the PWV and ICV methods was illustrated based on the simulation outcome of a two-spool turbojet engine. The author stated that the PWV method reflected better gas turbine transient performance. Although the difference between the two methods was small, it could be bigger when considering large volumes such as a bypass duct. However, due to a lack of testing programs and data, the accuracy of the predicted pressure wave effect on the gas turbine transient performance is difficult to validate. The calculation time was also significantly increased, taking 8 minutes compared to 5 seconds for the ICV method.

A hybrid approach that combined ICV and CMF methods was developed by Rahman and Whidborne [37] to alleviate the weaknesses of CMF and ICV methods (slow calculation speed for the former and lower accuracy for the latter). The idea was to use the ICV method to calculate the mass flow accumulation inside the component volume and then utilize the CMF method to solve the thermodynamic algebraic equations associated with each component. A three-

spool turbofan engine was selected, and a deceleration transient process was simulated in the hybrid and CMF methods. Results showed that the hybrid approach provided results close to those of the CMF method but with significantly faster calculation speed. However, the turbine map used by the CMF method was based on the enthalpy work function, while the hybrid method was based on the pressure ratio. This difference in the turbine map resulted in offsets in compressor working lines, especially for the high pressure compressor (HPC) working line. Nevertheless, the hybrid approach had the potential to improve accuracy compared to the ICV method [39].

Apart from the PW problem mentioned above, the ICV method was prone to slow convergence because it required interaction with the components' map. This was observed under certain conditions, such as when the compressor speed line was nearly horizontal and one pressure ratio corresponded to different mass flow values. To address this issue, Li proposed the imbalanced mass flow (IMF) method in 2019 [40]. The IMF term was calculated using volume dynamics and was added to the corresponding component to rebalance mass flow during the iteration process. This method combined the advantages of the traditional ICV and CMF methods, presenting the effect of mass storage and having no limitations on map reading due to its iterative matching procedure. However, when comparing the outcome with the CMF and ICV methods, a maximum difference of 9.8% was found, higher than other researchers' outcomes. Also, the author didn't mention the time step and calculation speed of the methodology.

2.1.2 Black Box Approach

The black box approach, also known as a data-driven based model, differs from the white box approach in that it relies on operational data from system performance measurements or data generated through physics-based models. Instead of focusing on dynamic equations, thermodynamic relationships, and energy balances, the black box approach focuses on the relationship between input and output data. One of the most significant methods in the black box approach is the use of artificial neural networks (ANN). ANNs are computing systems comprised of simple, highly interconnected processing elements

(neurons) with linear or non-linear transfer functions. These elements process information by dynamically responding to external inputs [41]. The use of the ANN models in gas turbine transient simulation emerged much later than the white box approach, and a limited number of studies focused on transient analysis because most of the ANN models were mainly used for static performance analysis[42]. However, despite limited research, the black box approach is still worth discussing as it may be one of the main development directions. Chiras et al. reviewed the initial development of black box methodologies [43]. Asgari and Chen also includes a brief review of black box approach development [41]. There are different types of ANN, including (but not limited to) multi-layer perceptron (MLP), radial basis function neural networks (RBF), convolutional neural networks (CNN), modular neural networks, and recurrent neural networks (RNN). These ANN models can be split into two categories: static (feedforward) neural networks (including MLP, RBF) and dynamic (feedback) neural networks (including RNN). This section will cover static networks, dynamic networks, and other black box approaches. However, only directly linked ANN based gas turbine transient performance models will be described in detail.

2.1.2.1 Static Network (Feedforward)

Static neural networks are the simplest neural networks and are characterized by memory-free non-linear equations, which means that there is no feedback element and no delay in the network. Therefore, the output parameters of the static neural network only depend on the current values of the input parameters.

The MLP neural network (NN) is the best-known type of static network model [41]. The MLP NN is composed of an input layer, one or more hidden layers, and an output layer. Dot products are applied between inputs and weights. And sigmoidal activation functions are normally applied to the hidden layer. A simple structure of an MLP NN model is illustrated in Figure 2-4, where the blue dots represent the input layer, the orange dots symbolize the hidden layer where the estimation takes place, and the green dots depict the output layer. All connection weights and the hidden layer will be trained during the learning process to minimize the forecasting error of the NN.

This kind of model was simple and easy to train and had been used by Ruano et al. to analyze the shaft speed dynamic of the Rolls-Royce Spey engine [44]. Two topologies of MLP NNs, with one and two hidden layers, were experimented with. It was concluded that MLP NN can provide a promising outcome, and MLP NN with two hidden layers can provide a more accurate outcome. Gambarotta et al. used MLP NNs to simulate the transient performance of mGT [45]. Instead of applying the black box approach to the whole system, only the compressor and turbine were considered. It was reported that using MLP NNs could reduce the calculation time and make the simulation suitable for real-time simulation. A lack of a methodology that can generate a good set of training data for the whole operating range and the flight envelope was mentioned by Rezvani et al. [46]. To fulfil this area, the MLP NN was used with the combination of the design of experiments methodology. The input data for the NN were altitude, Mach number, ambient temperature and fuel flow profile, and the net thrust, turbine entry temperature, and compressor corrected mass flow were the outputs of the NN. The simulation outcomes had been compared with the physically based model, and the differences in outputs were controlled within 2%. Rahnama et al. also used the MLP NN model to predict the performance of the GE 9001EA heavy-duty gas turbine[47]. Air temperature, shaft rotational speed, fuel stroke reference and inlet guide vane position were the input of the MLP NN and were used to predict the electrical output power and exhaust temperature of the gas turbine. The mean squared error for output power and exhaust temperature is 0.355 and 17.708, separately. It was mentioned that the MLP NN has the potential to offer a more accurate outcome than conventional non-linear models. Khalili and Karrari compared the MLP NN based gas turbine model with the white box based model and the outcome showed that the MLP NN based model could offer better accuracy on predicted power output and exhaust gas temperature [48]. Kolehini et al. investigated the effect of hidden layer number on MLP NN prediction accuracy for gas turbine exhaust gas temperature prediction during transient [49]. It was concluded that the model with more hidden layers had a better capability.

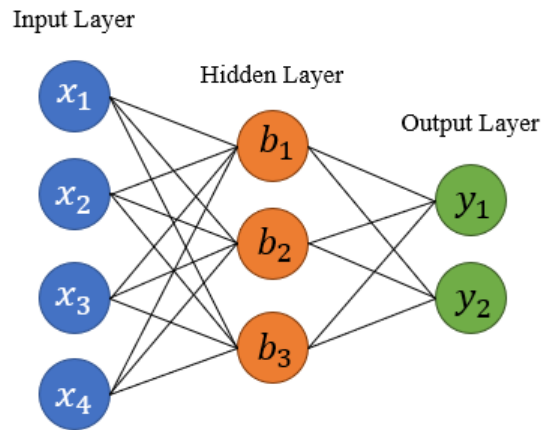


Figure 2-4 A simple structure of an MLP NN with input, hidden, and output layers

Another feedforward network is the RBF NN. Like the MLP NN, RBF NN also has the input, hidden, and output layers. A simple structure of an RBF NN model is illustrated in Figure 2-5. However, between the input and hidden layers, a non-linear transformation containing radial basis activation functions is utilized. Besides, a Gaussian function is used in the hidden layer as the transfer function in computational units. Compared with MLP NN, RBF NN needs a smaller number of hidden layers (one hidden layer is quite enough). Networks of this type can speed up and simplify data processing. Depending on the case, it is typically observed that the RBF NN requires less time to reach the end of training than MLP NN. However, RBF NN needs more training requirements due to the radial basis functions between the hidden layer and output layer.

Kim et al. utilized an RBF NN to calculate the performance of a gas turbine for aircraft engines during transient operations [50]. The input parameters used for training the RBF NN were the altitude, flight Mach number, fuel mass flow rate and additional power due to the moments of inertia of the rotating parts. The output parameters in the data-driven model were engine net thrust, shaft rotating speeds, the pressure and temperature at each station and efficiencies. Two types of RBF NNs were investigated. One was the conventional structure with no iterative process. The other was a proposed model with an iterative calculation. It is mentioned by the author that the latter RBF NN that includes an iterative calculation can offer a better similarity with the first principal model based on

NPSS. Wu also applied an RBF NN to estimate the transient performance of a split-shaft gas turbine [51]. The model was first used for predicting the characteristic curve of the compressor and then for the transient performance prediction. It was mentioned by the author that the RBF model had higher training speed and better time control performance and it was suitable for dynamic simulation which has a higher time requirement.

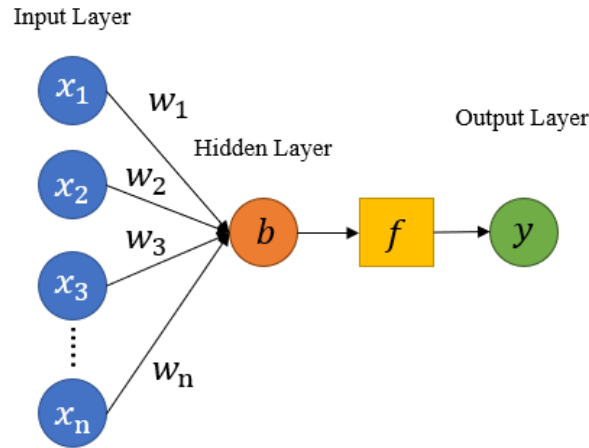


Figure 2-5 A simple structure of an RBF NN with input, hidden, radial basis function and output layers

2.1.2.2 Dynamic Network (Feedback)

A dynamic neural network is a type of neural network that can change its structure or topology during runtime based on the input data it receives. In other words, the network can adjust its architecture to best fit the input it is currently processing. This is in contrast to a static neural network, which has a fixed structure and cannot change its architecture during runtime. A comparison between the static neural network and the dynamic neural network has been investigated by Ibrahim et al. [52]. There are several types of dynamic neural networks, such as RNN, which have a "memory" that allows them to process sequences of variable length, and transformers, which use a self-attention mechanism to weigh the importance of different parts of the input sequence. Based on the comprehensive review, non-linear autoregressive exogenous (NARX) models, which is a type of RNN, are commonly used by researchers [53]. In this section, research related

to the application of the NARX model on gas turbine transient simulation will be focused on.

NARX is a non-linear autoregressive network with exogenous inputs [54]. It has a recurrent dynamic nature and is commonly used in time-series modelling. During the training process, the root mean square relative error (RMSE) is commonly used to determine the goodness of the training outcome. The structure of a typical NARX network is shown in Figure 2-6. The blue dots represent the input layer, the purple dots represent the recurrence layer, the orange rectangle symbolizes the NARX hidden layer where the estimation takes place, and the green dots depict the output layer.

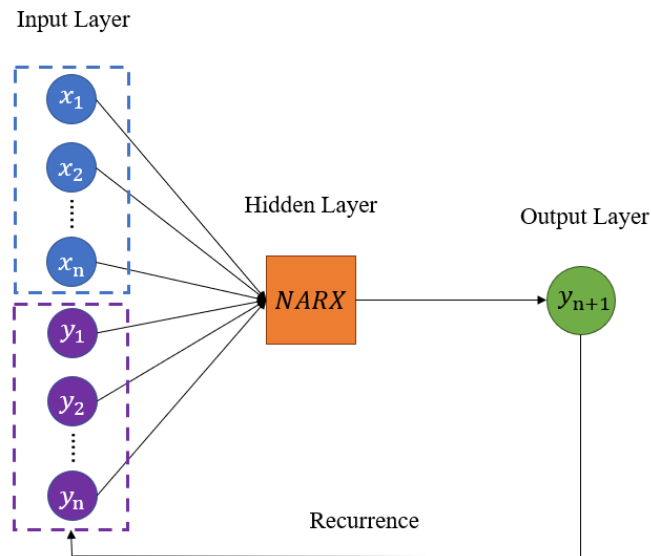


Figure 2-6 A schematic of a typical NARX network

NARX models had been used to estimate gas turbine transient performance in many studies. The transient performance of mGT in non-isolated mode was investigated by Jurado based on the NARX model [55]. Intentionally tripping one generator was simulated, and it was found that the whole system needed 5 seconds to return stability. Asgari et al. developed a NARX model to simulate the start-up simulation of a heavy-duty gas turbine [54,56]. The model consisted of three input variables and four output variables. The inputs were compressor inlet temperature, compressor inlet stagnation pressure, and fuel flow rate. The

outputs were compressor outlet temperature, turbine outlet temperature, compressor pressure ratio, and rotational speed. After training, the RMSE for the four outputs was limited to 12%. To verify the models, they were tested against three other available experimental data sets. The results showed that the NARX models can effectively capture and predict GT dynamics during the start-up process. The deviations between measured and simulated values were generally acceptable, with compressor and turbine outlet temperatures having a maximum deviation of approximately 3.5%. However, deviations could be higher for compressor pressure ratio and rotational speed, with maximum deviations of 7.4% and 7.1%, respectively. A further comparison between the open-loop NARX model and the close-loop NARX model was investigated by Asgari et al in 2021 [57]. It was concluded that the open-loop NARX model had a better performance with higher accuracy compared to the closed-loop NARX model. Yu et al. [58] employed fuel-air-ratio and ambient conditions, including flight altitude and Mach number, as input values of a NARX model to simulate the transient performance of a turbofan engine. Among the four established NARX models, the best one could have a fitting degree of 96.43% on LP shaft rotational speed and 96.95% on HP shaft rotational speed. Bahlawan et al developed a NARX model for a heavy-duty single-shaft gas turbine to investigate the cold, warm and hot start-up transient performance [59]. 14 measured time series (6 for cold start-up, 6 for warm start-up, 2 for hot start-up) were selected to train the NARX model. However, the author did not give detailed information on the input data series. The RMSE values for the estimated compressor outlet temperature, turbine outlet temperature and rotational speed were controlled within 2% when the value is within 2.5% for the compressor pressure ratio. To address the single neural model's limitations (low accuracy when the model operates outside the field in which it was trained and long training time), an ensemble of multiple-input single-output NARX models had been developed by Ibrahim et al [53]. This model used different configurations to represent each output parameter with the same input parameters. According to the author, the novel model could represent the transient performance of a three-spool aero-derivative gas turbine engine during

the full operating range with good accuracy, even with different input scenarios from different operation conditions.

Except for the NARX models, the ability of deep learning and machine learning techniques to predict gas turbine transient performance had been investigated by researchers. A machine learning-based model of a single-shaft gas turbine was developed by Asgari and Ory in 2021[60]. RNN was employed to train the datasets of the GT variables. The results demonstrated that the RNN models were capable of performance prediction of the system with high reliability and accuracy. A deep learning convolutional neural network was developed by Alsarayreh et al to predict the transient performance of a dual-fuel gas turbine [61]. The inputs for the model were the natural gas, control valve, pilot gas control valve and compressor variables, whereas the outputs for the model are output power, the exhaust temperature and the turbine speed or system frequency. However, the comparison with a NARX model showed a slight inferiority in the accuracy of the deep learning model.

2.1.3 Summary

Based on the discussion above, Table 2-1 summarizes the advantages and disadvantages of different methodologies. Since the linear model for the white box approach is no longer focused, they are not taken into comparison.

Table 2-1 Comparison between different transient simulation methodologies

	Method	Advantage	Disadvantage
White box approach	CMF [15]	• Suitable for long time step • Suitable for long-term transient simulation • Higher accuracy	• Only considering the shaft dynamic • Unsuitable for complex gas turbine model • Long calculation time for complex engine models
	ICV [24].	• Suitable for complex gas turbine model • Considering the volume dynamic effect • Faster calculation speed per point • Suitable for real-time transient	• High requirement on time step choice • Map reading problem • Lower accuracy compared with CMF Method
	Hybrid [37]	• Faster calculation speed than CMF method • Comparable accuracy compared with CMF method • Easy to be implemented	• Offset in compressor working line
	PWV [38]	• Consideration of PW effect	• Significantly increased calculation time
	IMF [40].	• No map reading problem	• Relative lower accurate • Unknown time step choice and calculation speed
	MLP [41]	• Simple and easy to train	• Long training time compared with RBF

Black box approach	RBF [50]	• Relatively higher accuracy compared with MLP • Low requirement on the number of hidden layers • Short training time	• Higher training requirement
	NARX [52]	• High capability • More suitable for dynamic condition • Higher generalization capability	• Longer training time and more complex structure compared with static network

2.2 Heat Transfer Effect on Transient Performance

The large temperature difference between the engine metal and the main gas path during the transient process produces a large heat flow, significantly affecting engine performance during and after transient control [36]. As research on gas turbine transient performance prediction continues, discrepancies between simulation and experimental results have led to the identification of heat transfer effects that affect the transient operation of engines. Heat transfer effects refer to the effects coming from the heat exchange during the gas turbine transient process, including the heat soakage effect, tip clearance effect, and component characteristic change effect. Heat soakage is the process of heat transfer between the working fluid and the engine component, while the tip clearance effect results from changes in clearance due to temperature, centrifugal force, and pressure difference. Component characteristics are described by the component map, which can be affected by changes in working flow physics. An illustration of the heat transfer effects on the performance of the components is shown in Figure 2-7. A brief introduction to the heat transfer effect was provided by Hashmi et al. [62]. MacCallum used the term 'thermal soak' to summarize the effects, which refer to the performance difference from the equilibrium performance after rapid acceleration and deceleration [63]. These effects significantly influence the engine acceleration or deceleration time and the rate of change of aerothermodynamic parameters [64].

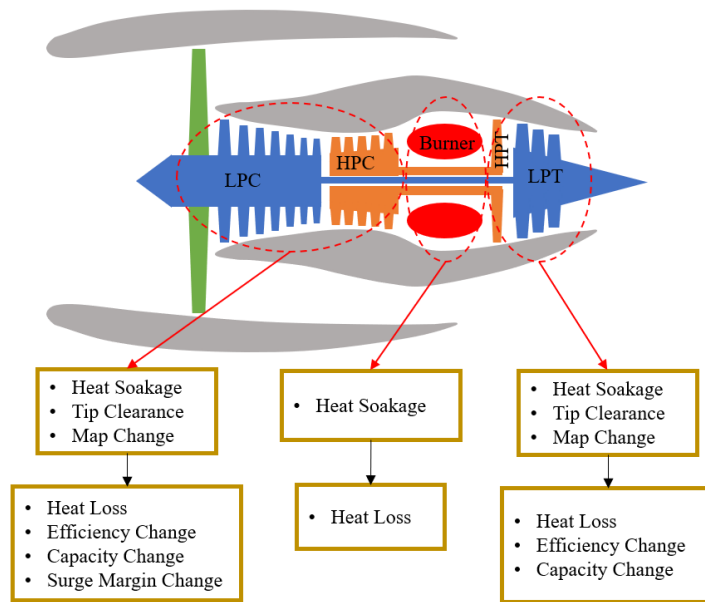


Figure 2-7 Heat transfer effects on the performance of the components

In this section, the heat soakage, tip clearance, and component characteristic change effects will be discussed separately. The methodology development and application will also be included.

2.2.1 Heat Soakage Effect

When investigating gas turbine steady-state performance, it is reasonable to assume that the compression and expansion process occurs adiabatically because the heat flows are exceedingly small compared to engine component powers [36]. However, during the transient manoeuvre, the heat transfer between the primary gas path and engine components is too big to ignore and will affect component performance. The large temperature leads to a large heat flux, which can lead to non-adiabatic expansion and compression both during and after a transient manoeuvre [63]. Also, some engines can consume a typical additional 30% energy excess to reach a new higher steady state operating temperature due to the thermal soakage [65]. According to the research findings, it was discovered that heat soakage causes a delay in the transient response of gas turbines. Specifically, during the acceleration phase, there is a reduction in the rate of increase of engine thrust and shaft rotational speed (PCN). Conversely, during deceleration, the rate of decrease is reduced. In this section, the

methodology development of heat soakage analysis and the effect of heat soakage on gas turbine performance will be discussed.

The significance of the heat soakage effect was first mentioned in 1968 by Bauerfeind [66], who simulated a two-spool turbofan engine to introduce the heat soakage effect. Additionally, MacCallum identified the transient factors that may impact the performance of a simple jet engine after rapid acceleration [63]. To analyse the effect of heat absorption, he utilized the F parameter as a correction factor for the polytropic efficiency. The F parameter was determined as the ratio of the heat transferred from the air to the work transferred from the air.

The F parameter method can predict the heat absorption effect, but it is more of an empirical formula for polytropic efficiency modification. To improve the simulation, the lumped parameter method (LPM) for heat soakage calculation was introduced to gas turbine transient simulation by Fawke and Saravanamuttoo [67]. The mean temperature based on the component inlet and outlet temperature is used to determine the values of the Prandtl number, thermal conductivity, and viscosity for the gas to calculate the heat transfer coefficient (HTC). However, for each component, only one HTC value is applied, making the investigation of the heat soakage affect an approximation. Based on the simulation outcomes for acceleration and deceleration, it was found that the operating path of compressor characteristics had not been significantly affected. Khalid and Hearne provided a full description of the LPM in their transient performance heat soakage model, which was used to investigate the heat absorption effect on a two-spool turbofan engine. [68]. The basic idea of LPM is that the interior temperatures of bodies remain essentially uniform at all times during a heat transfer process. During the process, the heat absorption of compressors and turbines was taken into consideration, except for the combustor. They also considered the effect of turbine cooling flow variations, control temperature sensor response and tip clearance variations, which will be discussed in subsequent sections.

LPM offers a convenient way to simulate the heat transfer between the solid and fluid. However, it assumes that the fluid temperature keeps constant during the heat transfer process, leading to overestimating the heat flux. Naylor moved the

LPM a step further by considering the fluid temperature change during the heat transfer process [69]. The stream method improved the original LPM. But it needed a specific gas turbine component structure to build the model. So, a thermal network model was developed by Visser and Dountchev to offer a more generic heat soakage estimation [70]. In the thermal network, several heat sinks represented the components. Each heat sink was calculated by the LPM method and then connected using Non-linear Differential Equations. With the link among each heat sink, the original 0-D thermal recuperator model can be seen as a quasi-1D model (in radius direction). However, this method had the same limitation as the original LPM. It could not represent the heat transfer process inside parts, and there would still be errors for complex heat transfer patterns. The stream method and the thermal network improved the LPM. However, the heat calculation between the surface to surface is still discontinuous. A real 1D method is still needed. Zhuojun fulfilled this gap by extending the original 0D method in the axial direction [40]. In the 1D method, each solid component was no longer treated as one point located at the centre of mass. By using the integral method, the temperature distribution along the component could be simulated with much more accuracy. However, temperature profile, area, and mass distribution along the length of turbomachinery were considered linear, which was inconsistent with the fact.

The finite element method (FEM) was introduced by Chapman et al. to estimate the heat transfer rate between the high pressure turbine (HPT) shroud, as well as the disc, and the working fluid [71,72]. Compared to the LPM, the finite element method provides a more detailed representation of the internal temperature distribution and can be applied more easily to components that consist of two materials. The central forward finite difference method was utilized when solving partial differential equations.

Both the LPM and FEM have their advantage. Progress had been made by creating a quasi-one-dimensional heat transfer model that combined both LPM and FEM by Chen et al. [73]. In this new model, the internal nodes in FEM had been replaced with lumped engine components, such as a lumped compressor

stage or a lumped combustor. This allowed for the consideration of all engine components with proper settings. Additionally, the new internal nodes could account for heat conduction and radiation, whereas the original LPM only considers heat convection.

The methodology mentioned above can be concluded as the replacement structure model (RSM) [74] because the idea is to use mathematical equations to represent the components in the gas turbine. However, there are two other models: the impulse response model (IRM) and the state-space model (SSM). The idea of IRM is to calculate the impulse response $g(t)$ when subjected to a unit impulse input. For the SSM, the idea is to calculate the temperature change using the state equation. Riegler proposed an IRM-based method whereby primary gas path heat soakage is calculated non-dimensionally using Biot and Fourier numbers[75].

The method describing heat transfer phenomena depends linearly on temperature. Therefore, steady-state and transient phenomena might be modelled separately, and the effects might be superimposed. However, Naylor pointed out that the workings of the method presented were still based on Nusselt numbers for flow through tubes, mean component inlet and outlet temperatures, and mean component temperatures [36]. Merkler et al. developed an SSM-based model for disc temperature calculation [74]. Both IRM and SSM have their advantage. However, after a comprehensive review, it shows that the RSM is the best popular model since it directly indicates the physical process during the heat transfer process.

After discussing the heat soakage calculation, it is necessary to discuss the heat soakage effect on component performance. A common way was to assume that the compressor and turbine were still working at the adiabatic condition [76]. Part of the heat transfer occurred before the adiabatic component process, and the rest occurred after the adiabatic component process. This would only affect the component inlet and outlet temperature instead of modifying the component power [77]. However, some researchers believe that the heat transfer process

will affect the compressor and turbine power [70,78]. And a modification of the component power should be included.

The above discussion illustrates the development in heat soakage methodology development. Significant progress had been made to improve the accuracy during the heat transfer calculation. However, since the LPM offers a convenient way to simulate the heat transfer between the solid and fluid, it has been used in extensive research. Zhili and Kuo developed an unstable heat exchange model being used in gas turbine transients [79]. The difference between their work and Khalid and Hearne's work was the choice of HTC. In Khalid and Hearne's investigation, only turbulent flow was considered. But both the laminar flow and turbulence flow were considered in Zhili and Kuo's work. Unfortunately, they didn't compare their outcome with the experiment data. In 2001, Stamatis and Mathioudakis also included the heat transfer effect in their real-time engine model [80]. LPM was being used, but the choice of HTC was different. However, the LPM was only applied to the compressor and turbine blades and casings for simplification. Hu and Xiaochun applied Khalid and Hearne's on a different gas turbine engine [81]. With the same simulation environment, they compared their outcome with Maccallum's work, and it showed that the LPM had a better prediction than the F parameter method. Tagade developed a gas turbine simulation model for the startup to shutdown process of a twin-spool turbofan engine [82]. The model takes into account thermal transients, shaft transients and gas dynamic transients. In the thermal transient simulation, LPM is used. Alves and Barbosa built a regenerative-cycle gas turbine model that considered friction and non-adiabatic volumes [83]. The importance of the heat soakage effect in the micro-turbine engine was mentioned by Davison and Birk [84,85]. The engine model was validated against the experimental data. Although there were overestimations in the transient performance, the correlations were good. The heat soakage model was closer to the experimental data compared with the CMF model. Qiuye used the lumped parameter method to analyze a two-spool engine transient performance [77]. Their work was similar to Zhili's, but they moved one step further to compare the simulation results with experimental data. Besides, they correct the Nusselt number of compressor blades by taking both laminar flow

and turbulence flow into consideration. Vieweg introduced the transient heat soakage code within the DLR's in-house gas turbine simulation framework GTlab and compared the outcome with the engine data[76]. The code uses the lumped parameter method but improves the method by using a scaling process to avoid manually updating the HTC. With that, only the design point needs to be estimated by the user. Then, the heat soakage code was extended to allow for heat transfer among different components, bleed flows, and the environment [86]. Ferrand investigated the turboshaft engine power losses caused by heat soakage in the combustion chamber [87]. However, with the method developed, there is only a maximum correction of 0.55% of the design speed, which shows that it is not sufficient to correct the heat soakage effect only considering the combustion chamber. Kim et al. developed a simulation tool for the dynamic behaviour during the start-up of heavy-duty gas turbines [88]. They adopted a multi-segment model instead of using a single control volume model where the compressor was calculated group by group and the turbine was calculated stage by stage. Besides, they used iterative calculations at each control volume to have a more accurate outcome.

Based on the discussion above, a comparison between features considered by different heat soakage models is listed in Table 2-2. It can be found that current state-of-art heat soakage model does not consider the combustor temperature distribution and cooling technologies.

Table 2-2 Comparison between different heat soakage model

Author	Heat Soakage Model	Combustor Model	Cooling Model
Naylor (2004) [69]	2D Model	No	No
Visser (2015) [70]	0D Model	Mean Temperature Model	No
Qiuye (2017) [77]	0D Model	Mean Temperature Model	No

Vieweg (2017) [76]	0D Model	Mean Temperature Model	No
Kim (2018) [88]	1D Model	Not Mentioned	No
Fengping (2019) [73]	Quasi-1D Model	Mean Temperature Model	No
Zhuojun (2020) [40]	1D Model	Mean Temperature Model	No

2.2.2 Tip Clearance Effect

During transient manoeuvres, gas turbines are expected to experience varying thermal and mechanical loading that can lead to small changes in the axial and radial dimensions of the components. These changes are caused by three main reasons: (1) The thermal growth due to the change in component temperature; (2) The centrifugal stresses on the rotating components; (3) The pressure stresses on the static components. Due to the change in radial dimensions of the components, the tip clearance of compressor and turbine will change during the gas turbine transient manoeuvre. Change in tip clearance will affect the secondary loss in flow passage and lead to change in component efficiency, which further effect the gas turbine transient performance. In this section, progress in tip clearance prediction and component performance modification will be discussed.

When estimating thermal growth, it is strongly linked with the heat transfer calculation that was discussed in Section 2.2.1. Based on the discussed LPM, each component was treated as a node with the mass at the centre of it, and the component temperature was known at each time interval[36]. It is easy to apply the thermal expansion equation to the lumped component. With the setting of the reference point, the deflection due to the temperature change can be estimated [89]. Progress had been focused on the development of a more accurate mean temperature prediction method. Kypuros et al. established a 1-D analytical

method to have a better mean temperature prediction on the temperature distribution along the rotor width direction and the casing radius direction [90,91]. FEM was used by Pilidis and MacCallum to develop a model for blade tip movement as well as seal clearance change [92]. The model represented a deep disc rotor with three elements: a thick hub, a thin diaphragm and an outer section or rim. Naylor applied the same idea to his ring method when calculating the disc radius change [36]. In the ring method, the disc was divided into small rings (typically 5 to 6 rings), and the overall radius change was the sum of the displacement of all the small rings. MacCallum developed a 2D disc temperature distribution model [93]. The geometry of the disc has been divided in the radial into 20 increments and then in the axial direction into 10 elements. The effect on the choice of the number of elements had also been studied, and it was found that the change in the average temperature was under 1K when increasing the axial elements to 14. Chapman et al. established a 1-D FEM model to predict the thermal displacement of the turbine shroud (along the radius direction) and rotor (along the thickness direction) [71,94]. Sheng et al. improved the thermal expansion model by considering the temperature-dependent heat expansion coefficient and made the simulation time suitable for real-time transient simulation [95].

Instead of using the mean temperature for thermal displacement estimation, it can be estimated directly based on the temperature field. Fang et al. derived the basic equation of the temperature field of uniform thickness and single rotor disk and then it was used to estimate the thermal expansion by integration [96]. Compared with the original thermal expansion method, this method considered the temperature difference along the disc radius. However, the calculation process was much more complex since it needed to solve the Bessel equation.

Another way is to express thermal displacement as an exponential function of time. It was known that the component temperature could be expressed as a function of time. Since the thermal growth of the component was directly proportional to the component temperature, it was able to express the thermal growth as an exponential function of time [68]. With the estimated thermal

expansion, the total tip clearance change can be estimated with the consideration of centrifugal stress and pressure stress.

The displacement caused by centrifugal force is also important during the tip clearance estimation. Two components are considered. One is the rotor blade, and the other is the disc. For the blade, a common simplification was that the blade was assumed as a constant cross-area flat plate [35]. Fang et al. improved the estimation method by considering a changing cross-section area blade [97]. Instead of the changing cross-section area, the effect coming from the shrouded turbine blade was taken into consideration by Naylor and Pilidis [35,36]. For the rotating disc, the centrifugal deformation will be affected by the disc's mass and the centrifugal force from the blade mass. When considering the disc displacement due to its mass, it is commonly assumed that the disc has a unique thickness [96]. When applying this method during the disc displacement due to centrifugal force, it was suggested that the thickness of the disc should be equal to the minimum diaphragm thickness of the actual disc profile [35]. The displacement due to blade mass was not considered by Kypuros et al. and Chapman et al. and it was mentioned that this force would lead to a 15.9% difference when estimating the tip clearance movement [95].

The pressure forces will lead to a change in the casing radius [90]. However, pressure deformation is relatively small compared to the effect from thermal deformation and centrifugal deformation. In some cases, it was assumed negligible and was not introduced to the calculation [78].

Research showed that the change in tip clearance will affect component efficiency, mass flow, and surge margin [98]. Several researchers had investigated the relationships between tip clearance and component efficiency. The modification methodology on the tip clearance effect can be concluded in two ways. The first is the empirical correlation modification, and the second is the direct modification. The idea of the empirical correlation modification method is to use the empirical correlation and tip clearance value to modify the component performance. In 1970, Lakshminarayana derived an expression for the decrease in compressor efficiency due to the tip clearance change [99]. The turbine

efficiency modification methodology was raised by Baskharone [100]. This method had been used by several researchers. Hongmei and Zhili utilized Lakshminarayana's equation during the tip clearance change in their transient study [101]. Jiamei also investigated the tip clearance movement and its effect on engine transient performance by using Lakshminarayana's correlation [102]. But only the last stage of the compressor was considered during the performance modification. An irregular finite volume mesh method for the disc that has universality was developed by Ma [103]. However, this method was finally applied to a transient simulation as a post-modification, and it might lead to an error since the transient performance will be affected by tip clearance change during the process and might lead to a different working line. The empirical correlation can offer a relative accuracy modification. However, it is hard to use when a certain blade profile is missing. Direct modification method is then recommended under this condition. MacCallum found that the change in compressor tip clearance caused a reduction in efficiency of 0.4%, and the change in seal clearance caused an increase in the cooling flow of 0.2% [63]. Walsh and Fletcher suggested that a 1% increase in tip clearance will lead to a 1-2% reduction in efficiency and a 2-15% reduction in surge margin [65]. Hourmouziadis and Albrecht suggested that the exchange rate is in the range of 2% (a 1% increase will result in a 2% decrease in efficiency) [104]. Shirzadi and Saeidi suggested that the exchange rate is in the range of 0.8% based on turbine polytropic efficiency [105]. This method is also applied in GasTurb, where the user can directly set the ratio between the change in clearance and the change in component performance [78].

2.2.3 Component Characteristics Change Effect

Changes in clearance and heat absorption can also affect the characteristics of engine components. The exchange of heat between the main gas path and the components changes the temperature of the gas path, leading to modifications in the development of the boundary layer and fluid density [106]. Furthermore, changes in tip clearance can impact component characteristics, including compressor efficiency and surge margin.

The ‘bulk’ effect on compressor characteristics caused by density change was first investigated by MacCallum [107]. It was found that a reduction in air density at the compressor outlet led to a decrease in the compressor surge margin. Later, Grant reported that heat transfer from the wall to the boundary layer affects the development of the boundary layer, accelerating separation [108]. MacCallum and Grant developed a method to predict the effects of boundary layer changes due to heat transfer on the characteristics of an axial-flow compressor [109]. The method included a predicted mean line method for compressor characteristic map generation and an empirical correlation for modification. They found that the thickening of the heated compressor blade boundary layer was more significant on the suction surface than on the pressure surface, causing a blockage effect (shown in Figure 2-8). They also proposed an empirical correlation for the change in flow outlet angle with respect to temperature change. Notably, the angle of departure had doubled compared to the adiabatic case due to the rapid growth of the suction surface boundary layer. The method provided a predicted output, but no experimental data was presented to confirm the theoretical predictions [110].

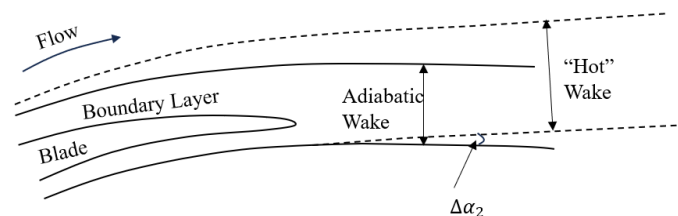


Figure 2-8 Change in boundary layer development due to heat transfer [109]

Further work was done by MacCallum and Pilidis by developing compressor map modification empirical correlations for speed line and surge line modification [106] [111]. This program was then used to analyze the compressor surge margin movement of a two-spool bypass engine during transient [112]. This similar method was then be used by Stamatis which allowing the evaluation of the effects related to heat transfer to the turbine blades on its performance characteristics [113]. In his research, he considered the effects of thermal dilatation and boundary layer development and then corrected the turbine map based on them.

Besides, the blade temperature took the consideration of nonuniform temperature distribution which will offer a better outcome for the turbine blade. The movement of the compressor speed line is shown in Figure 2-9.

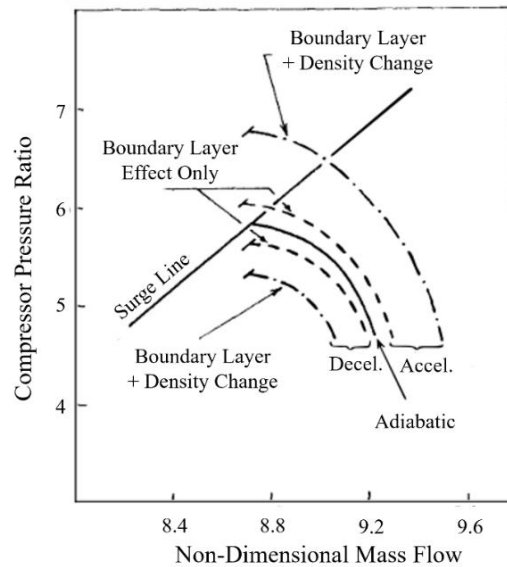


Figure 2-9 Compressor speed line movement due to boundary layer effect and density change [106]

Larjola investigated the effects of heat transfer on both compressor surge margin and turbine mass flow blockage[114]. His work found that changes in the boundary layer and air density were responsible for 66.2% of total surge margin reduction, while clearance change accounted for only 0.02%. Larjola's prediction method had better accuracy than MacCallum and Grant's, as it considered surge line movement. In the same year, Crawford and Burwell attempted to experimentally confirm MacCallum and Grant's findings [110]. Using experimental data from several "Bodie" transient events, they calculated the total temperature and steady-state adiabatic efficiency of the unsteady compressor during the process from take-off power to rapid deceleration and immediate re-acceleration after idling. Although they found no statistical correlation between stall margin loss and heat flow, they observed that average heat flow was greater when the compressor stalled.

Torella investigated the secondary transient effect (heat transfer, packing lag in the ducts, transient effect in the combustion) [115]. Since the change in

components efficiency was determined by exit temperature while the exit temperature depended on the efficiency, an iterative procedure was necessary.

In 2007, Morini analysed the heat soakage effect on the compressor by using a simple method based on an equivalent variation of the compressor polytropic efficiency [116]. They concluded that the heat soakage has a high impact at low shaft speed and negligible impact close to nominal shaft speed. In the same year, Shah analysed the heat transfer effect on compressor performance [117]. He raised the idea of non-dimensional heat flux which was a heat transfer rate that had been normalized by the blade inlet stagnation enthalpy inflow rate. Then he concluded the correlation of total pressure change coefficient and change in flow angle deviation with the nondimensional heat flux.

In 2015, Kiss and Spakovszky used Shah's correlation to analyse the effect on compressor stall margin during a re-acceleration phase of a Bodie manoeuvre [118,119]. To illustrate the non-adiabatic transient, they develop the "composite compressor map" that includes the time-dependent speed line, stall line and working point. Figure 2-10 illustrates a comparison between the original compressor map with the composite compressor map, where the map in black represents the former and the map in colour represents the latter. τ in Figure 2-10 represents the non-dimensional time, which equals t / t_{acc} . Besides, the model they created also included stage re-matching and deviation effects, which are not included in current software like NPSS. Compared with the NPSS, the new model captured an 8.0-point stall margin loss when NPSS is 1.2-point. However, this method calculates the heat transfer rate q^* based on the component temperature (based on steady-state operating point) and the flow field temperature (based on adiabatic Bodie transient) in advance. So, the heat transfer rate is not calculated during the transient operation. This led to some errors in the calculations, but the magnitude has not been analyzed.

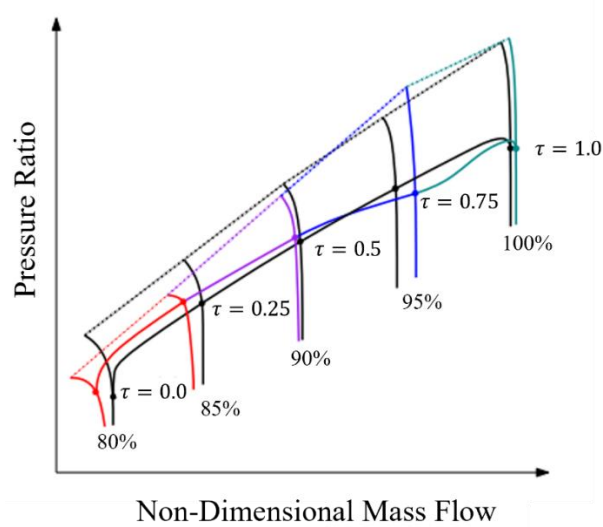


Figure 2-10 Example schematic diagram of a composite compressor map used to represent the unsteady properties of velocity and stall lines during transient with heat transfer [118]

Zhuojun also analysed the component characteristic change in his work [120]. The map update method was similar to the work of Pilidis and MacCallum. However, the new method enabled the characteristic map to be updated at each time interval with the updated scaling factor of mass flow, pressure ratio and efficiency.

A brief description of the different methodologies described above and a comparison among them is shown in Table 2-3.

Table 2-3 Comparison between different component map modification model

Author	Basic Idea	Advantage	Disadvantage
Pilidis and MacCallum (1986) [106] [111].	Obtaining new shaft rotational speed and surge margin based on the empirical correlation	• Easy to be implemented	• Only suitable for certain compressor • High cost of modelling • Limited agreement with experiment
Torella (1990) [115].	Obtaining a new component efficiency based on the empirical correlation.	• Easy to be implemented	• Without considering the flow dynamic change • Low Accuracy

Kiss and Spakovszky (2015) [118,119].	Using composite compressor map that includes heat transfer effect for a certain transient process.	• High accuracy • Direct use of map instead of modifying during transient	• Only suitable for certain transient process • Heat flow estimation based on adiabatic condition
Li (2019) [120].	Using scaling factor to modify the map at each time interval.	• Considering the time-dependent heat transfer effect on the map	• Low accuracy due to direct use of index value from Pilidis and Maccallum's work when the type of gas turbine was different

2.3 Transient Simulation Platform

The development and application of gas turbine transient simulation, as well as the heat transfer effect, are crucial, but the complexity of the system makes it challenging for researchers to use these methodologies easily. Moreover, launching each simulation from scratch every time is unrealistic. Therefore, easy-to-use software that can simulate gas turbine transient processes is necessary. The development of transient simulation platforms involved three domain techniques: analogue, digital, and hybrid computing. MacIsaac and Saravanamuttoo offered a parallel comparison of these three technologies [121], concluding that all three simulations were complementary to each other and had their place. Analogue simulations can provide actual time-based results of complex engines but lack accuracy due to presenting non-linear component characteristics. Digital simulations are most suitable for detailed studies, but obtaining real-time results was hard due to slow computer speeds at the time. Hybrid simulation combined the ease of integration of analogue simulation with the logic and stored program capability of digital simulation, but the system was too complex. The development of gas turbine transient simulation platforms will be discussed in this section.

NASA has made significant contributions to the gas turbine performance simulation field by developing several codes, including HYDES, DYNGEN, DIGTEM, DEAN, and others [122–125]. HYDES was a hybrid computer program

that investigated the dynamics of one-spool and two-spool turbojet and two-spool turbofan engines using the ICV method. DYNGEN, based on the GENENG method, was the first digital computer program for steady state and transient performance simulation, achieving better accuracy using the modified Euler method [126]. DIGTEM, an advanced version of DYNGEN, was the digital version of HYDES. DEAN improved DYNGEN by implementing pre- and post-processing features [127]. In addition to these codes, NASA has developed Onyx, a Java-based simulation framework that enables direct access to CAD geometry for mesh generation and visualization software tools. [128,129]. Onyx was developed as part of NASA's Numerical Propulsion System Simulation (NPSS) project, which integrates various disciplines to enable numerical zooming between 0-dimensional to 1-, 2-, and 3-dimensional component codes [130,131]. A illustration of gas turbine model established by NPSS is shown in Figure 2-11. In NPSS transient simulation, the Predictor-Corrector method was used. The basic idea of the Predictor-Corrector method is to form a loop. The predictor equation was used to estimate the initial predictor variable, and then the estimated variables were used to calculate the dependent derivative term, which will be used in a corrector equation to form a corrector estimate. The equation converged if the difference between the predictor variable and corrector estimate was small enough. Moreover, the volume dynamic was added to NPSS which filled the gap[132].

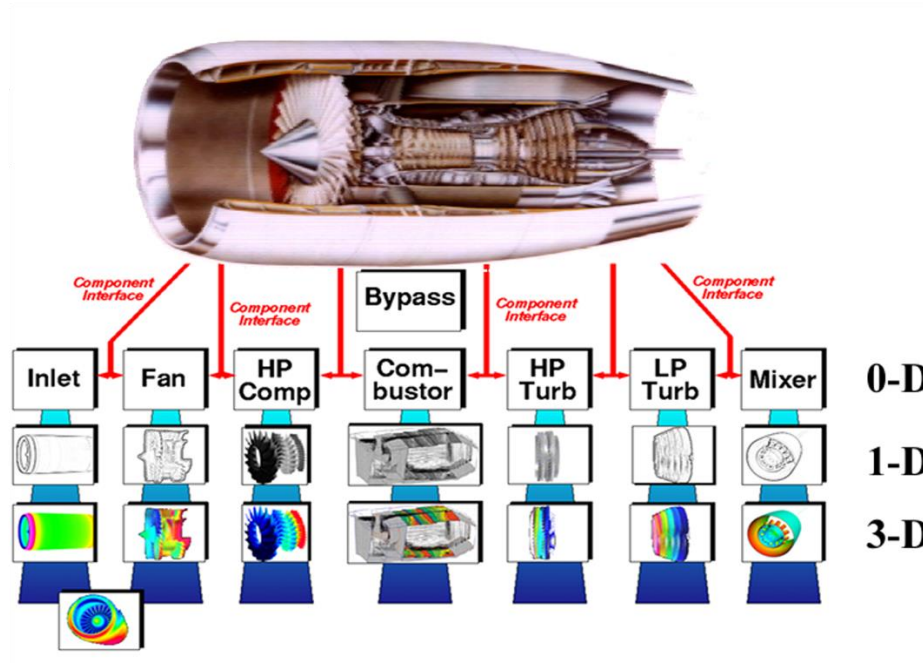


Figure 2-11 Gas turbine model based on NPSS software [133]

In addition to NASA, many other efforts are also being made to develop simulation systems.

Hirst created a program called SYSTRAN, which might be the first analogue computer program [134]. The program improved the unstable regimes in the lumped thermodynamic model created by Saravanamutto. The basic idea was to find equilibrium conditions independently of dynamic stability by programming the model differential equations. Merson's 4th-order Runge-Kutta integration procedure was used to solve the equations.

Cranfield University has also contributed to the development of transient simulation platforms. In 1982, Palmer and Chengzhong developed TURBOTRANS, which enabled transient calculations [135]. The program used Euler's method to replace differential equations with finite difference equations, similar to the method used in DYNGEN. TURBOTRANS introduced "Thermodynamic Bricks" to make the program suitable for different types of gas turbines. Today, TURBOTRANS has been developed into TURBOMATCH, which is suitable for all types of gas turbine transient simulations. TURBOMATCH allows users to choose between ICV and CMF methods for simulation. In

TURBOMATCH 3.0, a user-friendly interface has been developed, making it convenient for users to build their gas turbine model. The window of TURBOMATCH 3.0 is shown in Figure 2-12.

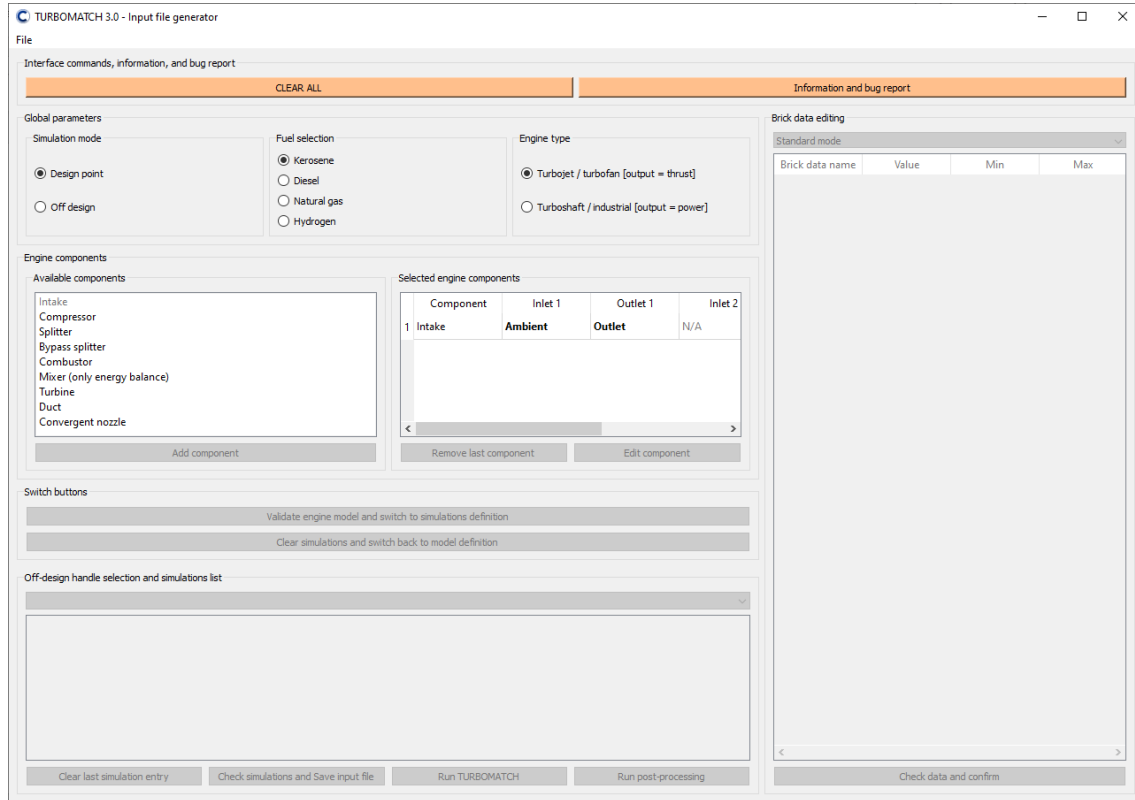


Figure 2-12 TURBOMATCH interface

TRANSEO code was developed by the Thermochemical Power Group at the University of Genoa based on the MATLAB-Simulink environment[136]. TRANSEO code has the capacity to simulate the transient performance of aircraft engines and heavy-duty gas turbines, and in 2000 it had been extended to the microturbine transient cycles. The modularity and flexibility of TRANSEO code enable the simulation of various engine models and the transition of a stationary microturbine to a complete hybrid fuel cell system. The volume dynamic effect is included in the TRANSEO code with the consideration of intercomponent volume.

Chappell and Blevins developed the Advanced Turbine Engine Simulation Technique (ATEST) [137], which is based on a modular approach and is suitable for building arbitrary cycles. ATEST uses the CMF method (an iterative method) during the transient simulation and provides capabilities such as multiple

interstage bleed and modelling of second effects during transient, such as tip clearance change and low Reynolds number effects. ATEST 3.0 improved on previous versions with better modelling techniques, component map representations, and other capabilities [138].

Kurzke developed the GasTurb simulator, a commercial gas turbine simulator that has been updated to version 14 [139]. It includes several engine models, such as jet engines, gas turbines, propulsion systems, and power generation, as shown in Figure 2-13 (GasTurb 12 as an example). In version 14, electric system simulation and afterburner transient were added. Transient simulation offers a choice between a simplified model and an enhanced model. The simplified model uses the CMF method, while the enhanced model includes the heat soakage and tip clearance effects. However, components with heat exchange are simplified to a zero-dimensional model with infinite thermal conductivity and uniform temperature distribution. Transient simulation is only possible after the off-design performance simulation, and the engine model in the library cannot be easily modified.

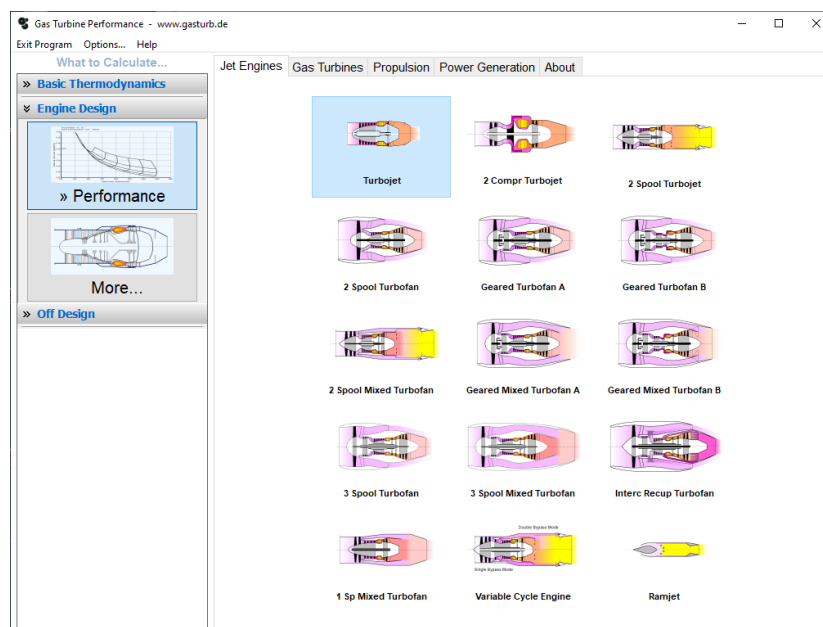


Figure 2-13 Types of Engines in GasTurb (GasTurb 12) [140]

The GSP, developed by Visser & Broomhead [141], is a user-friendly program designed to improve upon the weaknesses of DYNGEN. It enhances stability, the

speed of numerical iteration processes, and the user interface. The GSP window with an example of a turbofan model is shown in Figure 2-14. In GSP transient simulation, conservation equations are solved by a modified Euler Equation. The detailed transient simulation methodology was not illustrated, but the author mentioned that the simulation was calculated iteratively. So, the CMF method might be applied. Different from GasTurb, GSP consists of several component modules or subroutines that allow the user to invoke a single component and make some changes as needed. This provides greater flexibility and a more advanced program architecture. Moreover, heat soakage was taken into consideration in GSP 11.

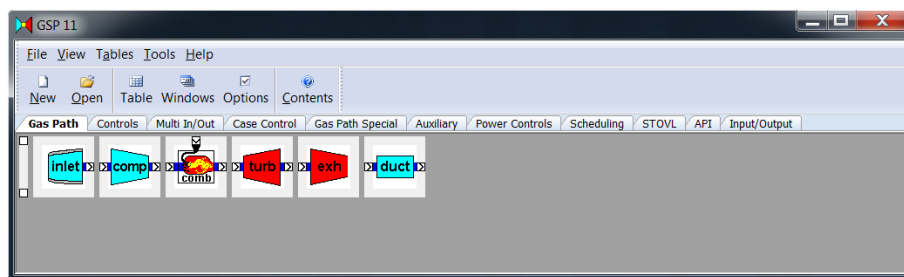


Figure 2-14 GSP window with a turbojet engine model [142]

Alexiou and Mathioudakis described the use of a generic object-oriented simulation tool called EcosimPro [143]. This tool was then used to develop the Propulsion Object Oriented Simulation Software (PROOSIS) [144]. The simulation window of PROOSIS is shown in Figure 2-15. Due to the fully developed friendly interface, this program was suitable for four levels of users ranging from core library developers to less experienced users who only run simulations using pre-generated customer decks.

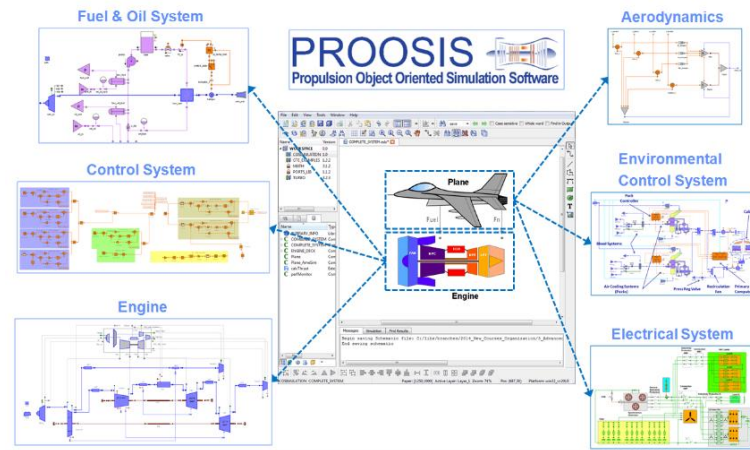


Figure 2-15 Multidisciplinary aircraft system simulation with PROOSIS [145]

Siemens also developed its own gas turbine performance software Simcenter Amesim [146]. However, not much detailed information related to its transient methodology can be found. Based on some researchers' work, the ICV method can be used in Amesim and the component thermal and materials library can be utilized for heat soakage analysis when relevant data is available[147].

Based on the discussion above, A brief description of the different methodologies described above and a comparison among them is shown in Table 2-4.

Table 2-4 Comparison between different gas turbine simulation platform

Software Name	Shaft Dynamic	Volume Dynamic	Tip clearance	Heat soakage	Component Map Modification
GasTurb	YES	NO	YES	YES	NO
GSP	YES	YES	YES	YES	NO
PROOSIS	YES	YES	YES	YES	NO
NPSS	YES	YES	YES	YES	NO
Amesim	YES	YES	NO	YES	NO
ATEST	YES	N/A	YES	N/A	NO

TURBOMATCH 3.0	YES	YES	NO	NO	NO
-------------------	-----	-----	----	----	----

2.4 Research Gap Identification

Based on the comprehensive literature review discussed above, the research gap for the heat transfer effect on gas turbine transient performance is identified. Two main gaps are concluded as described below:

1. Current heat transfer effect analysis lacks the consideration of a detailed combustor heat transfer model

Based on the comprehensive literature review and Table 2-2, the combustor heat soakage used by other heat soakage model are based on the mean temperature of inlet and outlet conditions or was even neglected during the heat transfer effect analysis. However, the temperature distribution along the combustor liner is non-linear due to the different combustion processes along the combustor. For conventional rich burn combustor, three zones are included: primary zone, secondary zone and dilution zone. For the modern low emission combustors, design like pilot and main stages, premixing are applied [148]. These will lead different temperature distribution along the liner and different heat transfer process. In order to have a more realistic prediction on combustor heat flow rate estimation, a proper combustor model that consider the temperature distribution should be established.

2. Current heat transfer effect analysis lacks the consideration of cooling technologies

Based on the comprehensive literature review and Table 2-2, it was found that the cooling technologies are neglected by other researchers when establishing the heat soakage model. For components under high temperature, such as high-pressure turbine, cooling technologies like thermal barrier coating, film cooling and internal cooling are applied to reduce the extreme high temperature effect. For combustor, cooling technologies like film cooling, multijet impingement and transpiration cooling are applied [149]. In

order to have a more realistic estimation, a heat soakage model with the consideration of cooling technologies should be developed.

3. Creating compressor map modification model for individual engine components

Compared to the attention given to the heat soakage and tip clearance effects, the modification of component characteristic maps has received relatively little consideration. Based on the comprehensive literature review in Section 2.2.3 and Table 2-3, only two novel methodologies have been proposed in the past 20 years. Naylor has highlighted that the most accurate, easiest, and quickest method for calculation is the component map look-up approach. Current component performance maps are typically based on adiabatic conditions. One potential solution is to create diabatic component maps that include the heat soakage and tip clearance movement effects and use them during transient simulations. Kiss's method follows this idea, but each composite map can only be applied to a specific transient process. However, to capture the effects of heat soakage on the whole engine performance in a variety of transient operating conditions, a more convenient compressor map modification model should be implemented. The model should capture diabatic information on the relationship between heat flux, tip clearance, rotational speed, and mass flow for engine compressor. Based on the compressor map modification model, extremely fast and accurate results under different gas turbine transient conditions can be obtained.

4. Current gas turbine simulation platform lacks the consideration of component map change during transient simulation

Based on the comprehensive review in Section 2.3 and Table 2-4, it is found that current state-of-art simulation platform does not consider the component map change effect on gas turbine transient performance. In order to have a comprehensive view on heat transfer effect, component map change model should be considered.

2.5 Author's Publication

This author's research work in the area of heat transfer effect on gas turbine transient performance resulted in two journal publications which are briefly mentioned here.

In 2023, a review paper related to gas turbine transient simulation and heat transfer effect on transient performance is published in applied thermal engineering with DOI: <https://doi.org/10.1016/j.applthermaleng.2023.121523>. This paper discusses the current state-of-art gas turbine transient simulation methodologies and heat transfer simulation methodologies. This published paper is related to Section 2 of this thesis.

In 2024, a research paper related to the proposed quasi-2D thermal network based heat soakage model is published in applied thermal engineering with DOI: <https://doi.org/10.1016/j.applthermaleng.2024.122340>. This paper discusses the improvement in heat soakage model by establishing a comprehensive heat soakage model with the consideration of combustor temperature distribution, film cooling, internal cooling and thermal barrier coating. The comparison between the proposed model and conventional heat soakage model reveals the importance of a more realistic heat soakage model during gas turbine transient simulation. This published paper is related to Section 3.3, Section 4.4, and Section 5.2 of this thesis.

3 Methodology

3.1 Introduction

The purpose of this chapter is to describe the establishment of the proposed heat transfer effect model for gas turbine transient simulation modification. Four main parts are included: the geometry identification model, the heat soakage model, tip clearance change model and component characteristic map modification model. Figure 3-1 illustrates the calculation flow of the overall project.

The gas turbine steady stage performance (including design point and off design point) is calculated in the beginning. Based on the design point performance, the component geometry calculation is launched to estimate the component mass and wetted area. Also, the tip clearance datum condition is calculated based on design point condition. Then, based on the last off design point performance, the initial condition of each component for further heat transfer calculation is determined. When the code enters the transient performance calculation, the heat soakage effect, tip clearance change effect and compressor map change effect are considered to modify the gas turbine transient performance. During the heat soakage effect estimation, the quasi-2D thermal network is established based on the geometry identification outcome. With the quasi-2D thermal network, the heat transfer process between the gas turbine metal and working fluid can be estimated and component outlet temperature can be modified. Then the tip clearance change is estimated based on the component thermal expansion, centrifugal stress expansion and pressure stress expansion. Compressor and turbine efficiency are modified based on the tip clearance change. Finally, the compressor map is updated based on the heat transfer rate by using the numerical correlations and compressor working condition will be updated. Through this workflow, the heat transfer effect on gas turbine transient performance can be analysed with the consideration both thermal (heat transfer) and non-thermal (mechanical) factors.

This working flow offers a systematic approach for heat transfer effect analysis. Comparing with other researchers' work, the novel points are the inclusion of

important parameters for heat transfer modelling such as combustor temperature distribution and cooling technologies including film cooling, internal cooling and thermal barrier coating. Therefore, the proposed model is able to capture the physics better improving the transient simulation. In the component map modification part, a novel compressor map modification model is developed. By using this model, numerical correlations for specific compressor that can be used to modify adiabatic compressor map into non-adiabatic map during transient simulation can be derived. A parallel comparison between the proposed model with other researcher's work is shown in Table 3-1. Only researcher's work in recent 20 years is selected. GasTurb is included due to it is a representative state-of-art simulation platform. Li and Naylor's work are included because their work offer a comprehensive view on heat transfer effect. Kiss's work is included because this is the only contribution in compressor map modification in recent 20 years. The area author made major contribution is highlighted in green. The area author made minor contribution is highlighted in orange. The area author made no contribution is highlighted in red.

In this chapter, the methodology development for heat transfer effect on gas turbine transient performance, including component geometry identification, heat soakage effect estimation, tip clearance change estimation, and compressor map modification will be discussed.

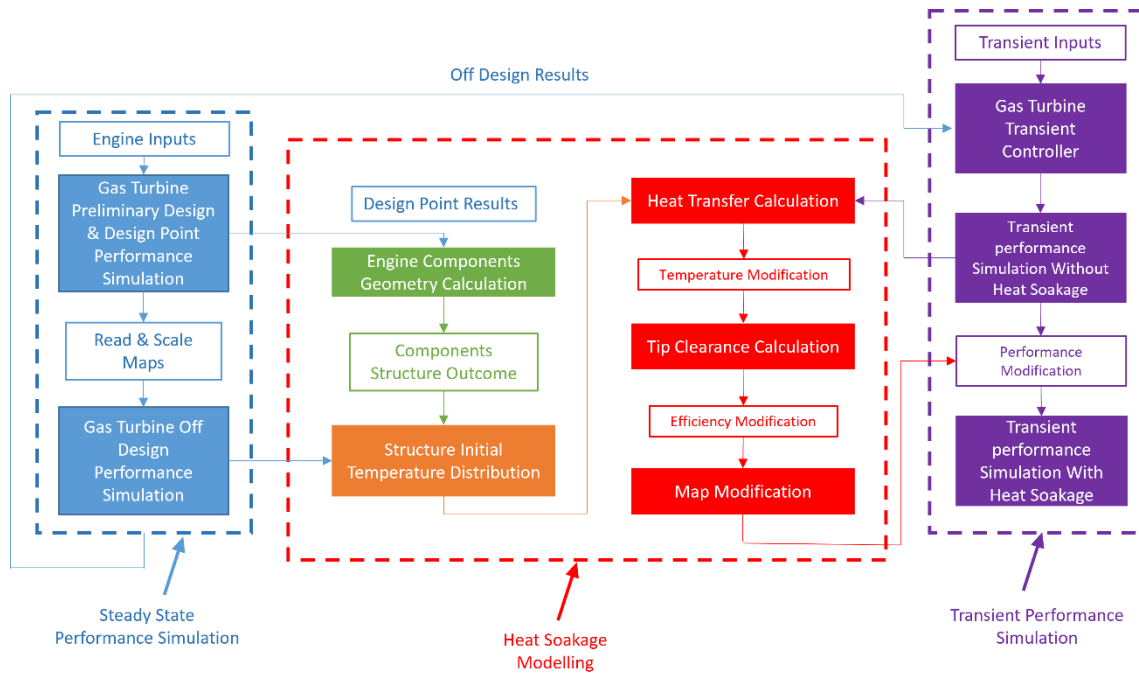


Figure 3-1 Workflow of the Overall Project

Table 3-1 Comparison between proposed model and other models

		Proposed Model	GasTurb [140]	Li [150]	Naylor [69]	Kiss [151]
Heat Soakage	Model Dimension	Quasi-2D Model	0D	1D based on axial direction	2D	Not Considered
	Convective Heat Transfer	Considered	Considered	Considered	Considered	
	Conductive Heat Transfer	Considered	Not Considered	Considered	Considered	
	Radiative Heat Transfer	Considered	Not Considered	Considered	Not Considered	
	Combustor Temperature Model	Temperature Distribution	Mean Temperature	Mean Temperature	Not Considered	
	Film Cooling	Considered	Not Considered	Not Considered	Not Considered	
	Internal Cooling	Considered	Not Considered	Not Considered	Not Considered	
	Thermal Barrier Coating	Considered	Not Considered	Not Considered	Not Considered	
Tip Clearance	Model Dimension	1D Model	0D	1D	2D	Not Considered
	Blade Length Change	Considered	Considered (Based on 0D Model)	Considered (Based on 0D Model)	Considered	
	Casing Radius Change	Considered (Based on 1D Model)	Considered (Based on 0D Model)	Considered (Based on 0D Model)	Considered	
	Disc Radius Change	Considered (Based on 1D Model)	Considered (Based on 0D Model)	Considered (Based on 0D Model)	Considered	

	Efficiency Model	Empirical	Empirical	Numerical	Empirical	
Component Map Modification	Model Type	Numerical Correlation	Not Considered	Considered but no improvement	Not Considered	Composite Map
	Flexibility	High				Low
	Accuracy	High				High

3.2 Component Geometry Identification

The heat transfer effect is strongly related to the geometry of the components, as it will affect the components' mass, wetted area, and heat transfer coefficient. The change in component geometry will lead to a variation in heat transfer rate during gas turbine transient process and further effect the gas turbine transient performance. As shown in Figure 3-2, gas turbine will include three main components: compressor, combustor, and turbine. In this section, the methodology development of component geometry identification will be discussed, three components are considered: the compressor, combustor, and turbine. They will be discussed separately.

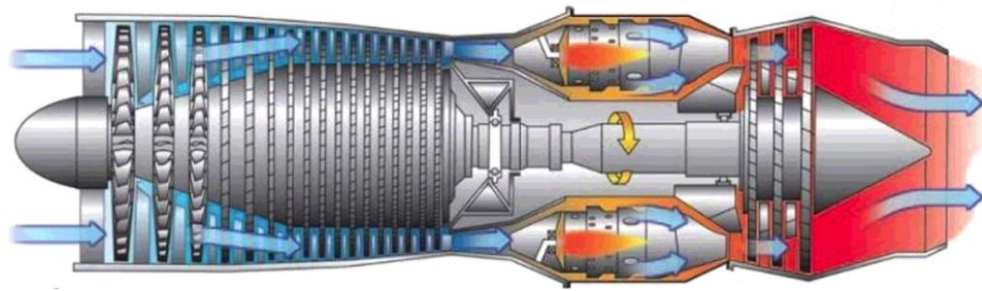


Figure 3-2 Cross Sectional Diagram of Gas Turbine Engine [152]

3.2.1 Compressor Geometry Identification

The gas turbine compressor is responsible for compressing the incoming air. It typically consists of multiple rotating and stationary blades arranged in rows. The cross-sectional diagram of a compressor is shown in Figure 3-2. The compressor includes three main parts: blades, casing and disc. Therefore, the equivalent compressor geometry model can be made based on the geometry of the flow path with the consideration of those three parts mentioned above, as shown in Figure 3-3.

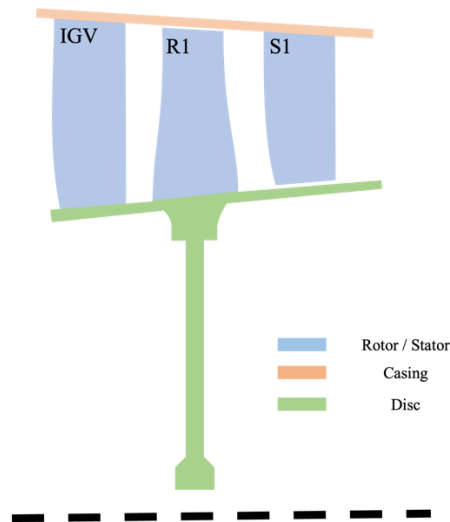


Figure 3-3 Equivalent compressor geometry model

Before estimating the compressor geometry, several data are required, as listed in Table 3-2. These data can be categorized into two types: performance data and geometry data. Performance data refers to the information utilized during the design point calculation. Geometry data encompasses user-defined component geometry details, read from the .hhs file elaborated upon in Section 7.8C.1.1.

Table 3-2 Input data during the compressor geometry identification

	Parameters	Unit
Performance Data	Compressor Design Point Pressure Ratio	N/A
	Compressor Design Point Mass Flow	Kg/s
	Compressor Design Point Efficiency	%
Geometry Data	Compressor number of stage	N/A
	Compressor shaft radius	m
	Compressor Inlet Hub/Tip Ratio	N/A
	Design Point Inlet Mach Number	N/A
	Annulus Configuration	N/A

	Compressor Blade Aspect Ratio	N/A
	Compressor Blade Pitch Chord Ratio	N/A
	Compressor Mean Blade Thickness	% of Chord
	Compressor Casing Thickness	m

Several assumptions are made during the geometry estimation:

- (1) Compressor stage temperature rise is kept constant;
- (2) The total temperature crosses the stator is kept constant;
- (3) The axial flow velocity at each stage inlet position is kept constant;
- (4) Blade is simplified as flat plate, casing is simplified as hollow cylinder and disc is simplified as hollow constant thickness disc;
- (5) The annulus configuration of the compressor is simplified into three types: constant hub diameter, constant tip diameter, and constant mean diameter, as shown in Figure 3-4.

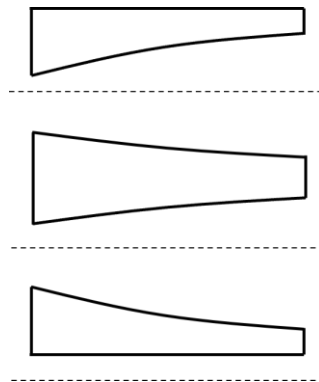


Figure 3-4 Different Compressor Geometries

Before estimating the compressor geometry, the gas turbine design point performance simulation will be launched first to obtain the total temperature and total pressure at the inlet and outlet position of the compressor. These two values are used to estimate the temperature and pressure distribution along the compressor axial direction. The stage temperature rise will be calculated by Eqs. (3-1). With the stage temperature rise and compressor inlet condition, the inlet

and outlet total temperature of each stage can be determined. Then the stage pressure ratio can be estimated by using Eqs. (3-2) and the axial velocity is estimated by Eqs. (3-3).

$$\Delta T_{stage} = \frac{T_{outlet} - T_{inlet}}{N_{stage}} \quad (3-1)$$

$$PR_{stage} = \left(\frac{\eta_{Poly} \cdot \Delta T_{stage}}{T_{stage\ inlet}} + 1 \right)^{\frac{\gamma}{\gamma-1}} \quad (3-2)$$

$$V_m = \sqrt{\gamma \cdot R \cdot \frac{T_{inlet}}{1 + \frac{\gamma-1}{2} M_{inlet}^2}} \cdot M_{inlet} \quad (3-3)$$

M_{inlet}	Compressor inlet Mach number
N_{stage}	Compressor stage number
PR_{stage}	Pressure ratio of compressor stage
R	Gas constant
ΔT_{stage}	Compressor stage temperature rise
T_{outlet}	Compressor outlet total temperature
T_{inlet}	Compressor inlet total temperature
$T_{stage\ inlet}$	Total temperature at each stage inlet plane
V_m	Axial velocity
η_{Poly}	Compressor polytropic efficiency
γ	Ratio of specific heats

With the total temperature and total pressure distribution as well as the mean axial velocity, the static parameters and the velocity at each stage can be determined, as shown in Eqs. (3-4) to (3-7).

$$T = t + \frac{V_m^2}{2 * Cp} \quad (3-4)$$

$$a = \sqrt{\gamma * R * t} \quad (3-5)$$

$$M_{local} = \frac{V_m}{a} \quad (3-6)$$

$$\frac{P}{p} = \left(1 + \frac{\gamma - 1}{2} * M_{local}^2\right)^{\frac{\gamma}{\gamma - 1}} \quad (3-7)$$

a Speed of sound

Cp Specific heat

P Total pressure

p Static pressure

t Static temperature

Then, the flow function illustrated by (3-8) and (3-9) will be used for the flow area calculation.

$$Q = M_{local} \cdot \sqrt{\frac{\gamma}{R}} \cdot \left(1 + \frac{\gamma - 1}{2} M_{local}^2\right)^{\left(-\frac{\gamma + 1}{2(\gamma - 1)}\right)} \quad (3-8)$$

$$A = \frac{W \sqrt{T}}{P Q} \quad (3-9)$$

A Flow area at each stage plane

Q Flow function at each stage plane

W Mass flow at each stage plane

The inlet position geometry will be determined first. Based on the hub tip ratio and calculated flow area, the tip radius can be estimated by using Eqs. (3-10), the hub radius can be estimated by using Eqs. (3-11), and the mean radius can be estimated by using Eqs. (3-12).

$$r_{tip} = \left(\frac{A}{\pi * \left(1 - \left(\frac{r_{hub}}{r_{tip}} \right)^2 \right)} \right)^{\frac{1}{2}} \quad (3-10)$$

$$r_{hub} = r_{tip} * \frac{r_{hub}}{r_{tip}} \quad (3-11)$$

$$r_{mean} = \frac{r_{hub} + r_{tip}}{2} \quad (3-12)$$

r_{tip} Tip radius at the compressor inlet plane

r_{hub} Hub radius at the compressor inlet plane

r_{mean} Mean radius at the compressor inlet plane

If the annulus configuration is selected as constant tip diameter, then the r_{tip} will remain constant for the following stages. If the annulus configuration is selected as constant mean diameter, then the r_{mean} will remain constant for the following stages. If the annulus configuration is selected as constant hub diameter, then the r_{hub} will remain constant for the following stages. Combined with the stage flow area, the geometry of each stage can be determined. Figure 3-5 illustrates an example of one compressor annulus dimension based on the constant tip diameter.

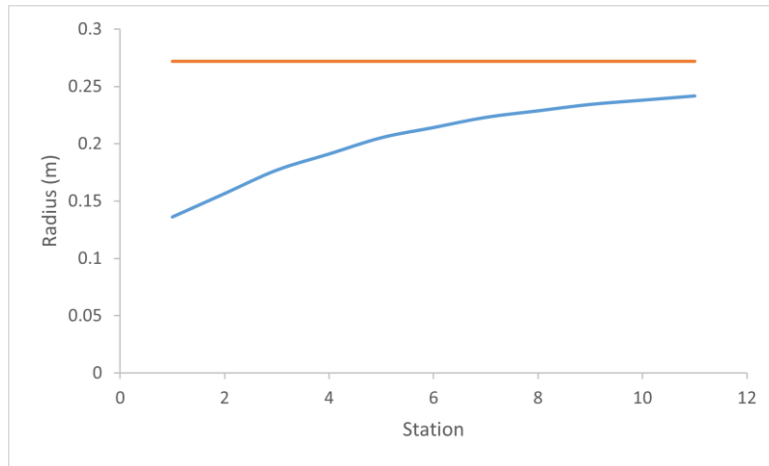


Figure 3-5 Calculation Example of the Annulus Dimension of Compressor

The number of blades for each stage will be estimated by using Eqs. (3-13).

$$N_b = \pi \cdot r_{mean} / (h_b / (APR \cdot PCR)) \quad (3-13)$$

$$h_b = r_{tip} - r_{hub} \quad (3-14)$$

APR	Blade aspect ratio
h_b	Blade hight at each stage
N_b	Number of blades at each stage
PCR	Blade pitch chord ratio

When estimating the mass and wetted area, the blades are assumed to be flat plates, the casings are assumed to be hollow cylinders and discs are assumed to be a constant thickness hollow disc. The chord of the blade is estimated by using Eqs. . it is assumed that the blade stagger angle equals to 45° , which lead to the stage length equals to $\frac{\sqrt{2}}{2}$ blade chord. Based on the above calculation, the wetted area of each component part can be determined. Once the material density is known, the mass can be estimated. Related equations are listed in Eqs. (3-15) to (3-21).

$$Chord_b = h_b / APR \quad (3-15)$$

$$A_b = N_b \cdot 2 \cdot h_b \cdot Chord_b \quad (3-16)$$

$$A_c = \pi \cdot 2 \cdot r_{tip} \cdot SL \quad (3-17)$$

$$A_d = \pi \cdot 2 \cdot r_{hub} \cdot SL \quad (3-18)$$

$$m_b = A_b \cdot w_b \cdot \rho_b / 2 \quad (3-19)$$

$$m_c = A_c \cdot w_c \cdot \rho_c \quad (3-20)$$

$$m_d = \frac{\pi(D_h^2 - D_{shaft}^2)}{4} \cdot SL \cdot \rho_d \quad (3-21)$$

A	Wetted area of each component part
SL	Stage axial length
w	Thickness or axial width of component
m	Mass of each component part
ρ	Density of the material

3.2.2 Turbine Geometry Identification

The turbine is responsible for expanding the incoming hot gas and extract the energy to drive the compressor on the same shaft. Similar with compressor, it also consists of multiple rotating and stationary blades arranged in rows, as shown in Figure 3-2. The turbine geometry is very similar to compressor, which also includes blade, casing and disc. The only difference is the inlet area is much smaller than the outlet area. Thus, the similar geometry identification approach of the compressor can be applied to the turbine. The equivalent turbine geometry model is shown in Figure 3-6.

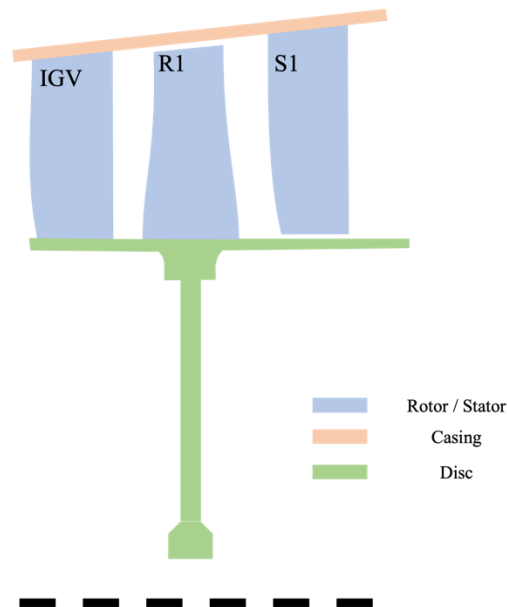


Figure 3-6 Equivalent turbine geometry model

The required data for turbine geometry identification is listed in Table 3-3. Same as the compressor, two types of data are required: the performance data and the

geometry data. For the design point mass flow and efficiency, they are obtained based on gas turbine design point data, and the geometry data are controllable input.

Table 3-3 Input data during the turbine geometry identification

	Parameters	Unit
Performance Data	Turbine Design Point Mass Flow	Kg/s
	Turbine Design Point Efficiency	%
Geometry Data	Turbine number of stage	N/A
	Turbine shaft radius	m
	Turbine Inlet Hub/Tip Ratio	N/A
	Design Point Inlet Mach Number	N/A
	Annulus Configuration	N/A
	Turbine Blade Aspect Ratio	N/A
	Turbine Blade Pitch Chord Ratio	N/A
	Turbine Mean Blade Thickness	% of Chord
	Turbine Casing Thickness	m

The same assumptions are applied to the turbine:

- (1) Turbine stage temperature drop is kept constant;
- (2) The total temperature crosses the stator is kept constant;
- (3) The axial flow velocity at each stage inlet position is kept constant;
- (4) The blade is simplified as flat plate, the casing is simplified as hollow cylinder and disc is simplified as hollow constant thickness disc;
- (5) The annulus configuration of the compressor is simplified into three types: constant hub diameter, constant tip diameter, and constant mean diameter.

Then the procedure of compressor geometry calculation can be applied to the turbine. By using the same method, the wetted area and mass of each turbine stage can be estimated.

3.2.3 Combustor Geometry Identification

The combustor of the modern gas turbine engine has three main types: tubular combustor, tubo-annular combustor and annular combustor. Although the geometry design of these three types of combustors is different, they can all be treated as hollow cylinders with certain mass and wetted area. Hence, the equivalent combustor configuration could be defined in Figure 3-7, where the green part represents the fuel spray, the blue part represents the combustor outer casing, and the orange part represent the flame tube.

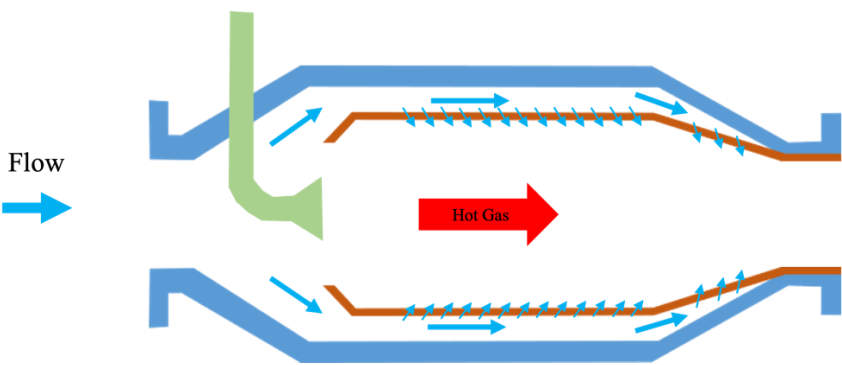


Figure 3-7 Combustor Equivalent Configuration

During the combustor geometry design, the outer casing of the combustor will not be considered since it will not directly contact the high-temperature flame. Only the flame tube will be considered during the combustor preliminary design. Three types of combustors are considered: tubular combustor, tubo-annular combustor, and annular combustor.

Before estimating the geometry of the combustor, several data are required, as listed in Table 3-4. For combustor pressure loss, it is read from the Turbomatch input file. For the rest of the data, they are read from the .hhs file.

Table 3-4 Input data during the combustor geometry identification

	Parameters	Unit
--	------------	------

Performance Data	Combustor Pressure Loss	%
Geometry Data	Combustor Type	N/A
	Combustor Length	m
	Number of Flame Tube	N/A
	Flame Tube Wall Thickness	m
	Combustor Casing Inner Diameter	m

The combustor reference area will then be calculated by using Eqs. (3-22).

$$A_{ref} = \left(\frac{R}{2} \left(\frac{W_3 \cdot \sqrt{T_3}}{P_3} \right)^2 \cdot \frac{\Delta P_{3-4}}{q_{ref}} \cdot \left(\frac{\Delta P_{3-4}}{P_3} \right)^{-1} \right)^{0.5} \quad (3-22)$$

$\frac{\Delta P_{3-4}}{q_{ref}}$ Pressure drop factor (PDF)

ΔP_{3-4} Combustor pressure loss

PDF's value varies for different types of combustors. For the tubular combustor, the PDF is 37; for the tubo-annular combustor, it is 28; and for the annular combustor, it is 20 [149].

With the estimated reference area, Eqs. (3-23) will be applied to estimate the sectional area of the combustor, as we are only considering the flame tube instead of the entire combustor. Eqs. (3-24) will be used to estimate the tubular and tubo-annular flame tube diameter.

$$A_L = 0.66 \cdot A_{ref} \quad (3-23)$$

$$D_{tube} = \sqrt{A_L \cdot 4 / (N_{tube} \cdot \pi)} \quad (3-24)$$

A_L Sectional area of the combustor

D_{tube} Diameter of tubular and tubo-annular flame tube

N_{tube} Number of tubular and tubo-annular flame tubes

For the annular combustor, the structure is illustrated in Figure 3-8. Since the area is between two hollow cylinders, we need to calculate the outer and inner diameter of the cylinder by using $dR = (D_{ref} - D_L)/2$. The equations used for estimation are listed in Eqs. (3-25) to (3-28).

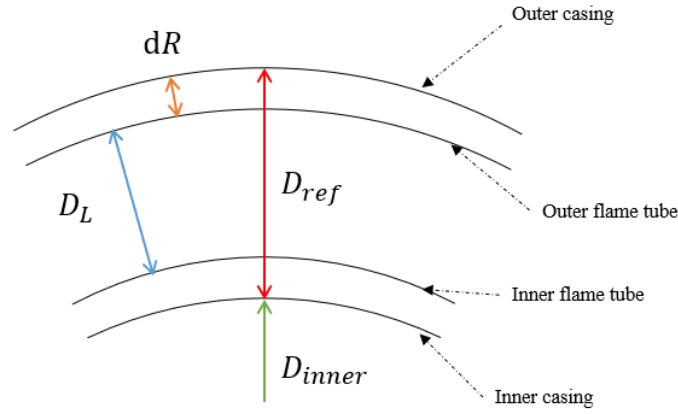


Figure 3-8 Structure illustration of annular combustor

$$D_{ref} = \left(\sqrt{4 \cdot A_{ref} / \pi + D_{inner}^2} - D_{inner} \right) / 2 \quad (3-25)$$

$$dR = (D_{outer}^2 - D_{inner}^2 - 4 \cdot A_L / \pi) / (8 \cdot (D_{outer} + D_{inner})) \quad (3-26)$$

$$D_{ft_outer} = D_{outer} - 2 \cdot dR \quad (3-27)$$

$$D_{ft_inner} = D_{inner} + 2 \cdot dR \quad (3-28)$$

D_{ref} Reference diameter of annular combustor

dR Radius gap between casing and flame tube

D_{ft_outer} Outer flame tube diameter

D_{ft_inner} Inner flame tube diameter

The flame tube is approximated as a hollow cylinder when estimating the mass and wetted area. The total mass and wetted area can be estimated by using Eqs. (3-29) and (3-30).

$$A_{burner} = \pi \cdot (D_{ft_{outer}} + D_{ft_{inner}}) \cdot L_{burner} \quad (3-29)$$

$$m_{burner} = A_{burner} \cdot Th_{burner} \cdot \rho_{burner} \quad (3-30)$$

L_{burner} Axial length of the flame tube

Th_{burner} Flame tube wall thickness

Then, the combustor model will be divided into three parts: the primary zone, secondary zone, and the dilution zone. It is assumed that these three zones have the same length, mass and wetted area. So, the zone mass and wetted area are estimated by using Eqs. (3-31) and (3-32).

$$A_{zone} = \frac{1}{3} * A_{burner} \quad (3-31)$$

$$M_{zone} = \frac{1}{3} * M_{burner} \quad (3-32)$$

3.3 Heat Soakage Model

The geometry identification model provides the basement of the heat soakage model. Since the wetted area and mass of each part of each component have been identified, heat soakage model can be established for each component and estimate the heat flow rate between the hot gas and the component metal. The overall calculation flow for the proposed heat soakage model is shown in Figure 3-9. Based on the component identification model, a 0D and 1D heat soakage model suitable for different parts are applied for the heat flow estimation. Then the component performance will be modified based on the estimated heat flow rate. In this section, the development of heat soakage models suitable for different component will be discussed, including heat flow rate estimation model,

establishment of quasi-2D thermal network and component performance modification model.

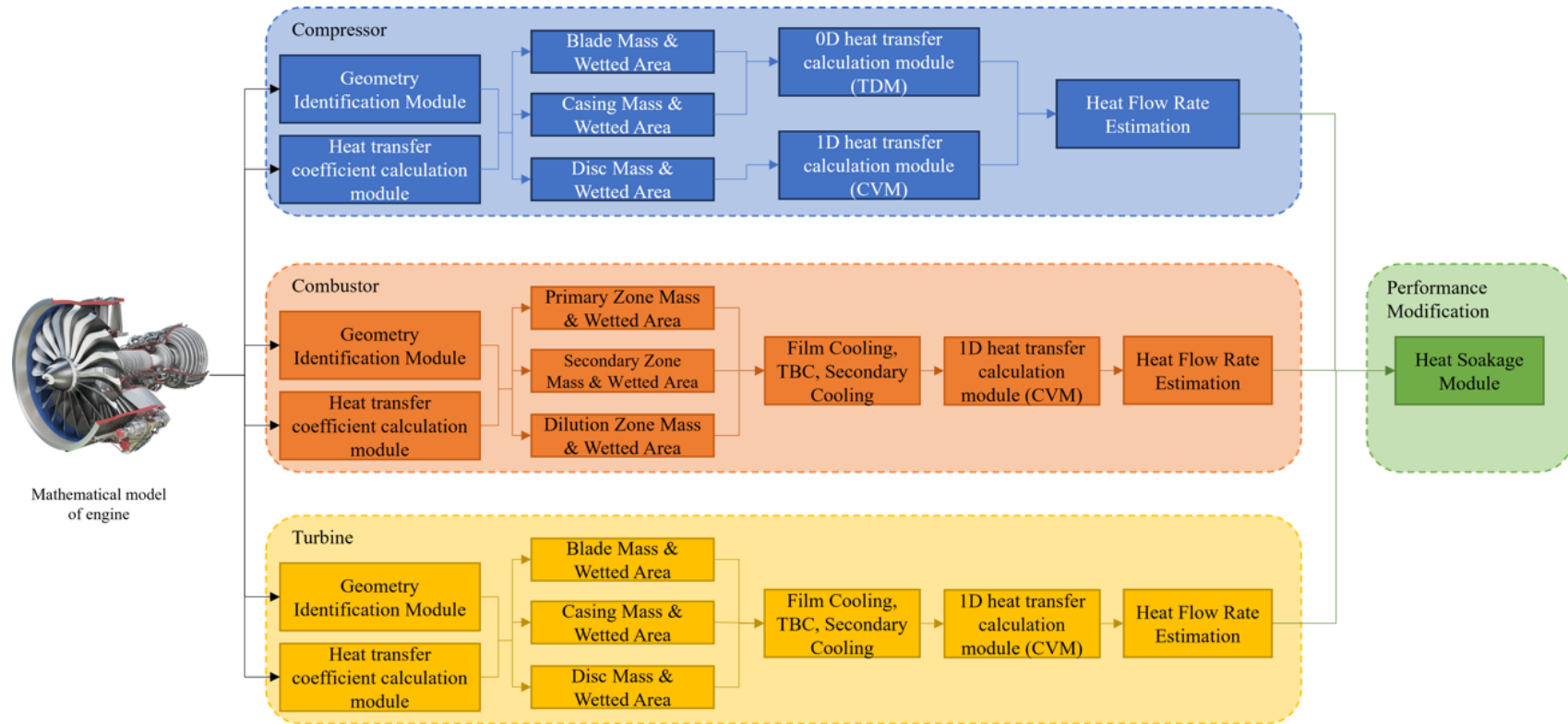


Figure 3-9 Heat soakage estimation logic during engine transient

3.3.1 Component Heat Flow Rate Estimation

Based on the geometry identification method mentioned above, the mass and wetted area for each compressor and turbine stage as well as the flame zone are estimated. Then, the heat flow rate can then be estimated. In order to have a more realistic prediction of the heat flow rate, two types of methods are developed. One is the temperature difference model (TDM), which is a 0D model and is suitable for low Biot number parts. The other is the control volume model (CVM), which is a 1D model. It has the capability to capture the internal temperature distribution and is suitable for high Biot number parts. Three types of cooling technologies are considered: film cooling, internal cooling and thermal barrier coating. In this section, the development of the proposed heat transfer model will be discussed.

3.3.1.1 Temperature Distribution Identification

Before estimating the heat flow rate during the transient process, it is necessary to identify the temperature distribution in each component. Figure 3-10 illustrates the temperature distribution in axial direction along the compressor. It can be clearly found that the temperature remains the same when crossing the stator of the compressor and rises linearly through the rotor. For each stage, the temperature rise has a slight difference. So, in our case, it was assumed that each stage of the compressor has the same temperature rise. In each stage, temperature rises through the rotor and keeps constant through the stator. The mean temperature based on the inlet and outlet station of rotor or stator will be adopted during the heat flow rate estimation. For the turbine, the same assumption was adopted. The temperature drop of each stage in turbine is assumed constant. Temperature drops across the rotor and keeps constant across the stator.

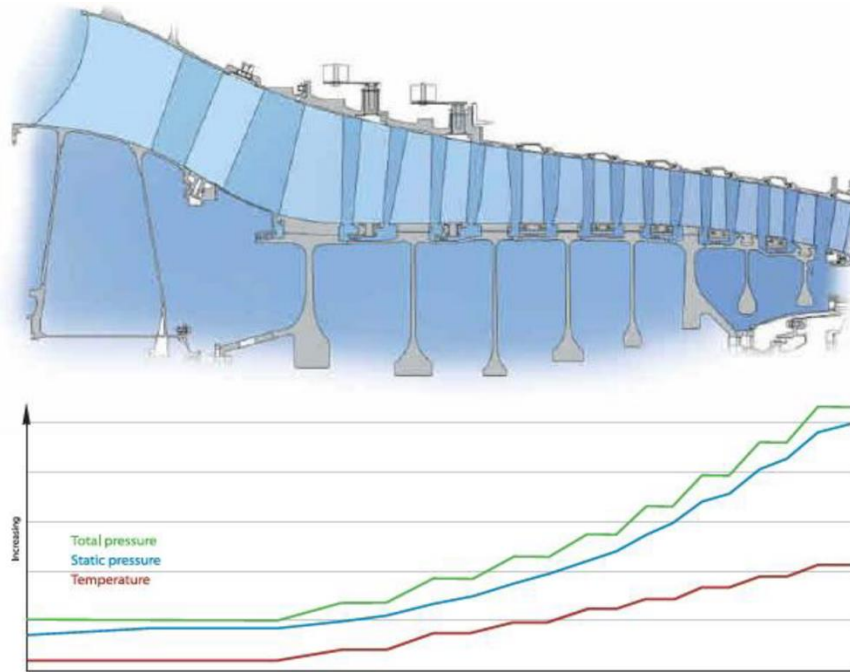


Figure 3-10 Temperature and Pressure Profile of Compressor

The combustor temperature distribution model is based on the conventional rich burn combustor. The model divides the combustor flame tube into different zones in the axial direction, each with its own temperature. Specifically, three combustor zones are considered: the primary zone, secondary zone, and dilution zone. It is assumed that 40% of the air will enter the primary zone for combustion, an additional 50% of air will enter the secondary zone, and the remaining 10% will enter the dilution zone. Based on the air distribution, the fuel to air ratio at each zone can be estimated by using

$FAR = \frac{W_{fuel}}{W_{air}}$	(3-33)
----------------------------------	---------------

The temperature profile for each zone is calculated by Eqs. (3-34). C_p is calculated based on the temperature by using equations listed in 7.8Appendix B.

$LHV \cdot FAR \cdot \eta_{burner} = C_p \cdot (T_{out} - T_{in})$	(3-34)
--	---------------

LHV Lower heating value of the fuel

FAR Fuel to air ratio in each combustor zone

η_{burner} Efficiency of the combustor

To achieve a more accurate estimation on temperature, an iteration loop is created as shown in Figure 3-11. The zone temperature will be guessed at the beginning to offer an initial value of C_p . Then, new zone temperature will be calculated based on the estimated C_p . The difference between last and current temperatures is calculated. If the difference fit the convergence criterion, the zone temperature is assumed to be found. Otherwise, C_p will be updated based on the new zone temperature.

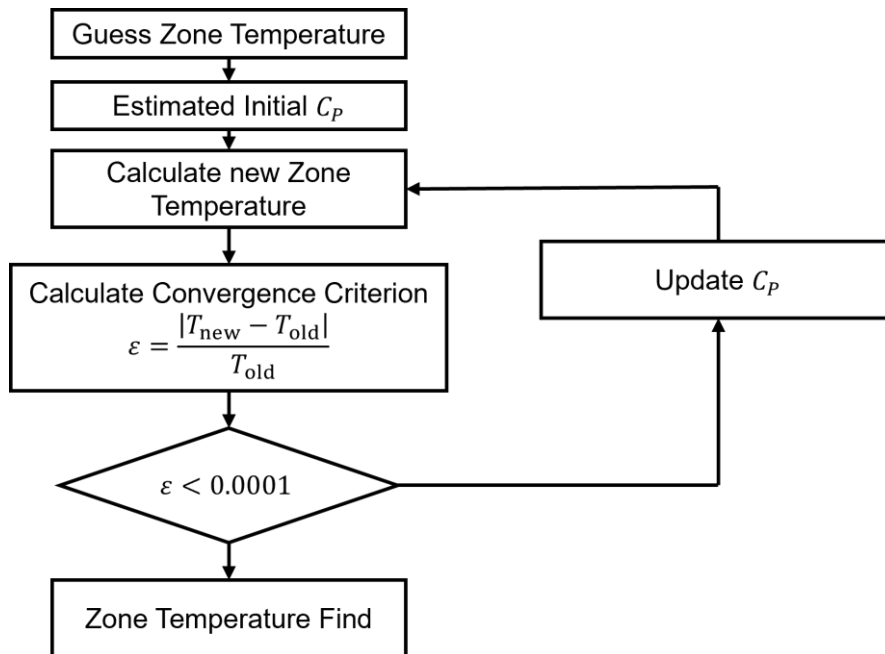


Figure 3-11 Flow chart for combustor zone temperature calculation

3.3.1.2 Heat Transfer Model for Low Biot Number Parts

For the gas turbine engine, the heat transfer process in different component varies a lot. The compressor works under a relative cooler condition and the heat transfer process is slight. The combustor will experience a severe heat transfer process because the combustion occurs, and the temperature will reach thousands Kelvin. Turbine will also experience a severe heat transfer process

cause the hot gas expanded in this region. The heat transfer process varies thus different heat transfer model should be selected. In this section, heat transfer model suitable for compressor blades and casing will be discussed. For blade and casing, the Biot number, which is the ratio of thermal-conduction resistance to the heat transfer resistance on the unit area, is relatively low ($Bi < 1$). Thus, their conduction resistance is much smaller than the heat convection at the surface. It can be assumed that the temperature gradients inside them are negligible and have an even temperature distribution.

Under this assumption, the internal temperature distribution of a component now can be neglected, and it can be treated as a mass node located at the centre of its volume and surrounded by the fluid. The temperature difference between the fluid and the mass will lead to heat exchange and eventually both fluid and solid will share a common temperature and a steady state will have been reached.

The heat flow rate between the fluid and mass at any given instant can be estimated by using Eqs. (3-35).

$$\Phi = \frac{dQ}{dt} = h * A * (T_f - T_m) \quad (3-35)$$

Φ	Heat flow rate
Q	Quantity of heat
h	Heat transfer coefficient
A	Component wetted area
T_f	Fluid temperature
T_m	Metal temperature.

Integration of Eqs. (3-35) with respect to time over a given time interval will yield the magnitude of the quantity of heat passed from fluid to solid. Knowing the amount of heat transferred during the time interval we can calculate the new temperature of the solid and the fluid using Eqs. (3-36).

$$\Delta T = \frac{\Delta Q}{m * Cp} \quad (3-36)$$

m Mass of the component

As discussed in Section 2.2.1, the LPM (a type of 0D model), which assumes that the temperature field inside the component is uniform and treats the component as a mass node located at the centre of the component, is applied widely by different researchers to investigate the heat transfer process between the main gas fluid and the engine metal. Naylor had concluded several ways to estimate the quantity of heat for a certain time interval based on the LPM. The simplest way is assuming that the temperature of the fluid and mass keep constant at their start values, or their average values during the heat exchange process, as shown in Figure 3-12.

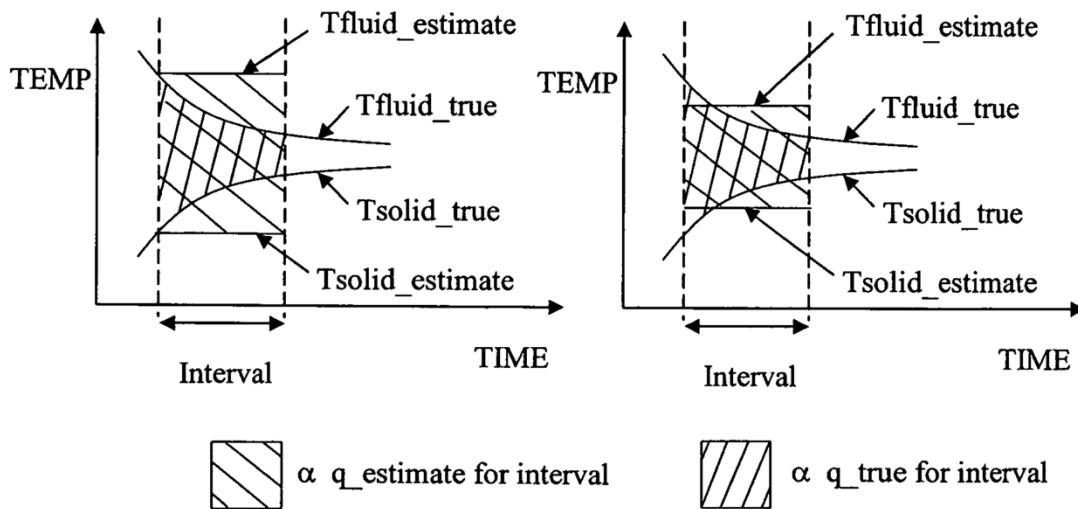


Figure 3-12 Heat Quantity Estimation Using Fixed Temperature [69]

Another slightly more sophisticated way is assuming that one remains the constant while the other of the bodies shows an exponential decay, as shown in Figure 3-13. Compared with constant temperature method, the exponential decay method can offer a more realistic estimation during the heat transfer process since the change in solid temperature is captured.

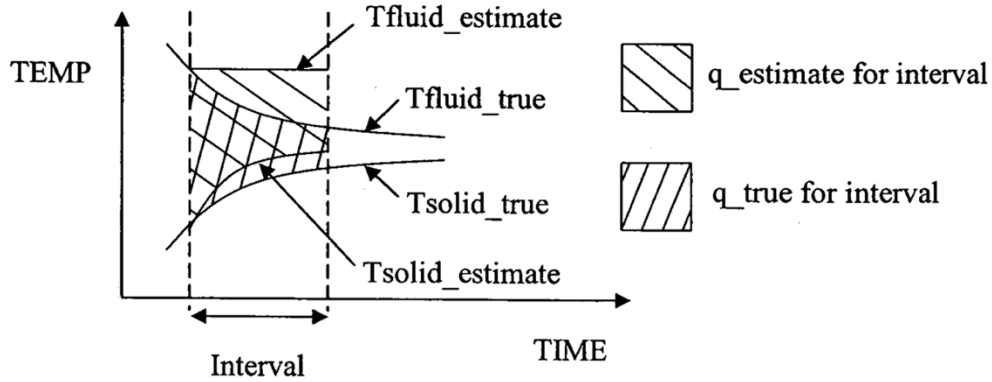


Figure 3-13 Heat Quantity Estimation Using Exponential Decay [69]

Based on the above discussion, it can be found that the traditional LPM assumes at least one constant temperature during the heat transfer process. This can lead to an overestimation of the quantity of heat transferred during the time interval, especially when the time interval is larger than the system response time. This can be improved by using small time interval. When the interval is small enough, it is reasonable to assume that the temperature of fluid and solid are kept constant. However, this will increase the computational time.

To have a more realistic heat transfer, similar approach with Naylor was adopted by allowing both fluid and solid temperature to change according to an exponential decay. Given the wetted area, heat transfer coefficient, fluid temperature, and mass temperature, the heat flow rate between the fluid and the mass can be estimated using Eqs. (3-35) and Eqs. (3-38).

By combining Eqs. (3-35) and (3-36), the relation between the heat flow rate and change in metal and fluid temperature can be established, as shown in Eqs. (3-37) and (3-38).

$$\Phi = m_m \cdot Cp_m \cdot \left(\frac{dT_m}{dt} \right) = h * A * (T_f - T_m) \quad (3-37)$$

$$\Phi = -m_f \cdot Cp_f \cdot \left(\frac{dT_f}{dt} \right) = h * A * (T_f - T_m) \quad (3-38)$$

Let $T_D = T_g - T_m$. Since both T_g and T_m are function of time, so T_D is also a function of time. Based on this, we can derive Eqs. (3-39)

$$dT_D = dT_g - dT_m \quad (3-39)$$

By combining (3-37), (3-38) and (3-39), Eqs. (3-40) can be derived.

$$\frac{dT_D}{dt} = - \left(\frac{h \cdot A}{M_m \cdot Cp_m} + \frac{h \cdot A}{M_f \cdot Cp_f} \right) * T_D \quad (3-40)$$

Integrating Eqs. (3-40) with respect to time once, the expression for the transient temperature of T_D as a function of time can be found, as shown in Eqs. (3-41). Then integrating Eqs. (3-41) with respect to time again, and let a time constant $\tau = \frac{h \cdot A}{M_m \cdot Cp_m} + \frac{h \cdot A}{M_f \cdot Cp_f}$, the quantity of heat transfer between fluid and mass in a certain time period can be estimated by Eqs. (3-42).

$$T_D = T_{D0} \cdot e^{-\left(\frac{h \cdot A}{M_m \cdot Cp_m} + \frac{h \cdot A}{M_f \cdot Cp_f}\right) \Delta t} \quad (3-41)$$

$$Q = \int_0^t \Phi dt = -h \cdot A \cdot (T_{g0} - T_{m0}) \cdot \frac{1}{\tau} \cdot (e^{-\tau t} - 1) \quad (3-42)$$

τ Time constant

T_{g0} Gas temperature at the beginning of time interval

T_{m0} Metal temperature at the beginning of time interval

The temperature of fluid and mass at the end of this time period can be estimated based on the estimated quantity of heat, as shown in Eqs. (3-43).

$$T_t = T_0 + \frac{Q}{m \cdot Cp} \quad (3-43)$$

Compared with the traditional LPM, this method can offer a more realistic estimation during the heat transfer process where either fluid or metal temperature will change. Compared with Naylor's method, this method neglects the effect coming from component thermal conductivity. Since it is only applied to

the low Biot number parts, where thermal conductivity is negligible, the accuracy can be guaranteed.

3.3.1.3 Heat Transfer Model for High Biot Number Parts

The OD method described in Section 3.3.1.2 is suitable for low Biot Number components. However, for those with a non-uniform temperature distribution (such as compressor discs, combustor flame tubes, turbine blades, casings, and discs), that method becomes unsuitable because it neglects the component internal thermal resistance. Besides, the OD model treats the component as a mass node located at the centre of its volume, the effect coming from the component shape was neglected. So, it is necessary to employ a more fitting model for these components.

To offer a more realistic heat transfer process estimation, the control volume method (CVM) is created. CVM is a type of 1D numerical solution. The idea is to capture the feature of the component heat transfer process. During this process, the effect coming from the convective heat transfer, conductive heat transfer, radiative heat transfer, thermal barrier coating, film cooling and internal cooling can all be captured. The calculation process of the CVM is illustrated in Figure 3-14, using the turbine disc as an example.

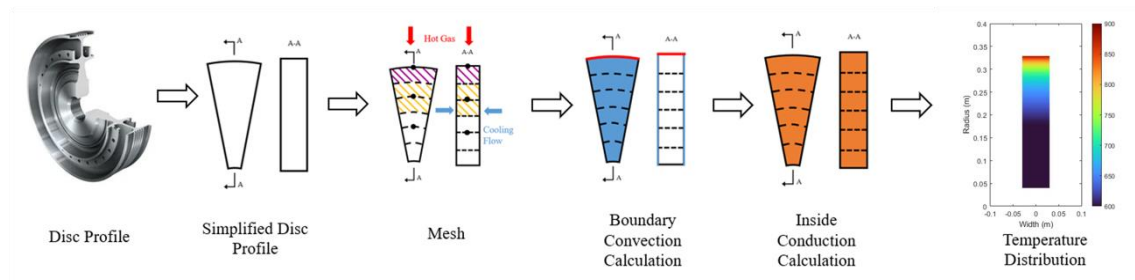


Figure 3-14 CVM calculation process (turbine disc as an example)

Five components are considered when applying the CVM, which are compressor disc, combustor flame tube, and turbine blade, casing and disc. As described in Section 3.2, the blade is simplified as flat plate, the casing is simplified as hollow cylinder, and the disc is simplified as hollow constant thickness disc.

The first step of CVM is to select the presentive section to capture the heat transfer process of the component and create the mesh for further calculation.

For compressor disc, it will work under a relative low temperature working condition. In the model, it is assumed that hot gas will flow through the disc outer rim and no cavity cooling is considered. Under this condition, the cross-sectional area is chosen as the analyzed part, with meshing also in the radius direction, as shown in Figure 3-15 (a) step 2 and step 3. During the meshing process, the distance between each node is kept the same. The volume of each node is estimated by Eqs. (3-44). The mass of each node is estimated by Eqs. (3-45). The outer and inner surface of each volume is estimated by using Eqs. (3-46) and (3-47).

$$V_{node} = \pi \cdot (r_{outer}^2 * w_{outer} - r_{inner}^2 * w_{inner}) - \frac{1}{3} \quad (3-44)$$

$$* (w_{outer} - w_{inner}) * (r_{outer}^2 + r_{inner}^2 + r_{outer} * r_{inner})$$

$$M_{node} = V_{node} * \rho \quad (3-45)$$

$$A_{outer} = 2 * \pi * r_{outer} * w_{outer} \quad (3-46)$$

$$A_{inner} = 2 * \pi * r_{inner} * w_{inner} \quad (3-47)$$

r	Radius of node volume outer and inner surface
w	Outer and inner surface axial width of node volume
A	Outer and inner surface of each volume

For combustor, the burning process occurs and the flame tube will experience extremely high temperature. Several cooling technologies are applied to the flame tube: film cooling, thermal barrier coating, and internal convective cooling. The film cooling technology and thermal barrier coating are applied to reduce the contacting temperature on the flame tube surface. The internal convective cooling is applied to remove the heat from the flame tube. In order to capture these cooling features, the flame tube wall (highlighted in the red box in Figure 3-15 (b)

step 2) is selected as the analyzed part. Meshing is performed in the radius direction, as shown in Figure 3-15 (b) step 3. The left end node represents the boundary node at hot gas side where convective and radiative heat transfer occur. The orange part represents the mesh volume for the thermal barrier coating. The blue part represents the mesh volume for the metal part. The right end node is the boundary node on the internal cooling flow side. The volume of each node is estimated by Eqs. (3-48). The mass of each node is estimated by Eqs. (3-49).

$$V_{node} = \pi * (r_{outer}^2 - r_{inner}^2) * w_{node} \quad (3-48)$$

$$M_{node} = V_{node} * \rho \quad (3-49)$$

For turbine blade, it will work under extremely high temperature condition since it located at the outlet of the combustor. Several cooling technologies are applied to turbine blade to avoid the thermal corrosion due to the high temperature. The internal cooling technology is applied to remove the heat from the blade metal. The film cooling technology and thermal barrier coating are applied to reduce the contacting temperature on the blade surface. Under this complex working condition, the blade wall (highlighted in the red box in Figure 3-15 (c) step 2) is chosen as the analyzed part, as it allows for the capture of film cooling, internal convective cooling, and TBC. Meshing is performed in the blade thickness direction, as shown in Figure 3-15 (c) step 3. The left end node is the boundary node on the hot gas side. The orange part represents the mesh volume for the thermal barrier coating. The blue part represents the mesh volume for the metal part. The right end node is the boundary node on the internal cooling flow side. The volume of each node is estimated by Eqs. (3-50). And the mass of each node is estimated by Eqs. (3-51).

$$V_{node} = \frac{1}{2} * A_b * w_{node} \quad (3-50)$$

$$M_{node} = V_{node} * \rho \quad (3-51)$$

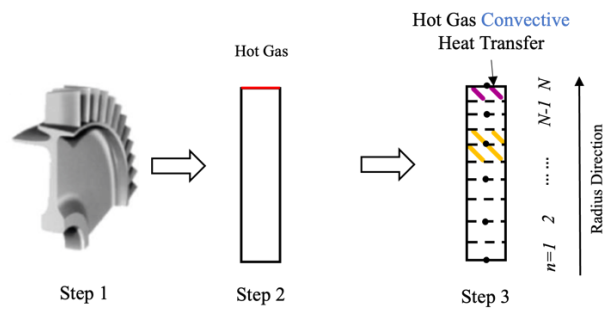
For turbine casing, the meshing process is as same as the meshing process of combustor. However, the turbine casing neglect the film cooling and thermal barrier coating.

For turbine disc, same as the turbine blade, it will also work under extremely high temperature. The hot gas will flow through the disc outer rim surface, and the disc cavity cooling flow will flow through the disc side surface for the cooling purpose. Under this complex working condition, the cross-sectional area is chosen as the analyzed part, with meshing also in the radius direction, as shown in Figure 3-15 (d) step 2 and step 3. The meshed volume has the same thickness in radius direction, but the wetted surface and mass of each volume are dependent on the radius of the volume. The top end boundary node represents the hot gas side, where convective heat transfer will occur. The bottom end boundary node represents the shaft side. For each volume, the side cooling area will also be considered due to the cavity cooling. The purple area represents the volume of the boundary node and the yellow area represents the volume of the internal node. The volume, mass and outer and inner surface of each node follow the same calculation process as the compressor disc. However, the side surface is required due to the cavity cooling. The side surface area is estimated by using Eqs. (3-52).

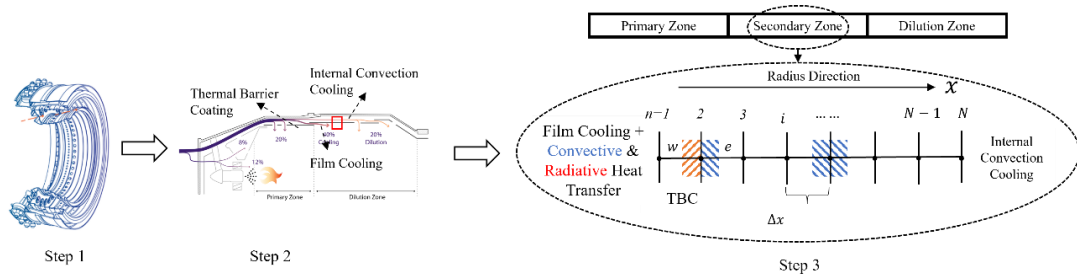
$$A_{side} = \pi * (r_{outer}^2 + r_{inner}^2) \quad (3-52)$$

The thermal barrier coating mentioned above is achieved by using additional layers of mesh, as shown in Figure 3-15 (e). It is assumed that thermal barrier coating has a perfect contact with the metal. The mass of node j in Figure 3-15 (e) is consists of two parts, one is the mass of thermal barrier coating, and the other is the metal. The estimation equation is shown in Eqs. (3-53).

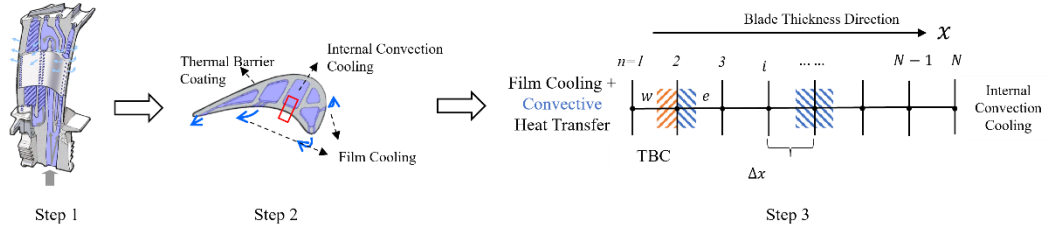
$$M_j = V_{TBC} * \rho_{TBC} + V_{metal} * \rho_{metal} \quad (3-53)$$



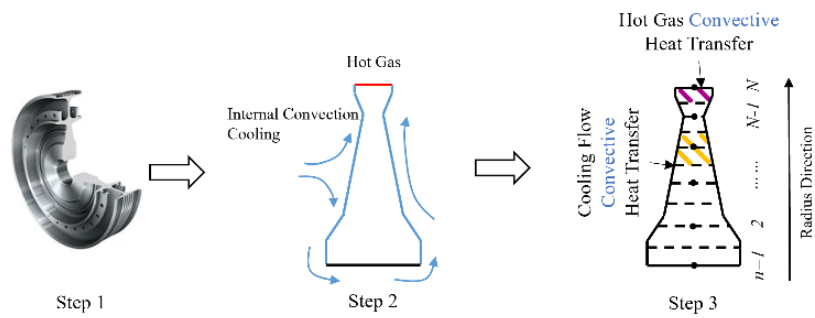
(a)



(b)



(c)



(d)

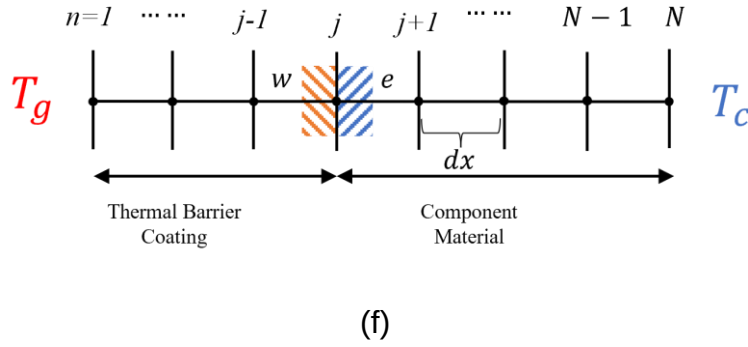


Figure 3-15 Component geometry feature capture and mesh

An example of the resulting mesh is shown in Figure 3-16. In the figure, the red curve signifies the hot gas convective heat transfer area of the boundary node, the green dashed curve represents the outer and inner conductive heat transfer areas of Node N, the blue line denotes the cooling area of Node N, and the purple and yellow areas correspond to the volume and mass of the boundary node and internal node, respectively.

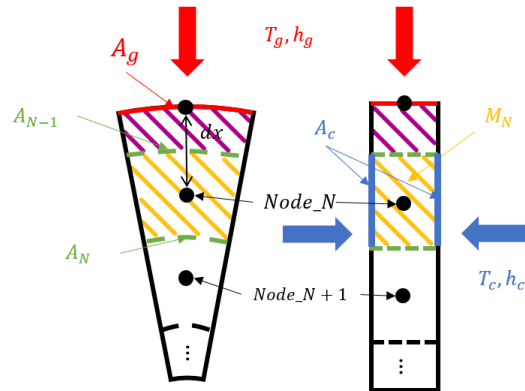


Figure 3-16 Mesh outcome (turbine disc as an example)

The mesh density has also been analyzed. Turbine disc mesh is chosen due to its complicated shape. The centre temperature change over time during a heat-up process is examined, as depicted in Figure 3-17. It is observed that when the mesh density exceeded 200 nodes per meter (nodes/m), the deviations caused by node density are sufficiently small to be negligible. Therefore, a mesh density of 200 nodes/m is selected for application.

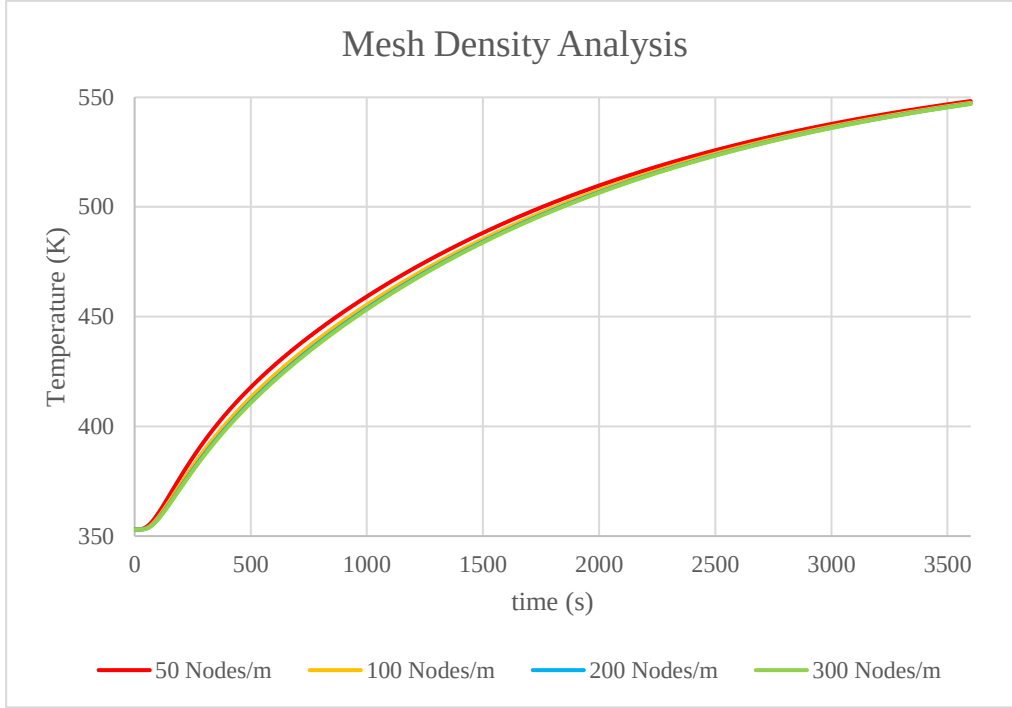


Figure 3-17 Mesh density analysis

Based on the mesh outcome, the heat transfer calculation can be launched. The boundary condition will be determined firstly.

For the compressor disc, one boundary needs to be considered, which is the gas side wall temperature. The wall temperature is determined by Eqs. (3-54).

$$T_{boundary} = T_g \quad (3-54)$$

For the combustor flame tube, two boundaries need to be considered: the hot gas side wall and the internal cooling side wall. As discussed in Section 3.3.1.1, combustor is divided into three zones: primary zone, secondary zone and dilution zone. The zone temperature is determined by the local FAR, LHV and combustor efficiency. Due to the film cooling, the wall temperature will be lower than the actual gas temperature and is determined by Eqs. (3-55). In this case, the cooling flow temperature equals to the combustor inlet temperature. For the internal cooling side wall, the wall temperature is determined by Eqs. (3-56).

$$T_{boundary1} = T_g + \eta_c * (T_c - T_g) \quad (3-55)$$

$$T_{boundary2} = T_{inlet} \quad (3-56)$$

η_c Film cooling efficiency

T_c Film cooling flow temperature

For the turbine blade, two boundaries need to be considered: the gas side wall and the internal cooling side wall. Same as the combustor, film cooling has been applied and the gas side wall temperature is determined by Eqs. (3-55). For the internal cooling side wall temperature, it is determined by Eqs. (3-57), where the cooling flow temperature is determined by the cooling flow extracted from the compressor.

$$T_{boundary2} = T_c \quad (3-57)$$

For turbine casing, two boundaries need to be considered: the gas side wall and the internal cooling side wall. The gas side wall temperature is determined by Eqs. (3-54). The cooling side wall temperature is determined by Eqs. (3-57).

For the turbine disc, two boundaries need to be considered: the gas side wall and the internal cooling side wall. The gas side wall temperature is determined by Eqs. (3-55). The cooling side wall temperature is determined by Eqs. (3-57).

After knowing the boundary condition, the heat transfer calculation can be launched. The idea of CVM is to use the energy conservation law and the Fourier's law of heat conduction to derive the difference equation. This method is general and is not limited by whether the mesh is uniformly divided or whether the material properties are constant. The fundamental equation of the CVM is shown in Eqs. (3-58). If we apply forward difference for $\frac{\partial T}{\partial t}$, then Eqs. (3-58) can be derived as Eqs. (3-59). It is evident that once the heat flow rate of node N in the i^{th} time step is known, the temperatures of node N in the $(i + 1)^{th}$ time step can be immediately calculated without solving simultaneous equations. Therefore, the computational scheme represented by Eqs. (3-59) is referred to as an explicit finite difference scheme for unsteady-state problems, where the unsteady term is a forward difference at the i^{th} time step. The advantage of the

explicit scheme lies in its lower computational workload, but it has the drawback of being sensitive to both time and space step sizes; otherwise, it may yield unreasonable results.

$$\Phi_N = Cp \cdot M_N \cdot \frac{\partial T}{\partial t} \quad (3-58)$$

$$\Phi_N^i = Cp \cdot M_N \cdot \frac{T_N^{i+1} - T_N^i}{dt} \quad (3-59)$$

Φ_N Total heat flow rate into the control volume (node N)

M_N Mass of the control volume

i Time step

Next step is to determine the heat flow rate of each node and calculate the node temperature at each time step.

A. Compressor Disc

For compressor boundary node, two types of heat flow rate are considered: one is the convective heat flow from the working fluid and the other is the conductive heat flow between the boundary node and the contact node. For the convective heat flow, the heat flow rate is determined by Eqs. (3-60), where N represents the number of the current calculating node. The heat transfer coefficient determination equations are listed in 7.8Appendix A. For the conductive heat flow, the heat flow rate is determined by Eqs. (3-61).

$$\Phi_{conv} = h_{c_d} * A_N * (T_g - T_N) \quad (3-60)$$

$$\Phi_{cond} = \frac{\lambda}{\Delta x} * A_{N,N+1} * (T_N - T_{N+1}) \quad (3-61)$$

h_{c_d} Heat transfer coefficient for compressor disc

λ Conductive coefficient

Δx Distance between boundary node and the contact node

$A_{N,N+1}$ Contacting area between boundary node and the contact node

For compressor internal node, one type of heat flow rate is considered: the conductive heat flow between nodes. For the conductive heat flow, the heat flow rate is determined by Eqs. (3-62).

$$\Phi_{cond} = \frac{\lambda}{\Delta x} * A_{N-1,N} * (T_{N-1} - T_N) - \frac{\lambda}{\Delta x} * A_{N,N+1} * (T_N - T_{N+1}) \quad (3-62)$$

B. Combustor Flame Tube

For the combustor gas side boundary node, three types of heat flow rate are considered: convective heat flow rate, radiative heat flow rate, and conductive heat flow rate. For the convective heat flow, the heat flow rate is determined by Eqs. (3-63), where h_g is the heat transfer coefficient, A_N is the boundary node wetted area, T_w is the wall temperature which calculated by using Eqs. (3-55). For the radiative heat flow, the heat flow rate is determined by Eqs. (3-64), For the conductive heat flow, the heat flow rate is estimated by using Eqs.(3-61). The emissivity of gas is calculated by using equation for luminous emissivity provided by Lefebvre [149], as shown in Eqs. (3-65). For kerosine, the luminosity factor value for primary zone, secondary zone and dilution zone are selected as 4,3 and 2.

$$\Phi_{conv} = h_{ft} * A_N * (T_w - T_N) \quad (3-63)$$

$$\Phi_{radi} = 0.5 \cdot \sigma \cdot A_N \cdot (1 + \varepsilon_w) \cdot \varepsilon_g \cdot (T_g^i)^{1.5} \cdot ((T_g^i)^{2.5} - (T_N^i)^{2.5}) \quad (3-64)$$

$$\varepsilon_g = 1 - \exp(-2.86 \times 10^4 * L * P * (r * l)^{0.5} * T_g^{-1.5}) \quad (3-65)$$

h_{ft}	Heat transfer coefficient for combustor flame tube
σ	The Stefan-Boltzmann constant
ε_w	The emissivity of the wall
ε_g	The emissivity of the gas
L	The flame luminosity factor
r	The fuel/air ratio by weight

l The radiation beam length

The combustor internal cooling side wall considers two types of heat flow rate: the convective heat flow and the conductive heat flow. The convective heat flow rate is estimated by using Eqs. (3-66) and the conductive heat flow rate is estimated by using Eqs. (3-61).

$$\Phi_{conv} = h_{cooling} * A_N * (T_c - T_N) \quad (3-66)$$

The combustor internal nodes consider the conductive heat transfer. The heat flow rate is estimated by using Eqs. (3-62).

C. Turbine Blade

For turbine blade gas side boundary node, two types of heat transfer flow will be considered: the convective heat transfer and conductive heat transfer. They are estimated by using Eqs. (3-67) and (3-61). Same as the combustor gas side boundary node, the wall temperature is estimated by using Eqs. (3-55).

$$\Phi_{conv} = h_{t_b} * A_N * (T_c - T_N) \quad (3-67)$$

h_{t_b} Heat transfer coefficient for turbine blade

For turbine internal cooling side boundary node and internal nodes, the heat flow rates are estimated by using Eqs. (3-66) and (3-61).

D. Turbine Casing

For turbine casing, the heat transfer flow estimation is as same as what discussed in combustor flame tube. The difference is that the radiative heat transfer and thermal barrier coating are not considered in turbine casing.

E. Turbine Disc

For turbine disc gas side node, three heat transfer flow will be considered: convective heat flow from the hot gas, convective heat flow from the cavity cooling flow and conductive heat flow from the contacting node. The hot gas convective flow rate is estimated by using Eqs. (3-68). The cooling flow convective heat flow

rate is estimated by using Eqs. (3-69). The conductive heat flow rate is estimated by using Eqs. (3-61).

$$\Phi_{conv} = h_{t_d} * A_N * (T_g - T_N) \quad (3-68)$$

$$\Phi_{conv} = h_{cooling} * A_{side} * (T_c - T_N) \quad (3-69)$$

h_{t_d} Heat transfer coefficient for turbine disc

For turbine internal nodes, two heat transfer flow will be considered: conductive heat flow from the contacting nodes and convective heat flow from the cavity cooling flow. The conductive heat flow is estimated by using Eqs. (3-62). The convective heat flow is estimated by using Eqs. (3-69).

F. TBC and metal contacting volume

For those components that has the thermal barrier coating, node that represent the contacting area between the TBC and metal need to be treat carefully. Figure 3-15 (e) illustrates a 1-D mesh for that includes the TBC, shown in orange, and the component material, shown in blue. The position of node j is where the TBC and blade material are in contact. It is assumed that there is perfect thermal contact between the TBC and the blade material and no temperature difference within a single control volume. The energy equation for node j and the conductive heat transfer process in node j is shown in Eqs. (3-70) and (3-71). Based on these equations, node j temperature at $i+1$ time step can be estimated.

$$\Phi_N = (Cp_{tbc} \cdot M_{tbc} + Cp_b \cdot M_b) \cdot \frac{T_j^{i+1} - T_j^i}{dt} \quad (3-70)$$

$$\Phi_{cond} = \frac{\lambda_{tbc}}{dx} \cdot A_{j-1} \cdot (T_{j-1}^i - T_j^i) - \frac{\lambda_b}{dx} \cdot A_j \cdot (T_j^i - T_{j+1}^i) \quad (3-71)$$

G. Node temperature and quantity of heat calculation

Based on the identification of heat flow rate, the temperature of each node can be estimated. The general equation for the node temperature estimation is shown in Eqs. (3-72). Based on the equation, the node temperature at $i + 1$ time step can be estimated.

$$Cp \cdot M_N \cdot \frac{T_N^{i+1} - T_N^i}{dt} = \Phi_{conv} + \Phi_{cond} + \Phi_{radi} \quad (3-72)$$

After all the nodes temperature are updated, the quantity of heat of one component during the time interval will be estimated by using Eqs. (3-73).

$$Q_{component} = \sum_{j=1}^N M_j * Cp_j * (T_j^{i+1} - T_j^i) \quad (3-73)$$

3.3.2 Establishment of Quasi-2D Thermal Network

A quasi-2D thermal network is established along the gas flow direction, with each node having the ability to estimate the temperature distribution along the radial direction. Figure 3-18 illustrates a quasi-2D thermal network established along the gas flow direction, where the red arrow represents the gas flow, and each black node represents one component stage or one combustion zone. Each node in the network estimates the temperature distribution along the radial direction and calculates heat soakage based on its own inlet and outlet condition. The calculated heat flow rate is then used to update the outlet condition, which becomes the inlet condition for the next node. This process includes three types of heat transfer: convective, conductive, and radiative heat transfer. Different heat transfer processes will be considered for different gas turbine components. The compressor blades and casing will only consider the convective heat transfer process. The compressor disc will consider convective and conductive heat transfer processes. The combustor flame tube will consider convective, conductive, and radiative heat transfer processes and the effects of film cooling and internal cooling flow. The turbine blade will consider two heat transfer processes: convective and conductive, considering film cooling, and internal cooling. The turbine disc will also consider two heat transfer processes: convective and conductive, with the consideration of cavity cooling. The turbine casing will consider two heat transfer processes: convective and conductive, considering the effect of internal cooling flow. A summary is listed in Table 3-5.

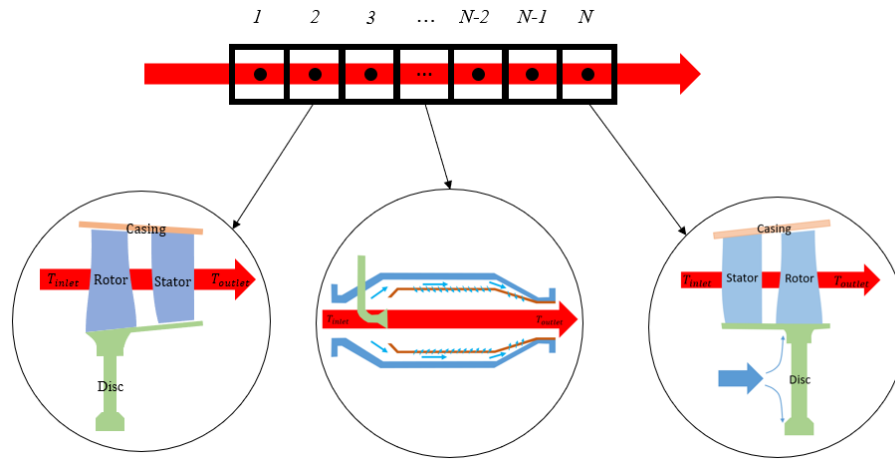


Figure 3-18 Quasi-2D thermal network illustration

Table 3-5 Summary of the quasi-2D thermal network

Component	Heat Transfer Model		TBC	Film Cooling	Internal cooling Flow
	TDM	CVM			
Compressor Blade	√	×	×	×	×
Compressor Casing	√	×	×	×	×
Compressor Disc	×	√	×	×	×
Combustor Flame Tube	×	√	√	√	√
Turbine Blade	×	√	×	√	√
Turbine Casing	×	√	×	×	√
Turbine Disc	×	√	×	×	√

3.3.3 Component Performance Modification

The estimated heat flow rate will affect the gas turbine component performance. Vieweg's method is applied during this process [153]. For the combustor, the heat flow rate leads to adding or taking from the outlet station's enthalpy, changing the outlet temperature. For the compressor and turbine, it is assumed that part of the heat is transferred before the compression or expansion, and the rest is transferred afterwards. In this way, compression and expansion can still be calculated using (adiabatic) map data. A user-defined ratio f is used to determine the part of heat that is transferred before the compression or expansion, as shown in Eqs. (3-74). Based on ratio f , the inlet temperature and outlet temperature of the compressor and turbine can be modified based on Eqs.(3-75) and (3-76) during the transient manoeuvre.

$$f = \frac{Q_{pre}}{Q_{total}} \quad (3-74)$$

$$T'_1 = T_1 + \frac{Q_{pre}}{W \cdot Cp} \quad (3-75)$$

$$T'_2 = T_2 + \frac{(1 - f)Q_{total}}{W \cdot Cp} \quad (3-76)$$

3.3.4 Component Initial Temperature Identification

During the heat soakage calculation, the initial temperature plays an important role because it will affect the entire heat transfer process. In order to have an accurate heat soakage estimation, the initial temperature distribution of the proposed quasi-2D thermal network need to be estimated accurately.

Because the initial working condition of the transient simulation in Turbomatch is based on the last off-design condition. So, the last off design condition is selected as the initial condition of the heat soakage calculation.

Instead of directly setting the temperature distribution of the quasi-2D thermal network as a constant temperature, an iterative process is launched to obtain the temperature distribution. By setting the initial value as 200K and simulate the heat transfer process for 2 hours simulation time, all components are assumed under

thermal balance condition. And the outcome of the iterative process will be used as the initial condition during the transient unsteady heat soakage calculation.

3.4 Tip Clearance

Compressor and turbine tip clearance could be affected by the component thermal deflection due to the change in temperature distribution. Beyond that, tip clearance could also change due to the centrifugal force change caused by change in rotational speed and pressure imbalance. During the gas turbine transient manoeuvre, the temperature of component is changing as well as the shaft rotational speed. This led to the change in tip clearance and further change the efficiency of compressor and turbine. Due to the change of efficiency, the transient performance of gas turbine is affected. The overall calculation logic for the tip clearance change effect is shown in Figure 3-19.

The proposed tip clearance model was divided into two parts: the tip clearance movement prediction model and the component performance variation. The tip clearance prediction includes the methodology development of the component thermal deflection and elastic deflection. The component performance variation includes the application of component efficiency change correlations.

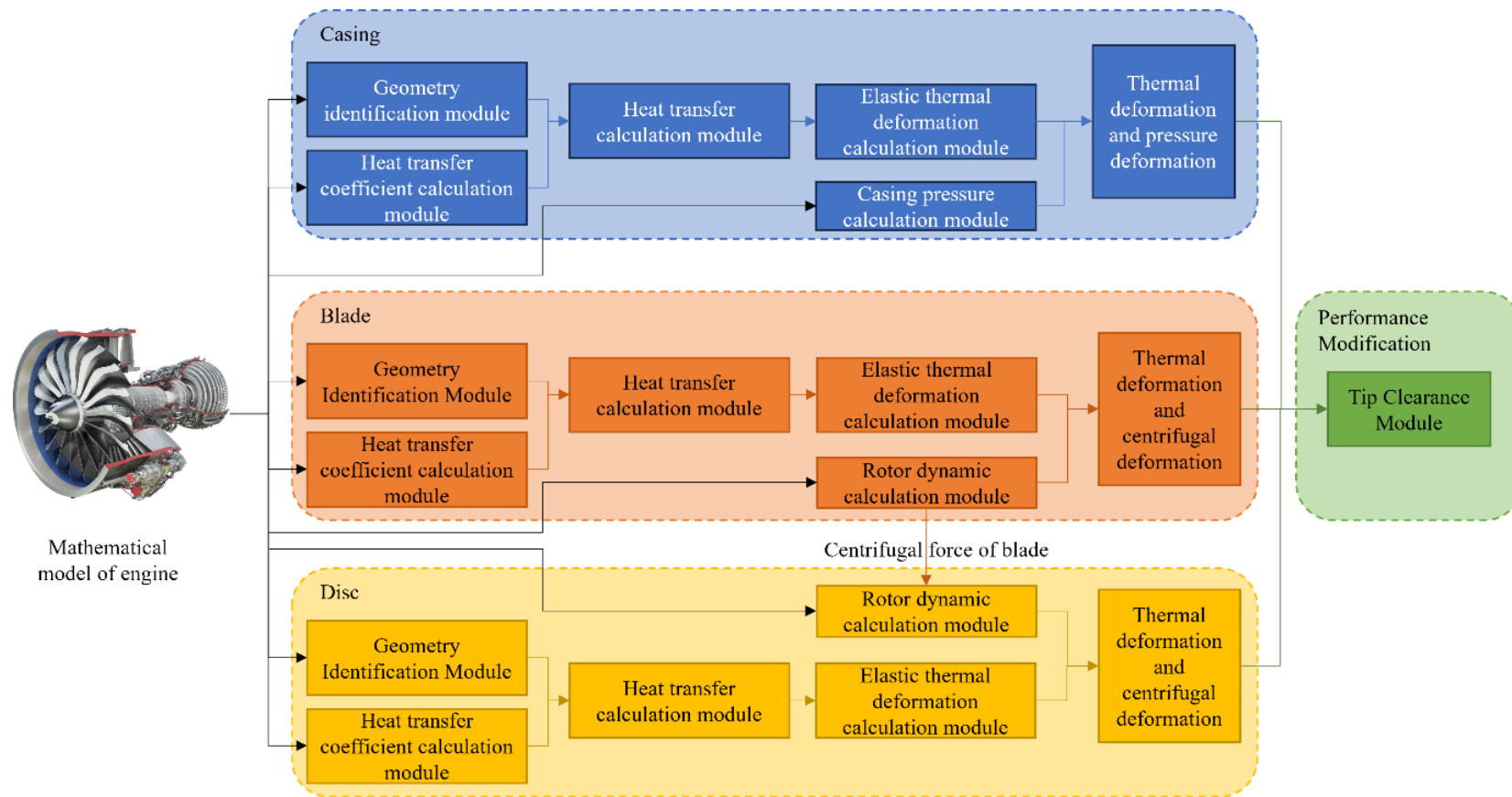


Figure 3-19 Tip clearance estimation logic during engine transient

3.4.1 Thermal and Mechanical Deflections

3.4.1.1 Blades

There are two types of blades discussed above: the compressor blade and the turbine blade. As discussed in Section 3.3.1.2, compressor blade is treated as a OD model due to its low Biot number. By using the proposed heat transfer calculation, the temperature of the blade can be easily represented by one single value. However, for the turbine blade discussed in Section 3.3.1.3, a meshing process is applied and there is a non-uniform temperature distribution along the blade wall thickness direction. By taking a mass weighted average we are able to again find a single temperature for the blade. With this single blade temperature, the thermal growth of the blade can be estimated by using Eqs. (3-77).

$$u_b = h_b * \alpha * (T_b - T_D) \quad (3-77)$$

u_b Change in blade height due to temperature change

h_b Blade height under datum conditions

T_b Temperature of the blade

T_D Temperature of the blade under datum condition

The datum condition is selected as the gas turbine design point working condition.

The centrifugal force of the blade is mainly and positively correlated to the rotational angular velocity, the mass of the blade and the length of the blade. The mechanical growth of the blade can be estimated by assuming each blade as a simple prismatic member, is constructed of an elastic material of constant density. As described in Section 3.2.1, the blade is simplified as flat plate. It is assumed that the blade is attached to the disc at its root, and is centrifugally loaded by its own weight as it rotates with the disc. Taking a thickness of dx as a microelement in the radial direction of the blade height (as shown in Figure 3-20). Integrating the microelements in blade height direction, we can obtain Eqs. (3-78) which can be used to estimate the centrifugal force at certain blade height x . By applying

the Hooke's Law, the change in blade height due to centrifugal force can be estimated by Eqs. (3-79).

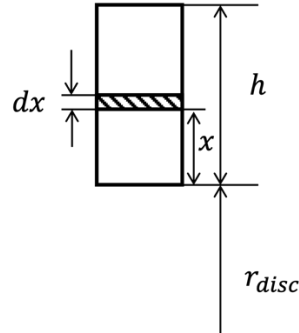


Figure 3-20 Blade centrifugal force calculation

$$F_{Nx} = \rho * \omega^2 * A * \left[r_{disc} * (h_b - x) + \frac{h_b^2 - x^2}{2} \right] \quad (3-78)$$

$$v_b = \frac{\rho * (\omega^2 - \omega_D^2)}{E} * \left(\frac{r_{disc} * h_b^2}{2} + \frac{h_b^3}{3} \right) \quad (3-79)$$

F_{Nx}	Centrifugal force at certain blade height
v_b	Change in blade height due to rotational speed change
E	Blade material Young's modulus
ω	Rotational speed

3.4.1.2 Disc

Same as the blade, disc radius will be affected by the thermal displacement and the elastic displacement.

As described above the disc temperature distribution is estimated by the 1D control volume method. To estimate the thermal displacement, a novel thermal displacement model is derived. Due to the internal thermal resistance, the temperature distribution along the radius direction is nonuniform. But the temperature distribution along the axial direction is uniform. Under this condition, we can assume that the temperature distribution is a function of radius, as shown in Eqs. (3-80).

$$T_d = f(r) \quad (3-80)$$

Using cylindrical coordinates, the stress equilibrium equation of the disk can be expressed as Eqs. (3-81), where σ_r and σ_θ can be expressed as Eqs. (3-82) and (3-83) by Hooke's law.

$$\frac{d\sigma_r}{dr} + \frac{\sigma_r - \sigma_\theta}{r} = 0 \quad (3-81)$$

$$\sigma_r = \frac{E}{(1 + \mu) * (1 - \mu)} * \frac{du}{dr} + \frac{E\mu}{(1 + \mu) * (1 - \mu)} * \frac{u}{r} - \frac{E\alpha T}{1 - \mu} \quad (3-82)$$

$$\sigma_\theta = \frac{E}{(1 + \mu) * (1 - \mu)} * \frac{u}{r} + \frac{E\mu}{(1 + \mu) * (1 - \mu)} * \frac{du}{dr} - \frac{E\alpha T}{1 - \mu} \quad (3-83)$$

σ Thermal stress

α The coefficient of thermal expansion

μ Material's Poisson's ratio

Combine Eqs. (3-81), (3-82) and (3-83), the displacement in radius direction can be expressed by Eqs. (3-84), where C_1 and C_2 are constant value based on the boundary condition.

$$u_d = (1 + \mu) * \frac{\alpha}{r} \int_{r_i}^r T_d r dr + C_1 * r + \frac{C_2}{r} \quad (3-84)$$

u_d Change in disc radius due to temperature change

r_i Inner radius of the disc

r_o Outer radius of the disc

T_d Disc temperature distribution

Since the thermal stress at $r = r_i$ and $r = r_o$ are 0, C_1 and C_2 can be expressed by Eqs. (3-85) and (3-86).

$$C_1 = (1 - \mu) * \frac{\alpha}{r_o^2 - r_i^2} \int_{r_i}^{r_o} T_d r dr \quad (3-85)$$

$$C_2 = (1 + \mu) * \frac{\alpha * r_i^2}{r_o^2 - r_i^2} \int_{r_i}^{r_o} T_d r dr \quad (3-86)$$

The radius displacement at a given radius then can be expressed as Eqs. (3-87).

$$u_d = \frac{\alpha}{r} \left[(1 + \mu) \int_{r_i}^r T_d r dr + \frac{(1 - \mu)r^2 + (1 + \mu)r_i^2}{r_o^2 - r_i^2} \int_{r_i}^{r_o} T_d r dr \right] \quad (3-87)$$

For the displacement at the outer radius, it can be expressed by Eqs. (3-88). Since the casing is meshed in radius direction, the temperature distribution is not continuous. By discretising Eqs. (3-88) we can obtain Eqs. (3-89), which is suitable for the CVM.

$$u_d = \frac{2 \cdot r_o \cdot \alpha}{r_o^2 - r_i^2} \cdot \int_{r_i}^{r_o} T_d r dr \quad (3-88)$$

$$u_d = \frac{2 \cdot r_o \cdot \alpha}{r_o^2 - r_i^2} \cdot \sum_{x=1}^N \frac{(r_{ox}^2 - r_{ix}^2) * (T_x - T_{x,D})}{2} \quad (3-89)$$

When considering disc radius displacement due to centrifugal force, two forces are considered: the centrifugal force caused by disc own weight and the centrifugal force caused by the blade weight. Here, Fang's method is applied. The displacement of disc radius is estimated by using Eqs. (3-90), where C is the constant value for the consideration of blade mass.

$$\begin{aligned} v_d = & \left(\frac{\rho R_o (3 + \mu) (r_o^2 + r_i^2)}{8} + C \frac{r_o^3}{r_o^2 - r_i^2} \right) \times \frac{(1 - \mu) \omega^2}{E} \\ & + \left(\frac{\rho (3 + \mu) (r_i^2 \times r_o)}{8} + C \frac{r_i^2 \times r_o}{r_o^2 - r_i^2} \right) \times \frac{(1 + \mu) \omega^2}{E} \\ & - \frac{\rho (1 - \mu^2) r_o^3 \omega^2}{8E} \end{aligned} \quad (3-90)$$

$$C = \frac{N\rho A \left(R_o h_b + \frac{1}{2} h_b^2 \right)}{2\pi r_o w_d} \quad (3-91)$$

v_d Change in disc radius due to rotational speed change

3.4.1.3 Casing

As described above, the heat soakage model for compressor and turbine casing is different. For the compressor casing model, the 0D heat transfer model is applied. For the turbine casing model, the 1D control volume method is applied due to the cooling flow.

For the compressor casing, the temperature can be represented by a single value. Under this condition, the thermal deflection of the casing can be estimated by Eqs. (3-92). Same as the blade, the datum condition is selected as the design point condition.

$$u_c = r * \alpha * (T_c - T_D) \quad (3-92)$$

u_c Change in casing radius due to temperature change

For the turbine casing, the temperature distribution along the radius direction is nonuniform. Under this condition, Eqs. (3-92) cannot be applied. Since the casing is meshed in radius direction, which is the same as disc, the same method described in Section 3.4.1.2 is applied to the turbine casing.

Different from the rotating parts, the only relevant mechanical loading is due to the inner and outer surface pressure difference that will lead to radius change. The change in casing radius due to pressure difference is estimated by Eqs. (3-93).

$$v_c = \frac{1}{E(r_o^2 - r_i^2)} [(1 - \nu)r_i(P_i r_i^2 - P_o r_o^2) - (1 - \nu)r_i^2 r_o^2 (P_o - P_i)] \quad (3-93)$$

v_c Change in disc radius due to pressure change

3.4.2 Component Performance Modification

3.4.2.1 Total deflection

The overall growth Δu of blade, casing and disc at one stage could be estimated by summing of their thermal growth and mechanical growth, as shown in Eqs. (3-94).

$$\Delta u_{stage} = (u_c + v_c) - (u_b + v_b) - (u_d + v_d) \quad (3-94)$$

In order to have a representative tip clearance change of the component, the root mean square tip clearance was estimated by using Eqs. (3-95).

$$\Delta u_{component} = \sqrt{\frac{\sum_{x=1}^N \Delta u_{stage}^2}{N}} \quad (3-95)$$

3.4.2.2 Efficiency Prediction

For the compressor, the empirical correlation raised by Lakshminarayana was applied, as shown in Eqs. (3-96), The modified efficiency is obtained by using Eqs. (3-97)

$$\Delta \eta = \frac{0.7 \Delta u_{component} \psi}{\cos \beta_m} \left[1 + 10 \sqrt{\frac{\phi \Delta u_{component} APR}{\psi \cos \beta_m}} \right] \quad (3-96)$$

$$\eta'_c = \eta_c - \Delta \eta \quad (3-97)$$

$\Delta \eta$ Change in compressor efficiency

β_m Mean air angle

ψ Blade loading coefficient

ϕ Flow coefficient

APR Aspect ratio of the blades.

For the turbine, the simple efficiency loss model suggested by Baskharone that utilizes relevant geometrical parameters are applied, as shown in Eqs. (3-98), where K is the value estimated by Eqs. (3-99). The modified efficiency is obtained by using Eqs. (3-100).

$$s_Eff_TC = 1 - K \left(\frac{\Delta u_{component}}{h_b} \right) \left(\frac{r_{disc}}{(r_{casing} + r_{disc})/2} \right) \quad (3-98)$$

$$K = 1 + 0.586(\psi_{Ztip}^{3.63}) \quad (3-99)$$

$$\eta'_t = \eta_t * s_Eff_TC \quad (3-100)$$

ψ_{Ztip} Zweifel loading coefficient

3.5 Compressor Map Modification

As described in Section 2.2.3, heat transfer effect will have an impact on gas turbine component performance. In the case of compressor, the change in flow temperature will lead to a change in flow density and boundary layer development. This will further affect the flow angle through the blades and change the compressor performance. Besides, the blade loss will also be affected due to the change in flow properties. In order to predict the heat transfer effect on compressor performance, the compressor 1D mean line model is selected due to its easy application and good enough accuracy. As described by its name, the mid-spans of each blade row are chosen as the calculation streamline to prediction the compressor performance based on aerodynamic parameters with a series of flow equations. In this section, the establishment of compressor mean line model, heat transfer effect on mean line model, and the map modification model will be described. The calculation logic of compressor map change effect on gas turbine transient performance is shown in Figure 3-21.

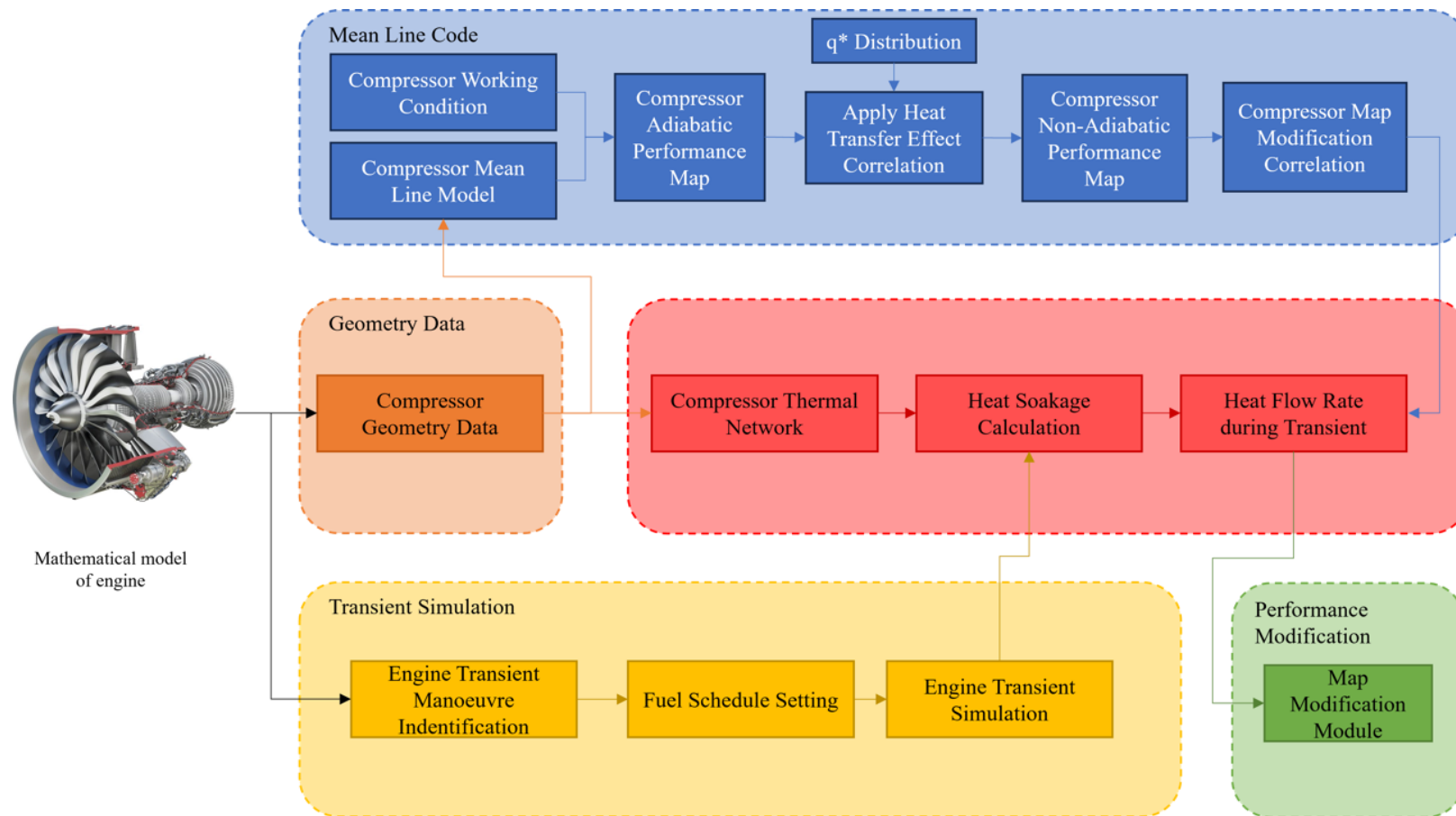


Figure 3-21 Compressor map modification model calculation logic

3.5.1 Compressor Mean Line Model Establishment

The calculation of mean line model can be divided into three steps: the input data for the mean line model, the performance calculation, and the speed line identification.

3.5.1.1 Input data for the compressor mean line model

As described above, the mean line model chooses the streamline at blade mid-spans to present the overall performance of the compressor. To create the mean line model, several data are required as the input. The most important data is the compressor flow passage geometry, as shown in Figure 3-22, where the red blocks represent the rotor blade, and the blue blocks represent the stator blade. There are two ways to establish the compressor flow passage geometry. One way is to use the geometry identification method, as described in Section 3.2.1, and the other way is to directly use the experimental data to build the compressor flow passage geometry. For the first geometry identification way, it has a lower requirement on the necessary input data, as shown in Table 3-2. However, the outcome will have a different with the real compressor geometry. Because the shape of the geometry is limited in constant tip diameter, constant hub diameter and constant mean diameter. The second way does not need to run the geometry identification simulation and the data is absolutely the same as the real compressor. However, it has an extremely high requirement on the necessary input data. In most cases, those data are classified and are not provided. Gladly, NASA provides the entire experimental data for NASA stage 35 and HPC for Energy Efficiency Engine. So, in our case, the geometry identification model was not used during the mean line model establishment and NASA experimental data are directly used as the input data of the mean line model.

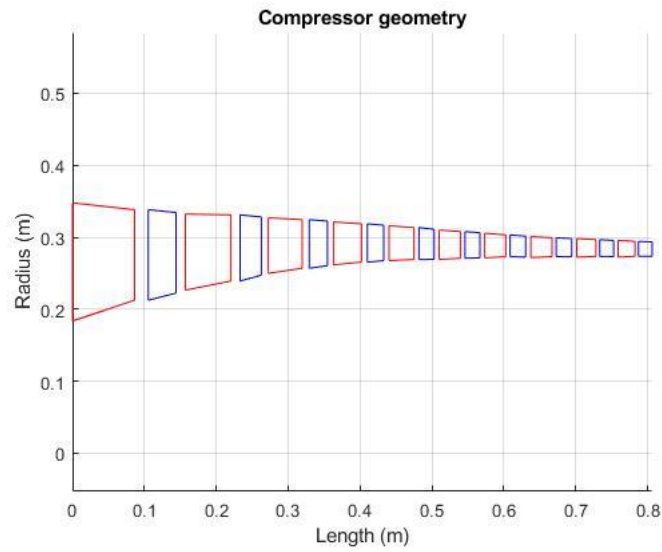


Figure 3-22 compressor flow passage geometry

The blade inlet and outlet tip and hub radius are read from the experimental data. The mean radius is calculated based on tip and hub radius by using Eqs. (3-12). For the passage area, it is calculated by using Eqs. (3-101). For the blade chord, minimum loss angle, chamber angle, stagger angle, and blade inlet and outlet angle, they are also read from the experimental data. The definition of each blade geometry parameters is shown in Figure 3-23, and the required data is listed in Table 3-6.

$$Area = \pi \cdot (r_{tip}^2 - r_{hub}^2) \quad (3-101)$$

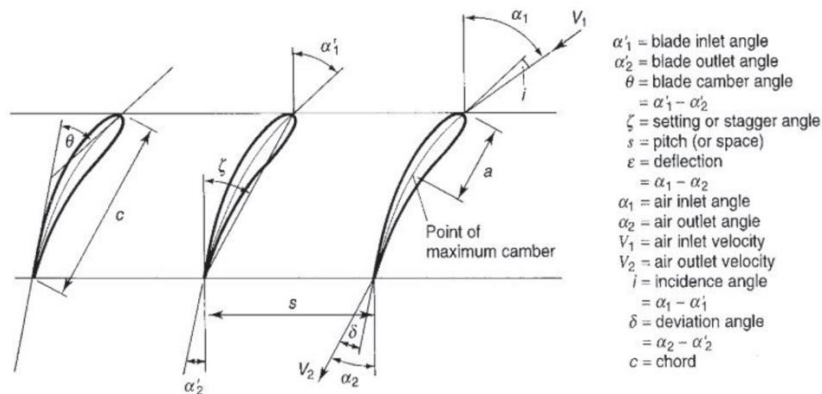


Figure 3-23 Compressor cascade notation [1]

Table 3-6 Geometry Data for Mean Line Model

Parameter	Unit
Blade inlet and outlet tip radius	m
Blade inlet and outlet hub radius	m
Blade inlet and outlet mean radius	m
Passage inlet and outlet Area	m ²
Blade Chord	m
Minimum loss incidence angle	°
Minimum loss deviation angle	°
Chamber angle	°
Stagger angle	°
Blade inlet angle	°
Blade outlet angle	°
Blade tip clearance chord ratio	N/A
Blade aspect ratio	N/A
Blade maximum thickness chord ratio	N/A
Blade Pitch Chord Ratio	N/A

Except the geometry data, the working condition data is also required for the mean line model. The working condition data define the compressor working condition and is listed in Table 3-7.

Table 3-7 Working Condition Data for Compressor Mean Line Model

Parameter	Unit
Compressor Rotational Speed	rpm

Inlet Total Pressure	Pa
Inlet Total Temperature	K
Inlet Mass Flow	kg/s
Inlet Flow Angle	°
Blockage Factor	N/A

3.5.1.2 Performance calculation

Based on the compressor geometry, the performance of the compressor can be estimated based on the mean line model. In this section, the calculation logic of velocity triangle, pressure loss model, and the deviation angle model will be discussed.

3.5.1.2.1 Calculation of Velocity Triangles

In an axial flow compressor rotor, the flow can be analyzed from two perspectives. One is the absolute or stationary frame, which is fixed to the ground, and the other is the rotating or relative frame, fixed to the rotor. The nomenclature used for compressor stage velocity triangle definition is shown in Figure 3-24. The velocities measured with respect to the rotating frame of reference are denoted by W for relative velocities, and by V for absolute velocities, measured with respect to a fixed frame of reference. The blade speed is represented by U , which is calculated by using Eqs.(3-103). The tangential component of the absolute or relative velocities is indicated by the subscript t . The absolute axial velocity is indicated by the subscript m . Subscript 1 to 3 represent the rotor inlet station, rotor-stator middle station, and stator outlet station. The velocity triangle can be depicted using the following vector relation:

$$V = W + U \quad (3-102)$$

$$U = \frac{2 \cdot \pi \cdot r_{mean} \cdot \omega}{60} \quad (3-103)$$

V	Velocity in absolute frame
W	Velocity in relative frame
U	Blade peripheral speed

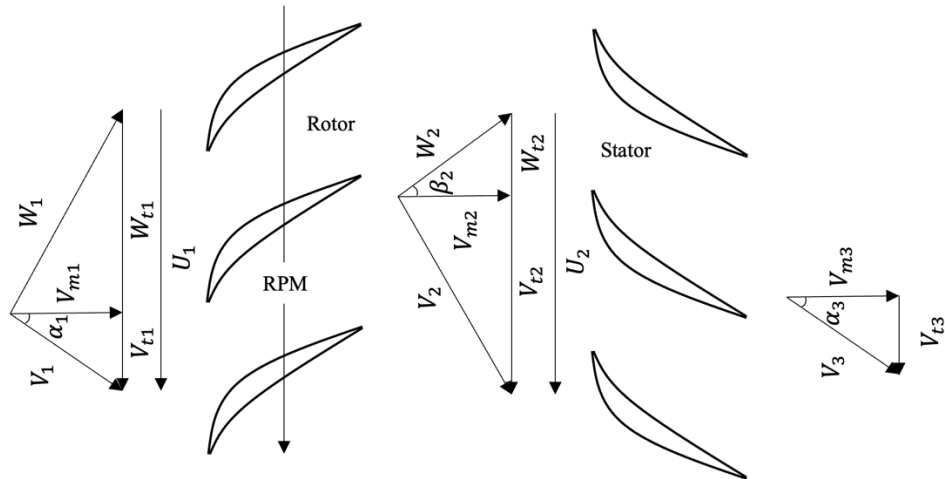


Figure 3-24 Axial compressor stage nomenclature

Based on the definition of velocity triangle, the compressor off-design performance calculation can be launched. The calculation is based on a stage-by-stage calculation. The outlet flow condition of the last stage will be the inlet condition of next stage and the process will be repeated until the last stage. For each stage, three stations will be considered: the rotor inlet condition, the rotor-stator middle condition and the stator outlet condition.

The off-design blade row inlet and exit conditions are calculated as follows:

The rotor inlet station:

At the rotor inlet condition, the know value is part of the flow absolute value (absolute total temperature and absolute total pressure). So, the flow absolute properties will be solved firstly. The inlet conditions are derived from either the stage inlet boundary conditions, or that of the preceding blade row exit conditions. Assuming there is no bleed or purge flow at the inlet station and there is no pressure loss.

If the stage inlet station has the boundary conditions, then absolute total temperature T and absolute total pressure P are defined by Eqs. (3-104) and (3-105).

$$T_{abs_1} = T_{Boundary\ Condition} \quad (3-104)$$

$$P_{abs_1} = P_{Boundary\ Condition} \quad (3-105)$$

T_{abs} Total temperature in absolute frame

P_{abs} Total pressure in absolute frame

If the stage inlet station has a preceding stage blade row, then absolute total temperature T and absolute total pressure P are defined by Eqs. (3-106) and (3-107).

$$T_{abs_1} = T_{outlet, last\ stage} \quad (3-106)$$

$$P_{abs_1} = P_{outlet, last\ stage} \quad (3-107)$$

Based on the flow absolute properties, the thermodynamic properties (C_p, γ, R) are calculated first. Then the absolute Mach number M_{abs_1} at rotor inlet station will be guessed. The static temperature and static pressure are estimated by using Eqs. (3-108) and (3-109).

$$\frac{T_{abs_1}}{t_1} = 1 + \frac{\gamma - 1}{2} M_{abs_1}^2 \quad (3-108)$$

$$\frac{P_{abs_1}}{p_1} = \left(1 + \frac{\gamma - 1}{2} M_{abs_1}^2 \right)^{\frac{\gamma}{\gamma - 1}} \quad (3-109)$$

Having calculated the static temperature and static pressure, the speed of sound is determined by Eqs. (3-110), the absolute velocity is determined by Eqs. (3-111), the static density at station 1 is determined by Eqs. (3-112).

$$a_1 = \sqrt{\gamma \cdot R \cdot t_1} \quad (3-110)$$

$$V_1 = M_{abs_1} \cdot a_1 \quad (3-111)$$

$$\rho_1 = \frac{p_1}{R \cdot t_1} \quad (3-112)$$

Since the flow inlet absolute angle α_1 and blade peripheral speed U_1 are given value, the velocity triangle can be estimated. Based on the velocity triangle, the expression and relation among V_1 , V_{m1} and V_{t1} is shown in Eqs. (3-113) to (3-116).

$$V_1^2 = V_{m1}^2 + V_{t1}^2 \quad (3-113)$$

$$V_{m1} = \frac{W_1 \cdot t_1 \cdot R_1}{BLK_1 \cdot Area_1 \cdot p_1} \quad (3-114)$$

$$V_1 = M_{abs_1} \cdot a_1 \quad (3-115)$$

$$V_{t1} = V_{m1} \cdot \tan(\alpha_1) \quad (3-116)$$

BLK Blockage factor

α_1 Flow inlet absolute angle

It can be found that V_1 , V_{m1} and V_{t1} can be expressed as a function of M_{abs_1} . The Newton-Raphson iterative scheme can be used to find the actual value of M_{abs_1} . To be more specific, the Newton-Raphson functions of $F(x)$, where $x = M_{abs_1}$ in this research context is shown in Eqs. (3-117) and (3-118).

$$F(x) = V_1^2 - V_{m1}^2 - V_{t1}^2 \quad (3-117)$$

$$F(x) = \left(\frac{W_1 \cdot T_{abs_1} \cdot R_1}{BLK_1 \cdot Area_1 \cdot P_{abs_1}} \right)^2 \cdot \left(1 + \frac{\gamma - 1}{2} x^2 \right)^{\frac{2}{\gamma - 1}} \cdot (1 + \tan(\alpha_1)^2) - \frac{\gamma \cdot R \cdot T_{abs_1} \cdot x^2}{1 + \frac{\gamma - 1}{2} x^2} \quad (3-118)$$

and its derivative is shown in Eqs. (3-119).

$$\begin{aligned}
F'(x) = & 2x \cdot (1 + \tan(\alpha_1)^2) \cdot \left(\frac{W_1 \cdot T_{abs_1} \cdot R_1}{BLK_1 \cdot Area_1 \cdot P_{abs_1}} \right)^2 \\
& \cdot \left(1 + \frac{\gamma - 1}{2} x^2 \right)^{\frac{3-\gamma}{\gamma-1}} - \gamma \cdot R \cdot T_{abs_1} \\
& \cdot \left(\frac{2x}{1 + \frac{\gamma - 1}{2} x^2} - \frac{(\gamma - 1) \cdot x^2}{\left(1 + \frac{\gamma - 1}{2} x^2 \right)^2} \right)
\end{aligned} \tag{3-119}$$

Solving the x_{n+1} with $F(x_{n+1}) = 0$ yields

$$x_{n+1} = x_n + \frac{F(x_n)}{F'(x_n)} \tag{3-120}$$

Eqs. (3-120) is applied repetitively until the following convergence criteria shown in Eqs. (3-121) is satisfied.

$$|x_{n+1} - x_n| \leq \varepsilon \tag{3-121}$$

After M_{abs_1} converged, the velocity triangle at rotor inlet station can be established. The absolute values are then converted to the relative flow values based on Eqs. (3-122) to (3-127).

$$W_{t1} = U_1 - V_{t1} \tag{3-122}$$

$$\beta_1 = \tan^{-1} \left(\frac{W_{t1}}{V_{m1}} \right) \tag{3-123}$$

$$W_1 = \sqrt{W_{t1}^2 + V_{m1}^2} \tag{3-124}$$

$$M_{rel_1} = \frac{W_1}{a_1} \tag{3-125}$$

$$T_{rel_1} = t_1 \cdot \left(1 + \frac{\gamma - 1}{2} M_{rel_1}^2 \right) \tag{3-126}$$

$$P_{rel_1} = p_1 \cdot \left(1 + \frac{\gamma - 1}{2} M_{rel_1}^2 \right)^{\frac{\gamma}{\gamma-1}} \tag{3-127}$$

β_1	Flow inlet relative angle
M_{rel}	Mach number in relative frame
T_{rel}	Temperature in relative frame
P_{rel}	Pressure in relative frame

The off-design incidence is then calculated as the difference between the relative inlet flow angle and the blade inlet angle at the leading edge based on Eqs. (3-128).

$$i = \beta_{flow} - \alpha'_{blade} \quad (3-128)$$

i	Off-design incidence angle
α'_{blade_inlet}	Blade inlet angle

The rotor-stator middle station:

A similar process is applied to the rotor-stator middle condition velocity triangle calculation. At the rotor-stator middle section, the relative outlet flow angle will be estimated firstly based on Eqs. (3-129).

$$\beta_2 = \alpha'_{blade_outlet} + \delta \quad (3-129)$$

β_2	Flow outlet relative angle
δ	Deviation angle

The next step is to determine the flow relative properties at station. Here one loop will be created to determine the specific heat at station 2 based on the relative total temperature. The initial value of relative total temperature is set as the value at station 1. Then from the conservation of rothalpy the exit relative total temperature is found by using Eqs. (3-130).

$$Cp_2 * T_{rel_2} - Cp_1 * T_{rel_1} = \frac{1}{2} U_2^2 - \frac{1}{2} U_1^2 \quad (3-130)$$

The total pressure across blade rows is not constant due to the presence of losses. For a rotating blade row, the relative total pressure is slightly reduced. For a stationary blade row this is sensed by the reduction in absolute total pressure. The reduction in total pressure is proportional to the losses generated within the blade passage. To predict the relative total pressure at rotor outlet station, Bullock and Johnse's definition of the total loss factor is applied, as shown in Eqs. (3-131) and the ideal relative pressure ratio is given by Eqs. (3-132) [154].

$$\omega_{total} = \frac{(P_{rel_2})_{ideal} - P_{rel_2}}{P_{rel_1} - P_1}$$

$$= \left(\frac{P_{rel_2}}{P_{rel_1}} \right)_{ideal} \cdot \left(\frac{1 - \frac{\frac{P_{rel_2}}{P_{rel_1}}}{\left(\frac{P_{rel_2}}{P_{rel_1}} \right)_{ideal}}}{1 - \left(\frac{1}{1 + \frac{\gamma-1}{2} M_{rel_1}^2} \right)^{\frac{\gamma}{\gamma-1}}} \right) \quad (3-131)$$

$$\left(\frac{P_{rel_2}}{P_{rel_1}} \right)_{ideal} = \left(1 + \frac{\gamma-1}{2} * \frac{(\omega * r_2)^2}{\gamma * R * T_{r1}} * \left(1 - \left(\frac{r_1}{r_2} \right)^2 \right) \right)^{\frac{\gamma}{\gamma-1}} \quad (3-132)$$

ω_{total} Total loss through blade passage

Based on the flow relative properties, the thermodynamic properties (C_p, γ, R) are calculated. Then the relative Mach number M_{rel_2} at rotor-stator middle station will be guessed. The static temperature and static pressure are estimated by using Eqs. (3-133) and (3-134).

$$\frac{T_{rel_2}}{t_2} = 1 + \frac{\gamma-1}{2} M_{rel_2}^2 \quad (3-133)$$

$$\frac{P_{rel_2}}{p_2} = \left(1 + \frac{\gamma-1}{2} M_{rel_2}^2 \right)^{\frac{\gamma}{\gamma-1}} \quad (3-134)$$

After knowing the static temperature and static pressure, the thermal properties at station 2 can be calculated by using Eqs. (3-135) to (3-137).

$$a_2 = \sqrt{\gamma_2 \cdot R_2 \cdot t_2} \quad (3-135)$$

$$V_2 = M_{rel_2} \cdot a_2 \quad (3-136)$$

$$\rho_2 = \frac{p_2}{R_2 \cdot t_2} \quad (3-137)$$

As described above, the rotor outlet relative flow angle β_2 is estimated by using Eqs. (3-129) and blade peripheral velocity U_2 can be calculated by using Eqs. (3-103), the velocity triangle at rotor outlet station can be estimated. Based on the velocity triangle, the expression and relation among W_2 , V_{m2} and W_{t2} is shown in Eqs. (3-138) and (3-141).

$$W_2^2 = V_{m2}^2 + W_{t2}^2 \quad (3-138)$$

$$V_{m2} = \frac{W_2 \cdot t_2 \cdot R_2}{BLK_2 \cdot Area_2 \cdot p_2} \quad (3-139)$$

$$W_w = M_{rel_2} \cdot a_2 \quad (3-140)$$

$$W_{t2} = V_{m2} \cdot \tan(\alpha_2) \quad (3-141)$$

The same Newton-Raphson iterative scheme can be applied to the rotor-stator middle station to find the M_{rel_2} .

After M_{rel_2} converged, the velocity triangle at rotor-stator middle station can be established. The relative values are then converted to the absolute flow values based on Eqs. (3-142) to (3-147).

$$V_{t2} = U_2 - W_{t2} \quad (3-142)$$

$$\alpha_2 = \tan^{-1}\left(\frac{V_{t2}}{V_{m2}}\right) \quad (3-143)$$

$$V_2 = \sqrt{V_{t2}^2 + V_{m2}^2} \quad (3-144)$$

$$M_{abs_2} = \frac{W_2}{a_2} \quad (3-145)$$

$$T_{abs_2} = t_2 \cdot \left(1 + \frac{\gamma - 1}{2} M_{abs_2}^2\right) \quad (3-146)$$

$$P_{abs_2} = p_2 \cdot \left(1 + \frac{\gamma - 1}{2} M_{abs_2}^2\right)^{\frac{\gamma}{\gamma - 1}} \quad (3-147)$$

The stator outlet station

At stator outlet station, only flow absolute angle, total absolute temperature and total absolute pressure are needed to be solved. The absolute flow angle will be estimated by using Eqs. (3-148).

$$\alpha_3 = \alpha_{blade_outlet} + \delta \quad (3-148)$$

For the absolute total temperature, it is assumed that there is no change through the stator blade passage. So, the absolute total temperature at stator outlet station is estimated by using Eqs. (3-149). For the absolute total pressure, it is estimated by using Eqs. (3-150).

$$T_{abs_3} = T_{abs_2} \quad (3-149)$$

$$\omega_{total} = \frac{P_{abs_2} - P_{abs_3}}{P_{abs_2} - p_2} \quad (3-150)$$

These values will be used as the inlet condition of next stage and the calculation process will be repeated until all stages are calculated.

3.5.1.2.2 Calculation of Blade Pressure Loss

Axial flow compressors are designed for optimal performance at their design point, where they operate at maximum efficiencies. At this point, the discrepancy between the blade angle and the inlet flow angle is termed the minimum loss incidence angle. However, in reality, axial flow compressors rarely operate precisely at the design point, and several kinds of losses, including profile losses,

annulus losses, secondary losses and etc., will reduce the pressure generated by the blade. A summary of pressure losses is shown in Figure 3-25. Therefore, correction factors for off-design conditions need to be applied in predicting profile pressure losses. To estimate the total pressure loss in an axial flow compressor, a combination of theoretically separable loss components is utilized through superposition.

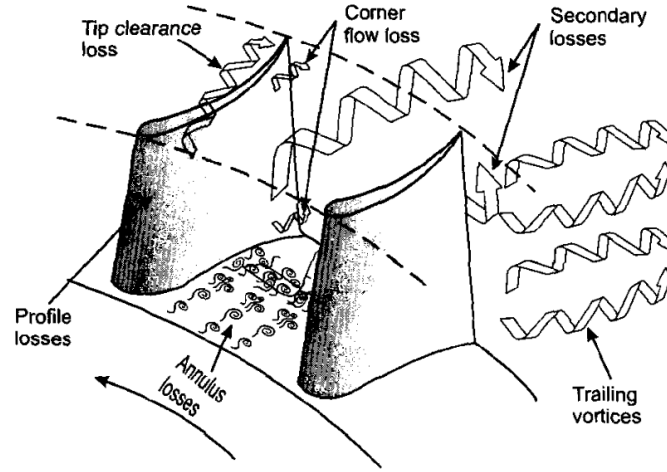


Figure 3-25 Losses obtained in an axial flow compressor [155]

The total pressure loss is estimated based on van Antwerpen's total loss model [155], as shown in Eqs. (3-151), where ω_p^* is the profile pressure loss coefficient at minimum loss angle, $\left(\frac{\omega}{\omega_i}\right)_{inc}$ is the off-design correction due to change in incidence angle, $\left(\frac{\omega}{\omega_i}\right)_{Re}$ is the off-design correction due to change in Reynolds number, $\left(\frac{\omega}{\omega_i}\right)_{Ma}$ is the off-design correction due to change in Mach number, ω_s is the secondary loss which also referred as the end wall loss and ω_a is annulus loss. They will be discussed separately in this section.

$$\omega_{total} = \omega_p^* * \left(\frac{\omega}{\omega_i}\right)_{inc} * \left(\frac{\omega}{\omega_i}\right)_{Re} * \left(\frac{\omega}{\omega_i}\right)_{Ma} + \omega_s * \left(\frac{\omega}{\omega_i}\right)_{Re} + \omega_a \quad (3-151)$$

Profile pressure loss coefficient at minimum loss angle

The profile pressure loss is caused by the development of boundary layers and wake flow in the Mid-Span apart from the end wall region of the compressor.

Before estimating the profile pressure loss coefficient at minimum loss angle, the diffusion ratio and diffusion factor need to be estimated. There are several methods on how to calculate the diffusion ratio as well for the diffusion factor. A simplified method on how to establish the diffusion ratio and the diffusion factor is shown in Eqs. (3-152) and (3-153), where.

$$\text{Diffusion Ratio} = \frac{W_{max}}{W_2} \quad (3-152)$$

$$\text{Diffusion Factor} = \frac{W_{max} - W_2}{W_1} = 1 - \frac{W_2}{W_1} + \frac{|\Delta W_t|}{2 \cdot \sigma \cdot W_1} \quad (3-153)$$

W_{max} Maximum relative velocity occurring on the blade suction surface, defined as $\Delta W_t = W_{t1} - W_{t2}$

σ blade solidity

Through the years, different authors optimized the diffusion ratio equations. Here, three diffusion ratio model will be discussed.

Diffusion Ratio Model I

Lieblein derived an equivalent diffusion ratio with minimum loss incidence as reference. The model is obtained in terms of inlet and outlet conditions and is given by Eqs. (3-154) [156],

$$D_{eq}^* = \frac{\cos(\beta_2)}{\cos(\beta_1)} \cdot \left[k_1 + k_2 \cdot \frac{\cos^2(\beta_2)}{\sigma} \cdot (\tan(\beta_1) - \tan(\beta_2)) \right] \quad (3-154)$$

k_i Constants determined from tests, $k_1 = 1.12$ and $k_2 = 0.61$.

Diffusion Ratio Model II

Koch and Smith optimized Lieblein's model to account for compressibility, Reynolds number effects and stream tube contraction effects found in axial flow compressors. Koch and Smith defined a semi-empirical approximation for the trailing-edge diffusion ratio with minimum loss incidence as reference and is given by Eqs. (3-155) [157],.

$$D_{eq}^* = \frac{W_1}{W_{te}} \cdot \frac{W_{max}}{W_{ps}} \cdot \frac{W_{ps}}{W_1} \quad (3-155)$$

W_{te} Free stream trailing-edge velocity, which equals to the blade outlet relative velocity W_2

W_{ps} Mean velocity in the blade passage throat region

The correlations between the different velocity ratios obtained in Eqs. (3-155) are estimated by Eqs. (3-156) and (3-157).

$$\frac{W_{max}}{W_{ps}} = \left(1 + 0.7688 \cdot \frac{t_{max}}{c} + 0.6024 \cdot \Gamma \right) \quad (3-156)$$

$$\frac{W_{ps}}{W_1} = \left[(\sin(\beta_1) - 0.2455 \cdot \sigma \cdot \Gamma^2) + \left(\frac{\cos(\beta_1)}{A^* \cdot \frac{\rho_{ps}}{\rho_1}} \right)^2 \right]^{0.5} \quad (3-157)$$

$\frac{t_{max}}{c}$ Ratio of maximum tip clearance and chord

Γ Minimum loss circulation parameter

A^* Minimum loss area contraction ratio

$\frac{\rho_{ps}}{\rho_1}$ Density ratio between the passage throat to entry

The minimum loss area contraction ratio from blade inlet to the throat is given by Eqs. (3-158), where $AVDR$ is defined by Eqs. (3-159).

$$A^* = \left(1 - \frac{0.4458 \cdot \sigma \cdot \frac{t_{max}}{c}}{\frac{\cos(\beta_1 + \beta_2)}{2}} \right) \cdot \frac{\left(2 + \frac{1}{AVDR} \right)}{3} \quad (3-158)$$

$$AVDR = \frac{\rho_2 \cdot V_{m2}}{\rho_1 \cdot V_{m1}} \quad (3-159)$$

The minimum loss circulation parameter for a two-dimensional incompressible cascade is defined by Eqs. (3-160).

$$\Gamma = \frac{r_1 \cdot W_{t1} - r_2 \cdot W_{t2}}{W_1 \cdot \sigma \cdot \frac{(r_1 + r_2)}{2}} \quad (3-160)$$

The density ratio between the passage throat to entry is defined by Eqs. (3-161).

$$\frac{\rho_{ps}}{\rho_1} = 1 - \left(\frac{M_{m1}^2}{1 - M_{m1}^2} \right) \cdot (1 - A^* - 0.2455 \cdot \sigma \cdot \Gamma \cdot \sin(\beta_1)) \quad (3-161)$$

Diffusion Ratio Model III

Wright and Miller modified the original diffusion ratio model to account for compressibility and thickness to chord ratio effects observed in high-speed cascades and compressor tests. This correlation has been established for transonic DCA aerofoils but is used for conventional circular arc aerofoils. The equivalent diffusion ratio with reference to minimum loss incidence is given by: (3-162) [158].

$$D_{eq}^* = \left(1 - \frac{W_2}{W_1} + \left(0.1 + \frac{t_{max}}{c} \cdot \left(10.116 - 34.15 \cdot \frac{t_{max}}{c} \right) \right) \cdot \frac{s}{c} \right. \\ \left. \cdot \frac{W_{t1} - W_{t2}}{W_1} \right) \cdot \frac{W_1}{W_2} + 1 \quad (3-162)$$

$\frac{s}{c}$ Ratio of blade spacing and chord

After obtaining the equivalent diffusion ratio, the profile loss is estimated by using a modified version of the two-dimensional low speed correlation of Lieblein, as shown in Figure 3-26 [158]. This correlation has been established for DCA aerofoils but is also used for conventional circular arc aerofoils.

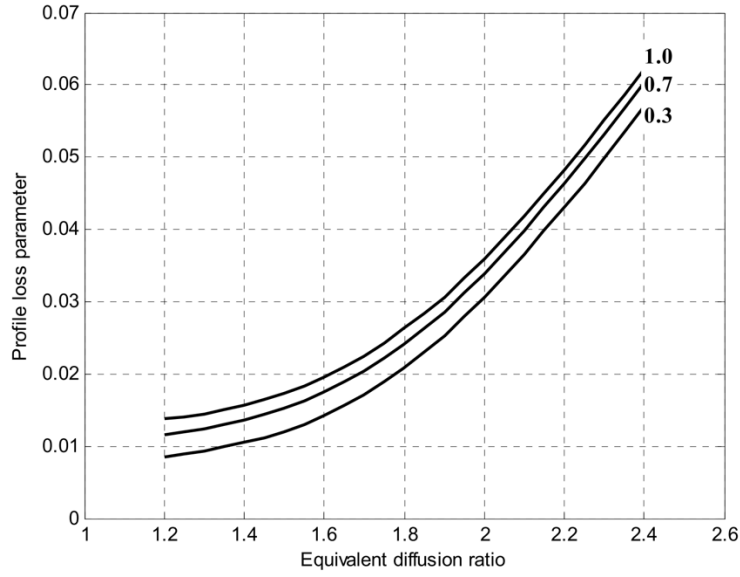


Figure 3-26 Profile Loss Parameter with Variation in Mach number [159]

The profile-loss parameter is expressed as Eqs. (3-163).

$$\omega_p \cdot 0.5 \cdot \frac{W_2^2}{W_1^2} \cdot \cos(\beta_2) = f(M_{rel1}, D_{eq}^*) \quad (3-163)$$

Off-design correction due to change in incidence angle

The discussion above is related to the profile pressure loss coefficient at design point. However, compressor will not only work at design point. The profile pressure loss coefficient needs to be modified during the off-design simulation. The change in incidence angle will change the development of boundary development and further affect the profile pressure loss coefficient. In this study, Miller and Wasdell's incidence correction factor is selected [155]. The off-design incidence correction is given by Eqs. (3-164).

$$\left(\frac{\omega}{\omega_i}\right)_{inc} = \begin{cases} \left(\frac{i - i^*}{i_{st} - i^*}\right)^2 + 1 & \text{for } i > i^* \\ 2.25 * \left(\frac{i - i^*}{i^* - i_{ch}}\right)^2 + 1 & \text{for } i < i^* \end{cases} \quad (3-164)$$

i Off-design incidence angle

i^* Design point incidence angle

i_{st} Stall incidence angle

i_{ch} Choke incidence angle

off-design correction due to change in Reynolds number

As discussed above, the profile loss is caused by the development of boundary layer over the blade surfaces. The magnitude of boundary layer development effect depending on the Reynolds number, the turbulence level and the pressure gradients. When Reynolds number is greater than 5×10^5 , the flow is considered as turbulent and has only a small impact on the overall blade profile loss. However, as Reynolds number falls and the boundary layer is gradually transforming from turbulent flow to laminar flow, flow separation will take place and the losses will be much higher that they would have been under turbulent conditions.

In order to include the effects of Reynolds number in profile losses, one approach for is to correct the profile loss coefficient by multiplying it with a Reynolds-effects correction factor. In this research, the Reynolds number correction from Wright and Miller is selected [158]. The correction for the influence of Reynolds number assumes that the flow is hydraulically smooth up to a Reynolds number of 10^6 . The off-design Reynolds number correction is shown in Eqs. (3-165).

$$\left(\frac{\omega}{\omega_i}\right)_{Re} = \begin{cases} 489.8 * Re^{-0.5} & \text{for } Re < 10^5 \\ 13.8 * Re^{-0.19} & \text{for } 10^5 < Re < 10^6 \\ 1 & \text{for } 10^6 < Re \end{cases} \quad (3-165)$$

off-design correction due to change in Mach number

Stage inlet Mach number will also affect the profile loss. When the Mach number reaches unity locally in an airfoil cascade, the corresponding inlet Mach number is said to have reached its critical value. Below the critical Mach number, the total pressure losses and turning angles are essentially constant. However, when the Mach number exceed the critical Mach number, the pressure losses increases.

In order to include the effects of change in Mach number in profile losses, Wright and Miller's method is taken. The correlation relates blade profile loss at reference conditions to equivalent diffusion ratio and inlet Mach number is shown in Figure 3-26. By reading the map, the profile-loss parameter in Eqs. (3-163) already include the effect from change in Mach number.

Secondary Pressure Loss Coefficient

The profile pressure loss is caused by the boundary layer development in the mainstream direction. For the flow that moves at right angles with respect to the direction of the primary flow, it is usually defined as secondary flow and is directly dependent on flow vorticity and circulation and also in the flow static pressure field. For the secondary pressure loss, it is caused by a complex three-dimensional secondary flow as a result of the end wall boundary layers passing through the cascade. Although the compressor performance model established in this research is a 1D mean line model. The secondary loss is still taken into account.

In this study, Bloch and O'Brien's method is obtained to estimate the secondary pressure loss through the blade passage [155]. The equation is shown in Eqs. (3-166), where β_m is defined by Eqs. (3-167).

$$\omega_s = \frac{0.72}{\sigma} * \frac{\cos^2 \beta_1}{\cos \beta_m} * (\tan \beta_1 - \tan \beta_2)^2 \quad (3-166)$$

$$\tan \beta_m = \frac{\tan \beta_1 + \tan \beta_2}{2} \quad (3-167)$$

Annulus Pressure Loss Coefficient

Annulus pressure loss refers to the additional drag effects caused by the walls of the compressor passage.

In this study, Bloch and O'Brien's method is obtained to estimate the annulus pressure loss through the blade passage. The equation is shown in Eqs. (3-168) [155].

$$\omega_a = 0.02 * \sigma * \left(\frac{c}{h}\right) * \frac{\cos^2 \beta_1}{\cos^3 \beta_m} \quad (3-168)$$

3.5.1.2.3 Calculation of Deviation Angle

Blade deviation angle is an important parameter during accurate compressor off-design performance simulation. The deviation angle is defined as the difference between the outlet flow angle and the blade outlet angle. Using the rotor blade as an example, the deviation angle is given by Eqs. (3-169).

$$\delta = \beta_2 - \alpha'_{blade_outlet} \quad (3-169)$$

The deviation angle is caused by the potential flow effect. Due to the Kutta-Joukowski boundary condition with zero load at the trailing edge of the blade, the air passing through the blade flow path cannot follow the blade camber direction up to the passage outlet. The blade and cascade geometrical characteristics including cascade solidity, blade camber angle and blade stagger angle will affect the deviation angle. Because they determine the chord-wise position where the flow direction starts to deviate from the tangential to the blade camber line. The boundary layer development does not have a huge impact on the deviation angle. The axial velocity ratio has been proven to influence the amount of flow deviation.

In order to calculate the off-design deviation angle, the minimum-loss deviation angle needs to be established first. Since the geometry data is read from the experimental report directly, the minimum-loss deviation angles for each blade row are calculated directly by using Eqs. (3-169) based on the design point flow angle and blade outlet angle.

When calculate the off-design deviation angle, Creveling's method is used[160]. The equation for deviation angle calculation is shown in Eqs. (3-170). ε^* in Eqs. (3-170) is the flow deflection angle calculated by Eqs. (3-171).

$$\delta = \delta^* + \left(\frac{\delta - \delta^*}{\varepsilon^*} \right) * \varepsilon^* \quad (3-170)$$

$$\varepsilon^* = \theta - \delta^* + i^* \quad (3-171)$$

δ Off-design deviation angle,

δ^* Minimum-loss deviation angle

ε^* Flow deflection angle

θ Blade camber angle

Creveling defined a parameter P_1 , which is the ratio of the incidence angle increment to the deflection angle, by Eqs. (3-172). Similarly, a parameter P_2 is defined by Eqs. (3-173).

$$P_1 = \frac{i - i^*}{\varepsilon^*} \quad (3-172)$$

$$P_2 = \frac{\delta - \delta^*}{\varepsilon^*} \quad (3-173)$$

Creveling and Carmody raised the correlation between P_1 and P_2 through experimental data and the correlation is shown in Eqs. (3-174). When P_1 is positive or equal to zero then $a = -0.2928$, $b = 0.5588$, $c = -0.0008090$. If P_1 is negative then $a = -0.3452$, $b = 0.48$, $c = 0.0001191$.

$$P_2 = a * P_1^2 + b * P_1 + c \quad (3-174)$$

With all the parameters defined above, the off-design deviation angle can be calculated by using Eqs. (3-175) and the air outlet angle can be calculated by using Eqs. (3-169).

$$\delta = \delta^* + P_2 * \varepsilon^* \quad (3-175)$$

3.5.1.3 Speed line Identification

The above discussion represents the calculation logic for a single off-design point. To create the compressor speed line, a series of performance point under one rotational speed with change in mass flow need to be determined. During the change in mass flow, two ending point will reach, one is the choking point and the other is the stalling point.

The choking point represents the compressor working point where the mass flow cannot be further increased and the conditions in the inlet region and upstream remain unaffected by changes in the downstream pressure. If we reduce the downstream pressure, the flow in the diverging part of the blade passage will be accelerated to higher supersonic velocities and lead to strong compressor pressure ratio. The stall point represents the compressor working point where the mass flow is too small for the compressor. At this point, the reduce in mass flow led to an increase in incidence angle and the streamline on the blade suction side cannot flow through the blade surface after the boundary layer separation point. If the mass flow keeps reducing and the air pressure downstream of the compressor does not fall quickly enough, the air flow will reverse and casing surge.

In this study, the choke point is determined by two ways. The first is to use the choke incidence angle. Herrig's correlations with a modification of Mach number shown in Eqs. (3-176) and (3-177) is used for the choke incidence determination [161].

$$R_{choke} = 1.5 \cdot \left(10 - \frac{\theta \cdot (\gamma - 40)}{450} \right) \cdot \left(0.5 + 5 \cdot \frac{t_{max}}{c} \right) \quad (3-176)$$

$$i_{choke} = i^* - R_{choke} / (1 + 0.5 \cdot M^3) \quad (3-177)$$

The second way to determine the choke point is based on the convergence of the mean line code. When mean line code cannot converge, it is also assumed that compressor reach its choking condition.

Stall point is also determined by two ways. Herrig's correlations for stall point is used as the first determination way [161]. The correlations are shown in (3-178) and (3-179), where the constant K_{sh} is the shape correction factor accounting for the particular blade profiles. K_{sh} is equals to 0.7 for MCA blade profile, 1 for NACA 65 series blade profile, and 1.1 for C4 series blade profile.

$$R_{stall} = 1.5 \cdot \left(10 + \frac{\theta \cdot (55 - \gamma)}{150} \right) \cdot \left(0.5 + 5 \cdot \frac{t_{max}}{c} \right) \quad (3-178)$$

$$i_{stall} = i^* + R_{stall} / (1 + 0.5 \cdot (K_{sh} * M)^3) \quad (3-179)$$

The second way is assuming the point where the slope of the speed line equals to 0 as the surge point.

3.5.2 Heat Transfer Effect on Mean Line Model

As described in Chapter 2, the heat transfer between the blade and working fluid will affect the compressor performance. In this study, the numerical correlation raised by Shah is applied to the proposed mean line model to simulate the heat transfer effect on compressor performance.

The non-dimensional heat transfer rate is defined first as the heat transfer rate normalized by the inlet total enthalpy, as shown in Eqs. (3-180) [117]. The heat transfer direction is from the working fluid to the compressor metal. When working fluid is cooled (the fluid temperature is higher than the metal temperature), the value of q^* will be lower than 0. When working fluid is heated (the fluid temperature is lower than the metal temperature), the value of q^* will be higher than 0. Typical values of q^* for one compressor stage in Shah's computations were on the order of -0.0025.

$$q^* = \frac{\Phi}{W * H_{t,i}} = \frac{\Delta H}{H_{in}} = \frac{\Delta T}{T_{in}} \quad (3-180)$$

H Enthalpy

Correlation for the change in deviation angle caused by heat transfer effect that raised by Shah is shown in Eqs. (3-181) [117]. The value of ζ for an arbitrary relative Mach number is found using linear interpolation.

$$\Delta\delta = \frac{q^*}{0.01} \zeta \begin{cases} \zeta = 0.1^\circ & \text{for } M_{rel,i} = 0.4 \\ \zeta = 0.25^\circ & \text{for } M_{rel,i} = 0.8 \end{cases} \quad (3-181)$$

$\Delta\delta$ Change in off-design deviation angle due to heat transfer effect

Correlation for the change in blade pressure loss caused by heat transfer effect that raised by Shah is shown in Eqs. (3-182) [117].

$$\Delta\omega = \frac{1}{1 - \left(1 + \frac{\gamma-1}{2} M_{rel,i}^2\right)^{\frac{\gamma}{\gamma-1}}} \frac{\gamma M_{rel,i}^2}{2} q^* \quad (3-182)$$

$\Delta\omega$ Change in off-design pressure loss coefficient due to heat transfer effect

These two correlations are added to the deviation angle model and total pressure loss model discussed above.

3.5.3 Map Modification Model

In order to modify the component map during the gas turbine transient simulation, the idea of scaling factor was adopted. The definitions of scaling factors are represented by Eqs. (3-183) and (3-184), which are the ratio between modified value and original value. The scaling factor is defined as a function of corrected rotational speed and non-dimensional heat transfer rate, as shown in Eqs. (3-185). When knowing the original parameter and the scaling factor, the new parameter can be calculated.

$$SF_{PR} = \frac{PR_{new}}{PR_{old}} \quad (3-183)$$

$$SF_{MF} = \frac{MF_{new}}{MF_{old}} \quad (3-184)$$

$$SF = f(CN, q^*) \quad (3-185)$$

SF Scaling factor for compressor map

3.6 Model Implementation to Turbomatch

The methods outlined above were incorporated into a set of FORTRAN subroutines so that they could be easily integrated with the FORTRAN source code for the Cranfield gas turbine simulation platform Turbomatch. Detailed information is listed in 7.8Appendix C.

4 Validation

This chapter discusses the validation of the proposed heat soakage model, tip clearance model, and mean line model. Heat transfer effect is an impact on the gas turbine transient performance. In order to validate the proposed heat transfer model, it is necessary to validate the adiabatic transient performance of Turbomatch at first place to ensure the baseline of heat transfer model. So, the steady-state and transient performance of Turbomatch is firstly compared with experimental data and GasTurb.

4.1 Turbomatch off-design performance validation

In this Section, the validation of Turbomatch off-design performance is discussed. The Energy Efficient Engine (E3) was selected for validation due to the comprehensive experimental data provided by NASA. The cross-sectional view of the E3 engine is shown in Figure 4-1. As shown in the Figure 4-1, the E3 engine is a two-spool mixed-flow turbofan. The corresponding Turbomatch model is presented Figure 4-2.

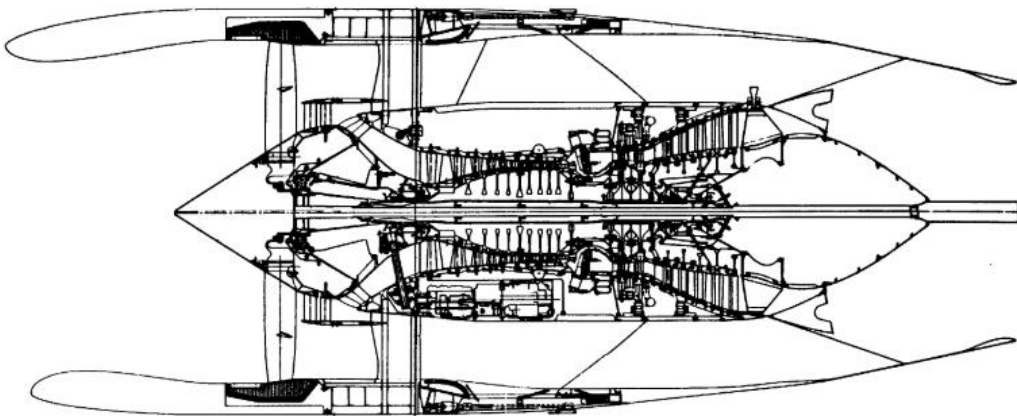


Figure 4-1 Energy Efficiency Engine Configuration

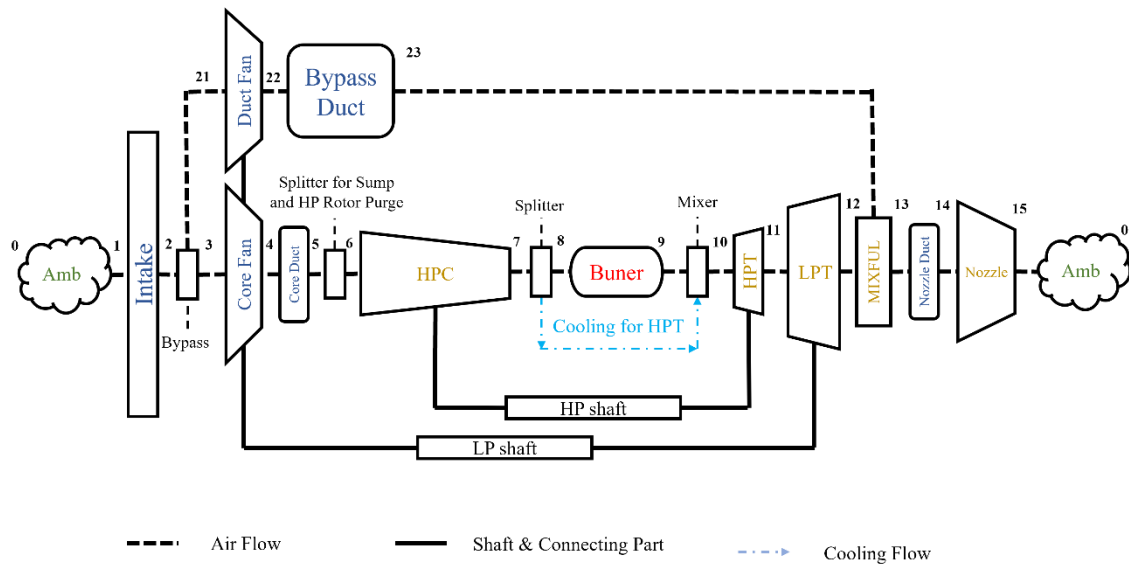


Figure 4-2 Energy efficiency engine Turbomatch model

The first step is the comparison with the real engine performance at the design point condition. The parameters for each engine components are directly obtained from NASA E3 engine report, as shown in Table 4-1. The criteria used during design point calculation is to set the combustor outlet temperature to match the simulated thrust with the experiment value.

Table 4-1 Energy efficiency engine parameters

Parameters	Value
Design Point	Maximum Climb
Altitude (m)	10668
Flight Mach Number	0.8
Ambient Temperature (K)	standard day +10K
Bypass Ratio	6.7
Overall Pressure Ratio	38.4
Corrected Mass Flow (kg/s)	646
Fan - Duct - Pressure Ratio	1.68

Fan - Core - Pressure Ratio	1.7
Fan - Duct - Efficiency (%)	88.8
Fan - Core - Efficiency (%)	90.5
Fan - Duct - Pressure Loss based on Maximum Cruise Condition(%)	1.7
Fan - Core - Pressure Loss based on Maximum Cruise Condition (%)	1.4
Compressor Inlet Temperature (K)	304.8
Compressor Inlet Pressure (N/m ² or kPa)	60469
Compressor Corrected Inlet Mass Flow (kg/s)	54.4
Compressor - Pressure Ratio	23.1
Compressor – Isentropic Efficiency (%)	0.86
CPD Nonchargeable Cooling (%W25)	16.15
Combustor Efficiency (%)	99.9
Combustor Pressure Loss (%)	5
HPT Efficiency (%)	92.7
LPT Efficiency (%)	92.7

The parameters chosen for design point validation are thrust, thrust specific fuel consumption (TSFC), compressor inlet total temperature, compressor inlet total pressure, high-pressure turbine (HPT) inlet total temperature, and HPT inlet total pressure. Table 4-2 presents a comparison between the actual design data from the NASA report and the results obtained with Turbomatch, along with the relative percentage errors. The differences between the simulation results and the NASA data are generally small. The largest discrepancy occurs at the HPT inlet temperature, with a difference of -1.82%. This may be due to the missing data for

fan duct pressure loss and fan core pressure loss at the design point condition, as the NASA report only provides these values for maximum cruise conditions. The difference in pressure loss could affect the thrust and, in turn, cause a variation in the high-pressure turbine (HPT) inlet total temperature.

Table 4-2 Design point performance comparison

	NASA	TM File	Difference
Net Thrust (Uninstalled) (kN)	40.211	40.211	0%
TSFC (Uninstalled) (g/kN/s)	15.0278	15.08357	0.37%
Compressor Inlet Temperature (K)	304.8	304.89	0.03%
Compressor Inlet Pressure (kPa)	60469	60420.1	-0.08%
Compressor Corrected Inlet Mass Flow (kg/s)	54.4	54.30	-0.18%
HPT Inlet Temperature (K)	1517.15	1489.43	-1.82%
HPT Exit Temperature (K)	1056.3	1051.75	0.43%

After defining the engine design point it was necessary to ensure that this model, with parameters defined in Table 4-1, works well in off-design conditions. Off-design study needs compressor and turbine component maps. Standard compressor and turbine maps available in the Turbomatch database are used. They are generic maps but can be used, as a first approximation, for most of the turbomachinery.

Two off-design conditions are selected to validate the proposed model: maximum cruise and takeoff conditions. During the off-design simulation, the high-pressure spool corrected rotational speed is chosen as the control parameter (referred to as a 'user-defined' parameter). The ambient conditions are manually set to match the engine's actual operating conditions. The remaining simulation data are automatically calculated by Turbomatch, allowing for a direct comparison

between the Turbomatch results and NASA data. The comparison between the Turbomatch simulation outcome and NASA data are listed in Table 4-3 and Table 4-4. Based on the comparison results, the error between the NASA data and the Turbomatch results does not exceed 3.6% in both maximum cruise and takeoff conditions. Considering that Turbomatch is a 0D simulation tool, this level of discrepancy is acceptable. Therefore, the Turbomatch steady-state calculations are validated.

Table 4-3 Off design (Maximum Cruise) performance comparison

	NASA	Turbomatch	Difference (%)
Working Point	Maximum Cruise		
Altitude (m)	10668	10668	0
Flight Speed (Mach)	0.8	0.8	0
Ambient Temperature (C)	ISA +10C	ISA + 10	0
Net Thrust (Uninstalled) (kN)	37.476	37.51	0.0928
TSFC (Uninstalled) (kg/N/hr)	15	14.99	-0.0719
Bypass Ratio	6.8	6.84	0.5882
Overall Pressure Ratio	36.5	36.42	-0.2109
LP Spool Corrected Speed (%)	97.1	97.79	0.7085
HP Spool Corrected Speed (%)	99.5	99.5	0
Mass Flow (kg/s)	243.6768	240.30	-1.387
Fan - Duct - Pressure Ratio	1.64	1.637	-0.1829
Fan - Core - Pressure Ratio	1.66	1.664	0.2410
Fan - Duct - Efficiency (%)	89.4	88.936	-0.5190
Fan - Core - Efficiency (%)	90.6	90.786	0.2053

Compressor Inlet Temperature (K)	310	302.73	-2.3452
Compressor - Pressure Ratio	22.4	22.381	-0.0848
Compressor - Efficiency (%)	86.1	86.228	0.1487
HPT Inlet Temperature (C)	1485.15	1454.1	-2.0907
HPT Efficiency (%)	92.7	92.67	-0.0324
LPT Efficiency (%)	92.5	92.988	0.5276

Table 4-4 Off design (Take Off) performance comparison

	NASA	Turbomatch	Difference (%)
Working Point	Take Off		
Altitude (m)	0	0	0
Flight Speed (Mach)	0	0	0
Ambient Temperature (C)	ISA +15C	ISA + 15	0
Net Thrust (Uninstalled) (kN)	162.36	164.87	1.5459
TSFC (Uninstalled) (kg/N/hr)	8.472	8.172	-3.5423
Bypass Ratio	7.3	7.153	-2.0096
Overall Pressure Ratio	32.4	31.442	-2.9556
LP Spool Corrected Speed (%)	88.7	89.06	0.4002
HP Spool Corrected Speed (%)	97.7	97.7	0
Mass Flow (kg/s)	563.42	559.68	-0.6633
Fan - Duct - Pressure Ratio	1.56	1.526	-2.1795
Fan - Core - Pressure Ratio	1.58	1.557	-1.4557

Fan - Duct - Efficiency (%)	89.1	90.14	1.1717
Fan - Core - Efficiency (%)	90.9	91.23	0.3630
Compressor Inlet Temperature (K)	349.5	347.86	-0.4692
Compressor - Pressure Ratio	20.8	19.995	-3.8701
Compressor - Efficiency (%)	86.3	86.28	-0.0231
HPT Inlet Temperature (C)	1616.15	1563.12	-3.2812
HPT Efficiency (%)	92.1	92.471	0.4028
LPT Efficiency (%)	92.8	93.77	1.0452

4.2 Turbomatch transient performance validation

Except the steady state calculation, the accuracy of transient simulations also needs to be assessed. However, open literature on gas turbine transients often lacks crucial information for repeatable simulations, such as the performance maps used and detailed experimental data. Additionally, many outcomes are presented in normalized forms, which are not directly readable. For this reason, GasTurb, a commercial software, is selected for validating transient simulations. GasTurb provides detailed outcomes, and its accuracy has been validated by other researchers.

4.2.1 Engine Model Identification

Firstly, a one-spool turbojet engine is chosen as the simulation object. The engine structure of the turbojet engine is shown in Figure 4-3. The specific engine data and performance parameters are listed in Table 4-5. The design point of the turbojet engine is selected as the take-off condition (Sea level, Mach = 0, ISA = 0).

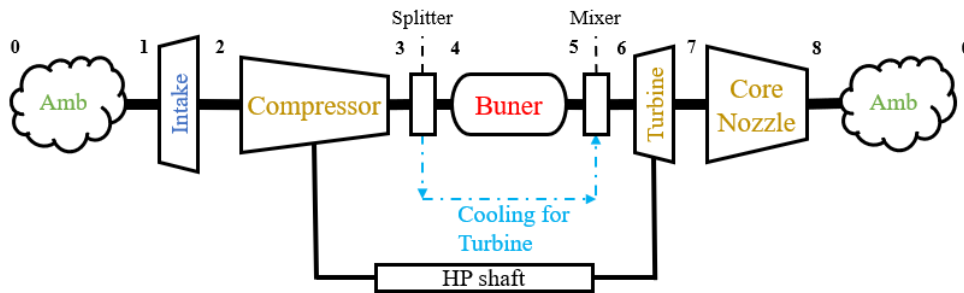


Figure 4-3 Schematic of the one-spool turbojet engine

Table 4-5 One-spool turbojet engine parameters (Sea Level, Static, ISA condition)

Parameters	Value
Mass Flow (kg/s)	32
Compressor Pressure Ratio	12
Compressor Isentropic Efficiency	0.85
Combustor Outlet Temperature (K)	1450
Combustor Efficiency	0.99
Turbine Efficiency	0.85
Polar Moment of Inertia (kg*m ²)	4.05
Cooling Flow (%)	10
Engine Shaft Speed (rpm)	14000
Thrust (kN)	26.73
SFC (g/(kN*s))	24.98

4.2.2 Component Map Identification

The component map that includes the component performance under different working conditions is the fundamental for gas turbine off-design and transient simulation because it represent the performance of real-world gas turbine components. In most condition, the performance map is based on adiabatic

experimental data. Since the transient simulation is assumed to be adiabatic in current stage, the map will be used in both off-design and transient simulation.

Besides the gas turbine model, the component map used by both software should be kept same.

The map used in GasTurb for the compressor is 99 NASA TM 101433 example compressor map and the map for the turbine is 99 AGARD two-stage turbine map. Although there are more selectable maps in GasTurb, in our case these two maps are used for validation. The unscaled map is shown in Figure 4-4 and Figure 4-5. For the rest of transient simulation in Section 5, standard compressor and turbine maps from the Turbomatch library are used.

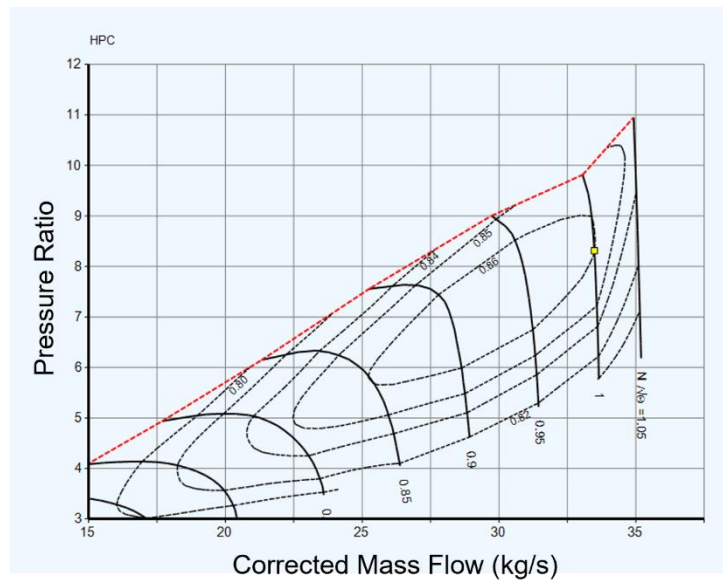


Figure 4-4 99 NASA TM 101433 example compressor map

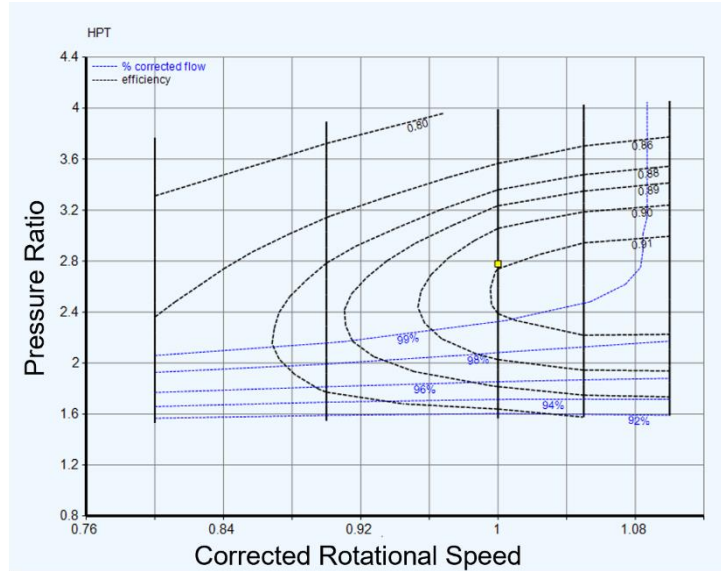


Figure 4-5 99 AGARD two-stage turbine map

The compressor map data in GasTurb consists of 10 speed lines, each with 10 points for each relative speed line. However, the map required by Turbomatch necessitates 20 speed lines with 20 points for each speed line. Therefore, it is necessary to interpolate the original GasTurb map to obtain more speed lines and data points. In this case, linear interpolation is chosen as the interpolation method for the map interpolation. To avoid inaccuracies caused by the interpolation process, the selected interpolation area of the map is positioned away from the designed compressor working range. The designed compressor's corrected rotational speed ranges from 0.7 to 1.0, and interpolation is not performed within this range. This approach helps to eliminate the effects of interpolation during the validation process. The interpolated compressor pressure ratio and efficiency maps are shown in Figure 4-6 and Figure 4-7.

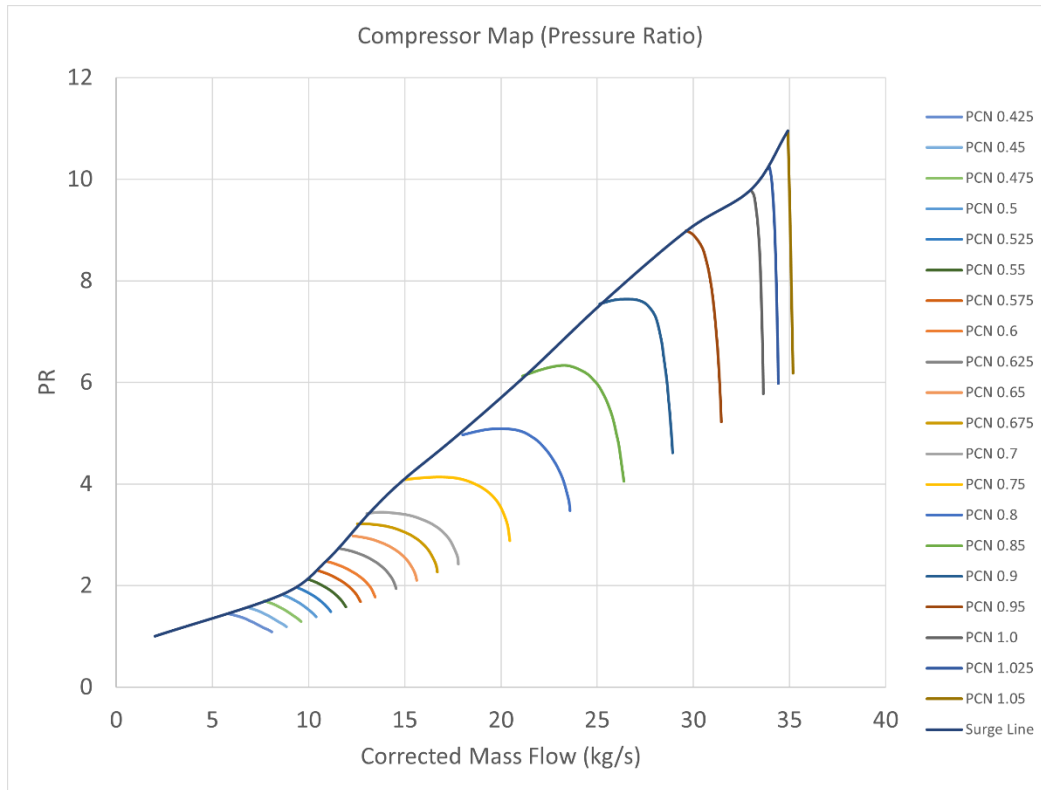


Figure 4-6 Interpolated compressor pressure ratio map

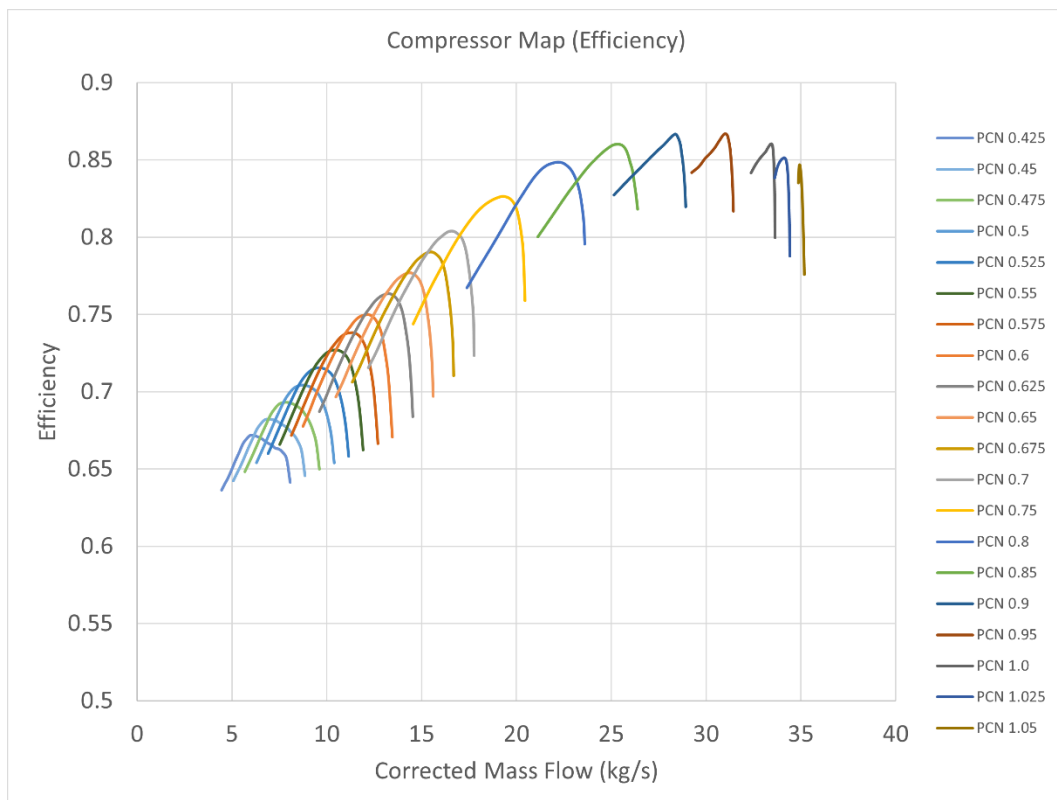


Figure 4-7 Interpolated compressor efficiency map

For the turbine map, more work is needed cause the map used by GasTurb is based on the turbine pressure ratio while that used by Turbomatch is based on the turbine enthalpy drop ratio. The equation used to transfer pressure ratio into the enthalpy drop ratio is shown by Eqs. (4-1), where $\frac{\Delta H}{T}$ is the turbine enthalpy drop ratio, which is the enthalpy change upon inlet total temperature, PR is the pressure ratio. During this process, it is assumed that the expansion in turbine is isentropic. Due to the temperature is unknown, the C_p value is assumed to be 1150 J/kg·K and the γ value is assumed to be 1.333, which are typical value during thermal dynamic calculation. The choice of C_p value and γ value will have an impact on the magnitude $\frac{\Delta H}{T}$ but will not affect the shape of the speed line. When using the map during off design and transient simulation, the map will be scaled based on the design point performance of actual gas turbine model. This will eliminate the inaccuracy caused by the choice of C_p and γ value.

$$\frac{\Delta H}{T} = C_p \times \left(1 - \left(\frac{1}{PR} \right)^{\frac{\gamma-1}{\gamma}} \right) \quad (4-1)$$

The data included in GasTurb turbine map has 7 speed lines and 10 points for each line. Same as the compressor, the linear interpolation method is also chosen here. The interpolated map is shown in Figure 4-8 and Figure 4-9.

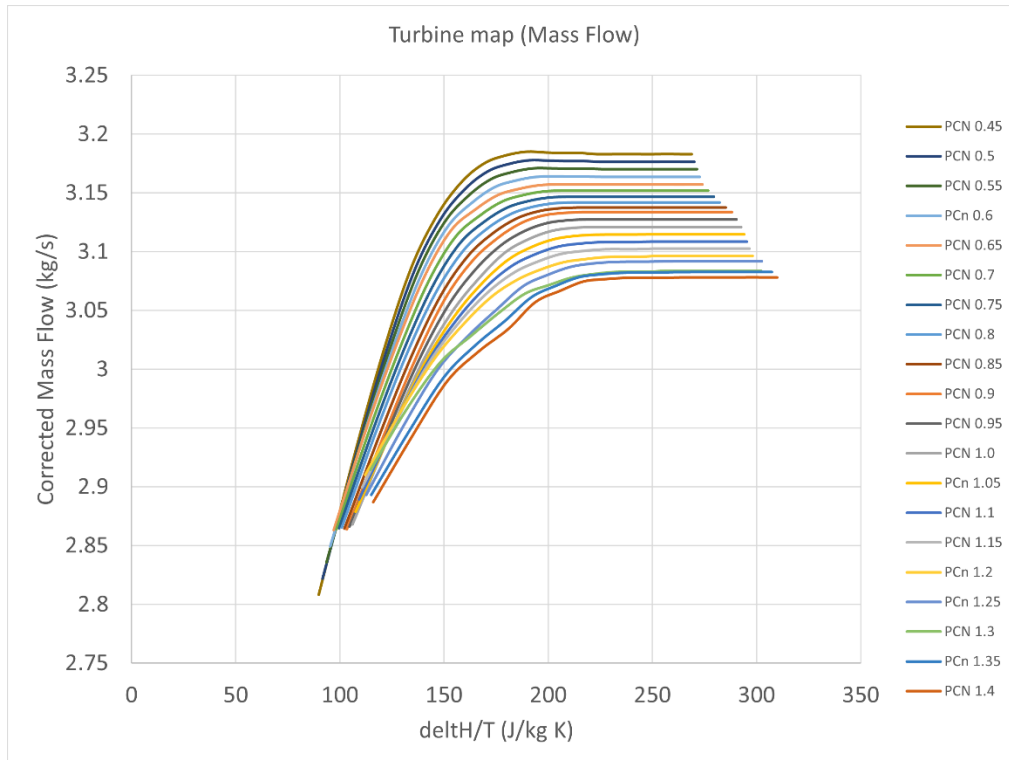


Figure 4-8 Interpolated turbine mass flow map

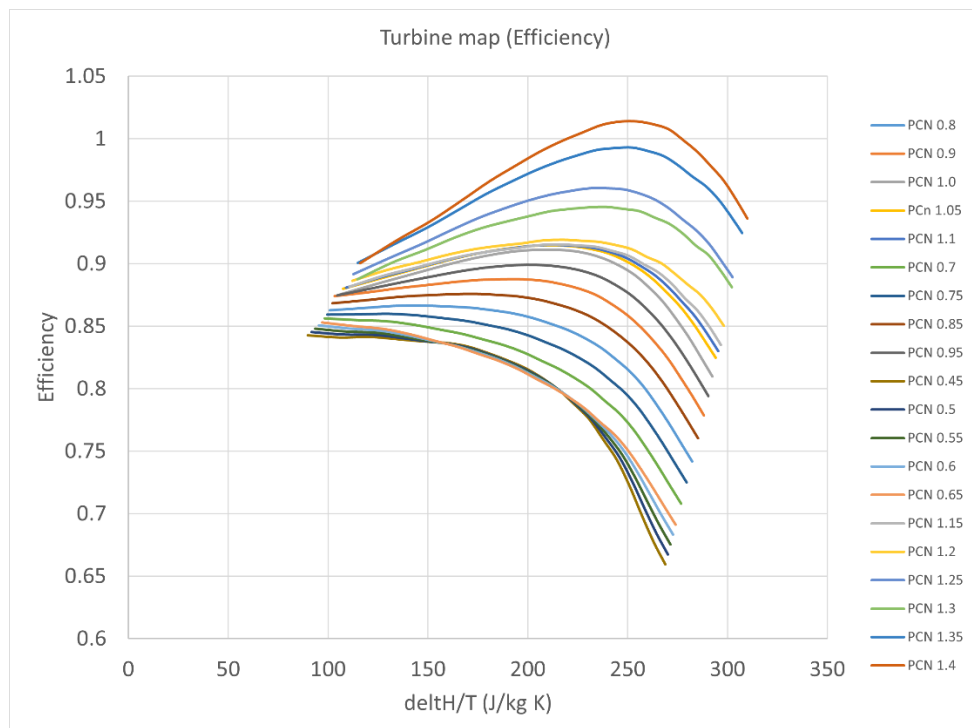


Figure 4-9 Interpolated turbine efficiency map

4.2.3 Map Scaling Point Selection

Since the pressure ratio in the GasTurb compressor map does not consistently increase along the mass flow for some relative speed lines (or we can say that the pressure ratio forms a concave function of mass flow), Turbomatch's map reading method may encounter errors when identifying the peak point in certain speed lines. Therefore, the selection of the scaling point becomes crucial as we aim to position our gas turbine operating region away from the peak region in the map.

Here, the beta value in the GasTurb compressor map, utilized for map scaling, is set at 0.3. While this alters the engine's design point position within the map, it effectively mitigates the map reading issue. To maintain consistency with the design point position, the Z value in Turbomatch is chosen as 0.33, ensuring uniform scaling factors for pressure ratio, mass flow, and efficiency.

For the turbine map, W/W_{max} value in Turbomatch is selected as 0.6443, and corrected rotational speed (CN) is set as 0.58 to have the same scaling factor with GasTurb and the same design point CN.

The final map scaling factors for the compressor and turbine in both software are listed in Table 4-6.

Table 4-6 Scaling Factor in Both Software

Parameter	GasTurb	Turbomatch
Compressor Mass Flow Scaler	/	0.9532999
Compressor PR Scaler	1.744	1.744
Compressor Eff Scaler	0.998	0.99821
Turbine Flow Scaler	/	1.957
Turbine PR Scaler	1.145	/
Turbine Δh Scaler	/	0.9597

Turbine Eff Scaler	0.9786	0.9786
--------------------	--------	--------

4.2.4 Nozzle Outlet Area Calculation

In GasTurb nozzle outlet area calculation, a parameter named after ‘Design Nozzle Petal Angle’ is used for nozzle discharge coefficient C_{D8} calculation. The C_{D8} map is shown in Figure 4-10. This index will change the nozzle outlet area by Eqs. (4-2) and (4-3), where $A_{eff,8}$ is the effective nozzle area calculated by nozzle outlet station flow parameters, W_8 is nozzle outlet mass flow rate, $T_{s,8}$ is nozzle outlet static temperature, $P_{s,8}$ nozzle outlet static pressure, V_8 is nozzle outlet flow velocity and A_8 is the geometric nozzle area used for thrust calculation.

$$A_{eff,8} = \frac{W_8 \cdot R \cdot T_{s,8}}{P_{s,8} \cdot V_8} \quad (4-2)$$

$$C_{D8} = A_{eff,8} / A_8 \quad (4-3)$$

Since the nozzle area in TURBOMATCH is the effective nozzle area, so the value for ‘Design Nozzle Petal Angle’ is set as 0 in GasTurb.

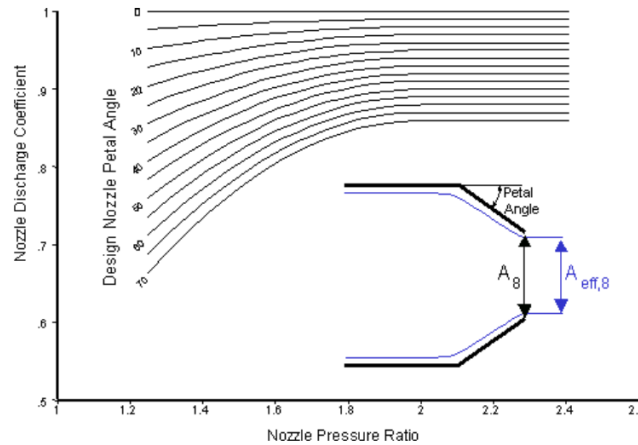


Figure 4-10 Nozzle discharge coefficient map [140]

4.2.5 Transient Performance Validation

This section will introduce transient fuel flow logic setting up and the transient outcome comparison between GasTurb and TURBOMATCH when keeping all controllable parameters, the same.

4.2.5.1 Transient Fuel Schedule Setting

To simulate the transient performance of gas turbine engine, it is necessary to set fuel flow schedule used for transient simulation. Several types of fuel schedule control logic can be selected, including fuel flow as a function of simulation time, fuel flow as a function of shaft corrected rotational speed, or fuel flow as a function of power level angle. Selection of a different fuel schedule control logic will lead to a different fuel flow value outcome through the simulation time and transient manoeuvre will be different. For the control variable purpose, the fuel schedule control logic used in both software is selected as the fuel flow as a function of simulation time. By setting this fuel schedule, the fuel flow value at every simulation time step can be controlled the same, which guarantee the exact same transient manoeuvre. Moreover, the transient simulation methodology used by GasTurb is the constant mass flow method. To keep the consistency, constant mass flow method was chosen in Turbomatch. Additionally, component performance maps used by both software are kept the same, as discussed Section 4.2.2. The transient manoeuvre was selected as a simple acceleration process. An acceleration from idle (shaft corrected rotational speed of 70%) to the maximum power (shaft corrected rotational speed of 100%) was set up. To prevent potential issues caused by a sudden increase in fuel flow, the fuel flow was linearly increased from the idle value to the maximum power value over a period of 1 second, as shown in Figure 4-11. The entire transient manoeuvre lasts for 15 seconds.

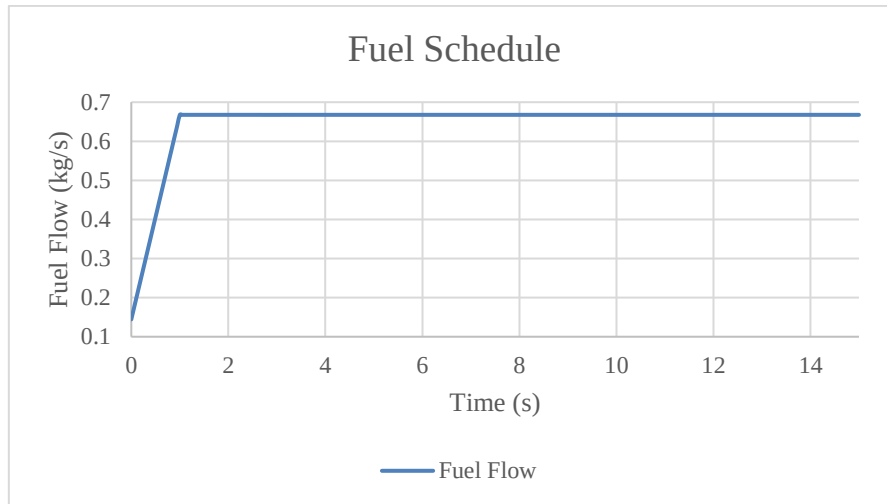


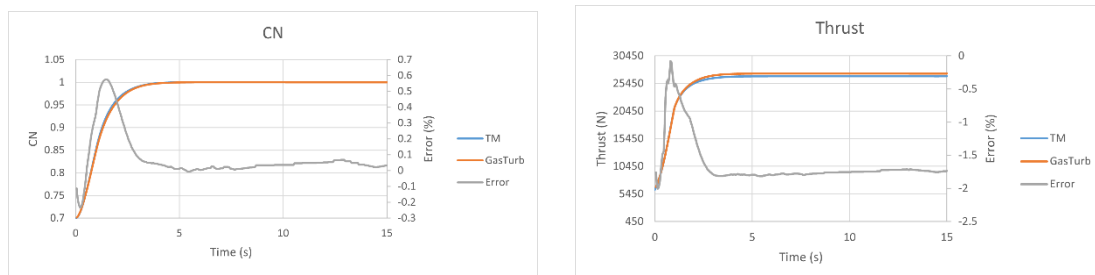
Figure 4-11 Transient Fuel Schedule for Gas Turbine Transient Performance

4.2.5.2 Burner Time Constant Setting

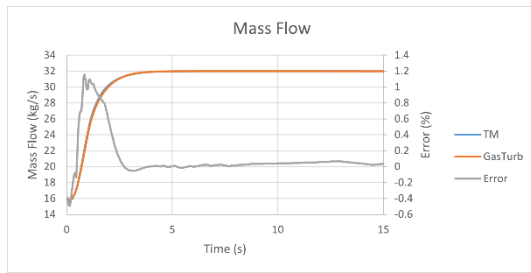
In GasTurb there is one parameter named after the ‘Burner time constant’ in the transient setting. This parameter is used to simulate the combustion delay in the combustor. In TURBOMATCH, there is no such setting, and combustion delay is not considered. In order to keep the consistency, this parameter in GasTurb is set as 0 during the transient performance simulation. A sensitivity analysis of the burner time constant is discussed in 7.8 Appendix D.

4.2.5.3 Transient Outcome Validation

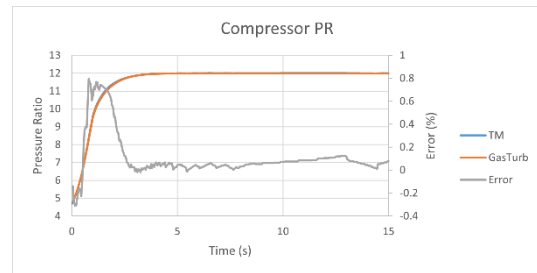
Transient outcomes are simulated and compared. The outcomes are shown in Figure 4-12 (a) to (i), where the blue line represents the Turbomatch outcome, the orange line represents the GasTurb outcome. Both of them use the primary coordinate axis on the left. The grey line represents the difference between Turbomatch and GasTurb when using GasTurb outcome as the baseline. It uses the secondary coordinate axis on the right.



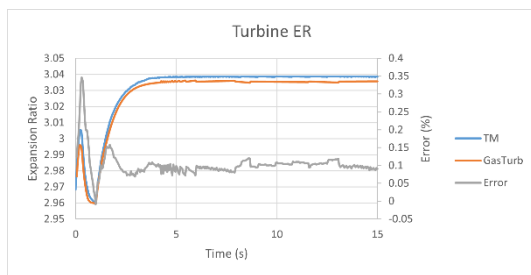
(a)



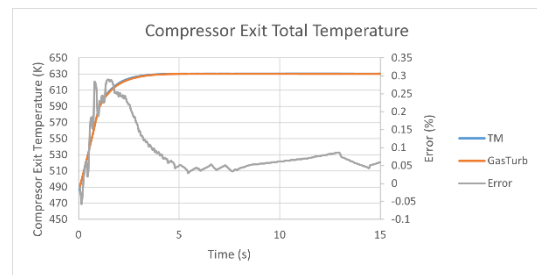
(b)



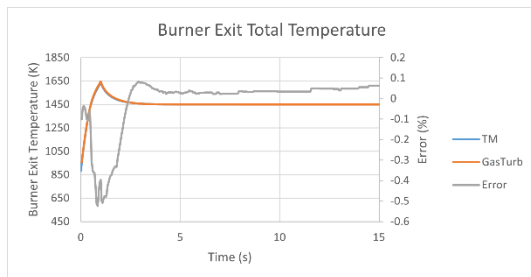
(c)



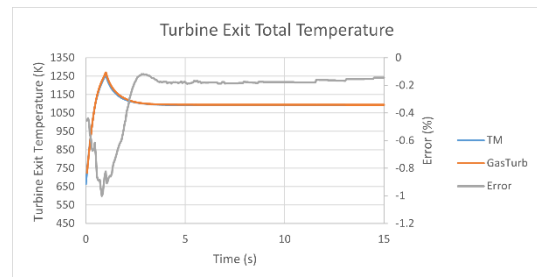
(d)



(e)

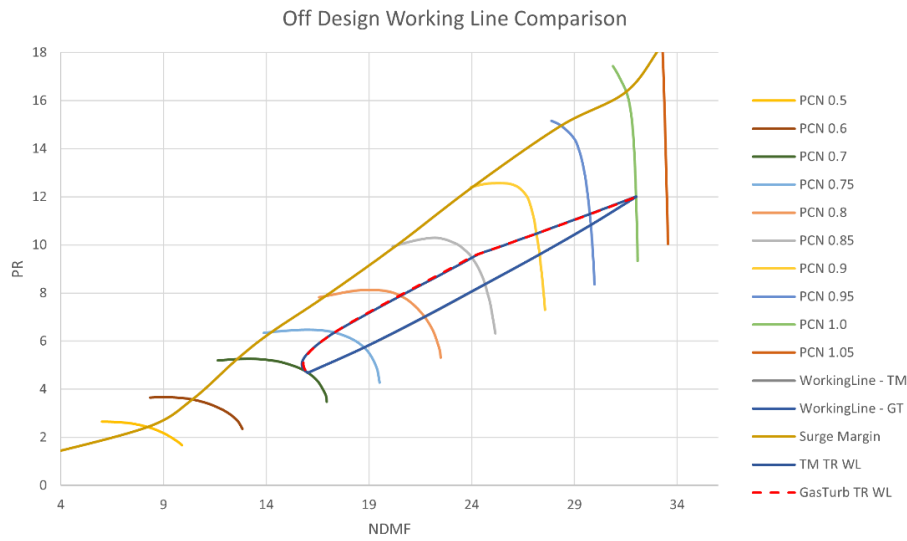


(f)



(g)

(h)



(i)

Figure 4-12 Transient Performance Comparison between Turbomatch and GasTurb

The corrected rotational speed over time is illustrated in Figure 4-12 (a), revealing slightly faster acceleration in Turbomatch compared to GasTurb. However, the difference between them is limited to 0.6%. The thrust and mass flow rate comparison are illustrated in Figure 4-12 (b) and (c). The maximum difference observed in thrust throughout the entire transient manoeuvre is -1.7%, while for mass flow, the difference reaches 1.2%. Figure 4-12 (d) and (e) show the compressor pressure ratio and turbine expansion ratio, respectively, with differences within 1%. The exit total temperatures of components are depicted in Figure 4-12 (f) to (h), where differences are also within 1%. The transient working line is shown in Figure 4-12 (i), demonstrating a good agreement between Turbomatch and GasTurb. Based on this comparison, the accuracy of Turbomatch transient simulation is assured.

4.3 Geometry Identification Model Validation

During the geometry identification model validation, it is firstly compared with the actual geometry of energy efficiency engine compressor. The design parameters for energy efficiency engine compressor is listed in Table 4-7. To have a better

prediction, the blade aspect ratio and pitch chord ratio for each stage blade are obtained from the report. The axial gap (which is a constant percentage of blade axial length) between the blade is added as a handle to control the total length of the compressor.

Table 4-7 Energy efficiency engine compressor parameter

Parameter	Value
Inlet Total Pressure (N/m ²)	59641.8
Inlet Total Temperature (K)	304.4
Number of Stage	10
Mass Flow (kg/s)	31.1545
Pressure Ratio	25
Inlet Hub/Tip Ratio	0.5074
Inlet Axial Velocity (m/2)	200

The simulation outcome is shown in Figure 4-13. The red solid line represents the geometry identification model simulation outcome. The blue dashed line represents the actual compressor geometry data from NASA report.

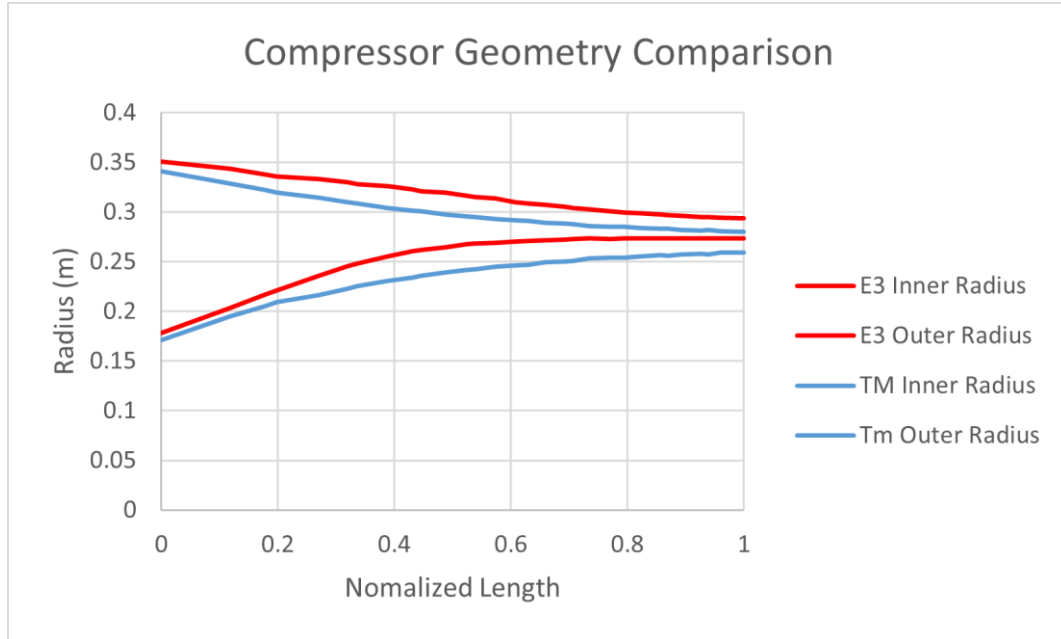
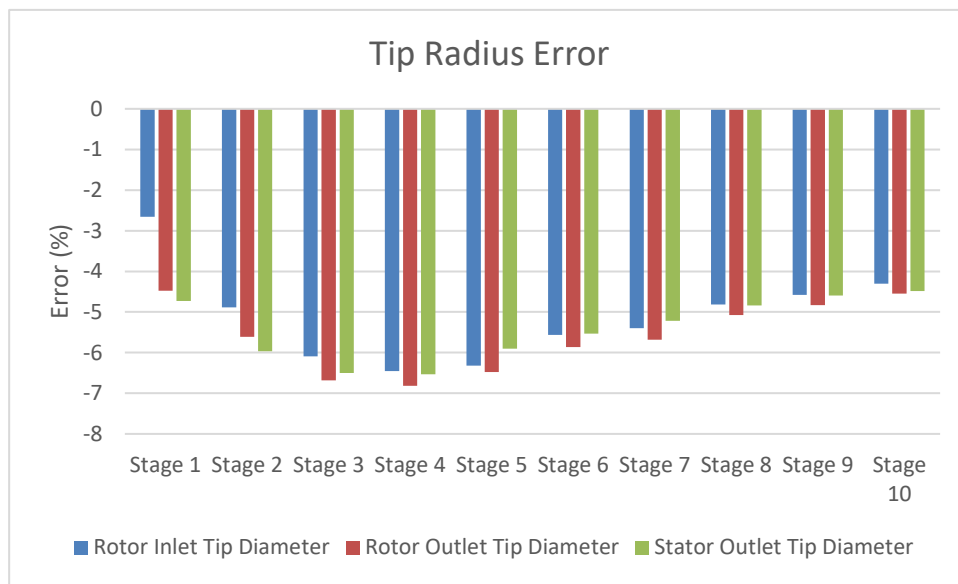
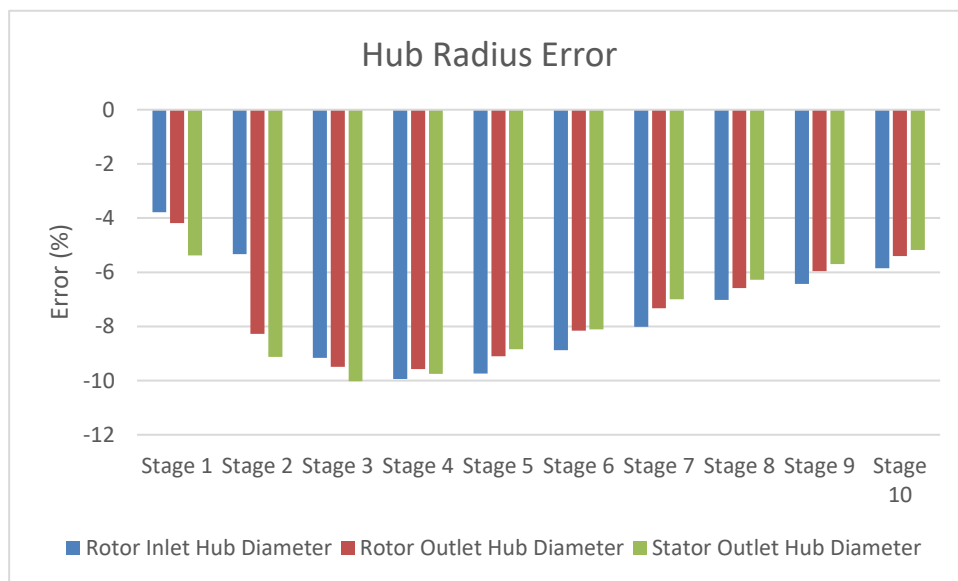


Figure 4-13 Comparison between E3 compressor and simulation outcome

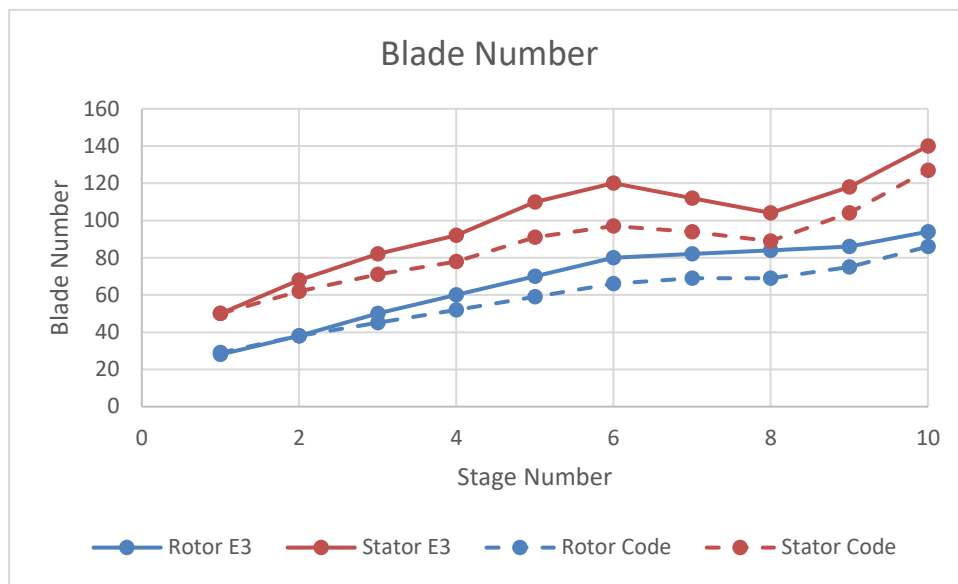
The inner radius and outer radius at each stage, the blade axial length, and blade number at each stage are chosen as the validation data. The difference between the simulation outcome and the actual compressor data are shown in Figure 4-14 (a) to (d). Based on the comparison, the maximum difference tip radius is -6.8%, and the maximum difference hub radius is -9.57%. The maximum difference in blade number is -19.2%. The primary reason for the discrepancy is the geometry of the flow path. The proposed geometry identification model is a preliminary design tool, constrained to constant tip diameter, constant hub diameter, and constant mean diameter. In reality, compressor geometry does not adhere to these constant values due to aerodynamic and mechanical stress considerations, among other factors. This leads to significant differences in the actual compressor geometry. Due to the differences in geometry, the inner and outer radii will differ from the actual values, leading to discrepancies between the estimated and actual blade heights. The estimated blade number is further affected by the actual and estimated geometry. In the model, the number of blades is estimated based on the pitch and chord ratio and the perimeter at the mean radius. A smaller mean radius led to a smaller perimeter and a lower estimated blade number.



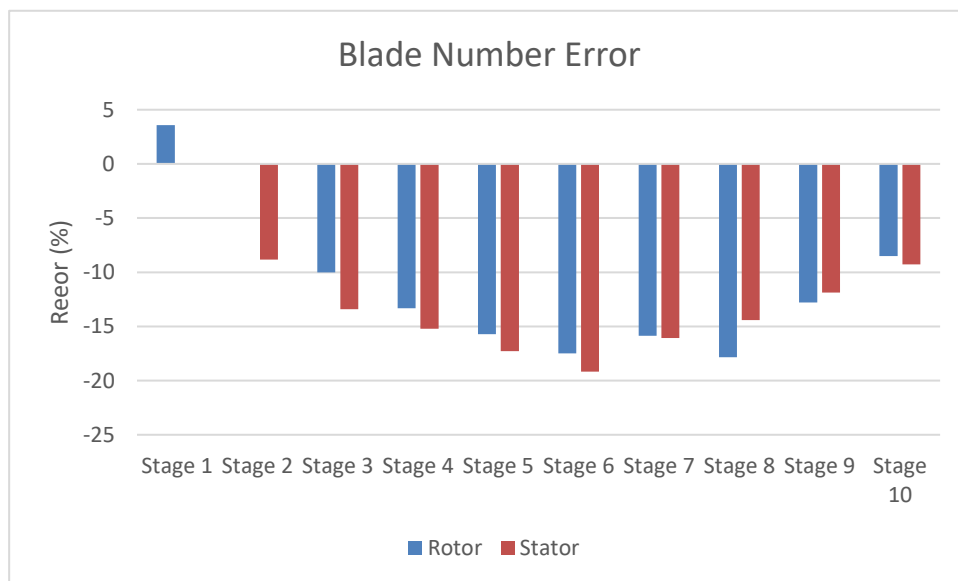
(a)



(b)



(c)



(d)

Figure 4-14 Difference between actual compressor geometry and simulation outcome

During the previous comparison, the mass and wetted area value are not provided by NASA report. In order to validate the mass and wetted area estimation accuracy. GasTurb is chosen again because it can offer a comprehensive data. Besides, the geometry model used by GasTurb is also a

preliminary design model, which is similar with the proposed geometry identification model. Gas turbine engine model described in Section 4.2.1 is used during the validation. The compressor in the engine model is a 5-stage axial compressor with the constant tip diameter geometry selected. The combustor is an annulus shape combustor. The turbine is a single stage turbine with the constant mean diameter geometry selected. For the detailed geometry data are listed in Table 4-8.

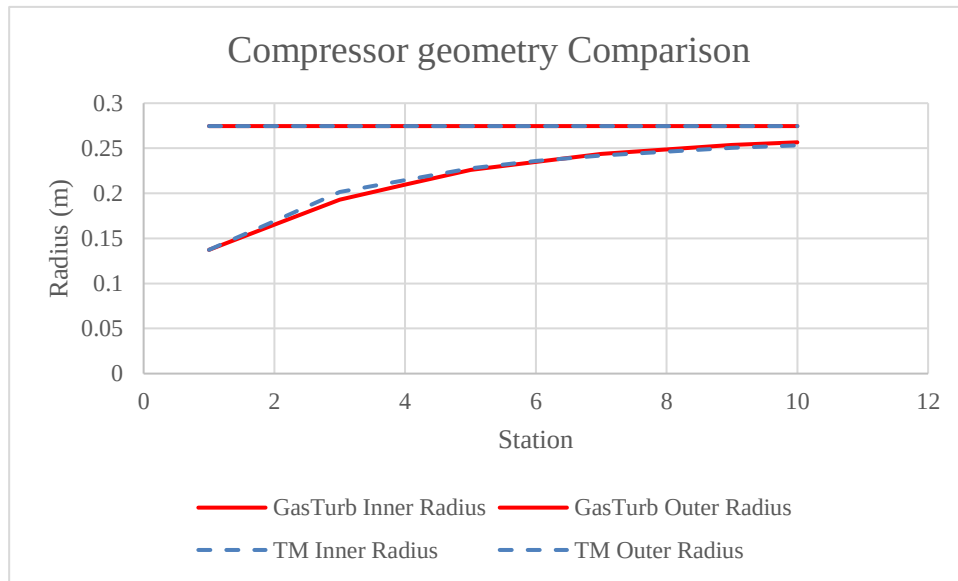
Table 4-8 Geometry Data

Parameters	Value
Compressor Shaft Radius (m)	0.067
Compressor Number of stage	5
Compressor Inlet Hub/Tip Ratio	0.5
Design Point Inlet Mach Number	0.5
Annulus Configuration	Constant Tip Diameter
Compressor Blade Aspect Ratio	1.5
Compressor Blade Pitch/Chord Ratio	0.5
Mean Blade Thickness (% of Chord)	1
Casing Thickness (m)	0.005
Material	Ti-6Al-4V
Combustor Type	Annulus
Combustor Length (m)	0.5
Flame Tube Wall Thickness (m)	0.005
Combustor Casing Inner Radius (m)	0.368
Material	N-155

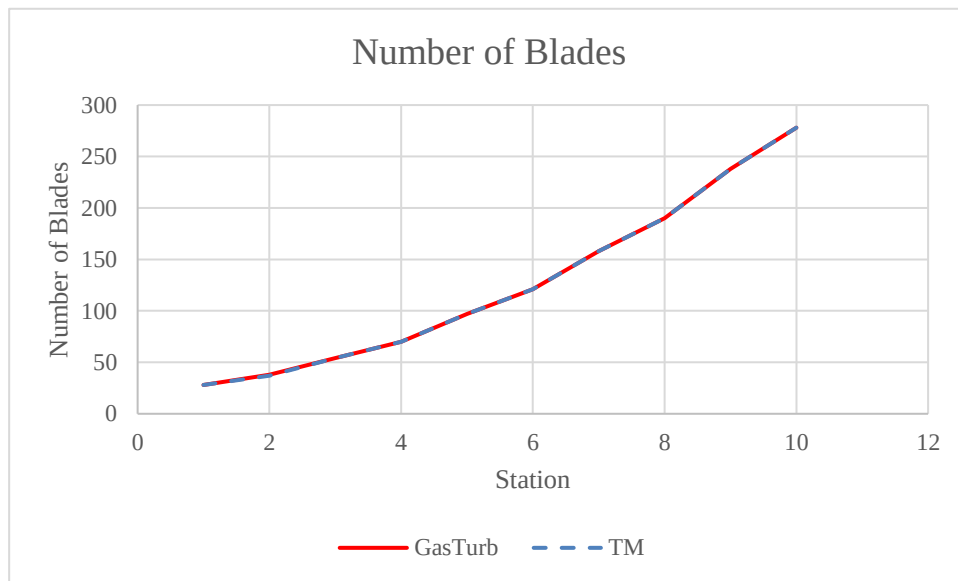
Turbine Shaft Radius (m)	0.0557
Turbine Number of Stage	1
Turbine Inlet Hub/Tip Ratio	0.5
Turbine Inlet Mach Number	0.5
Annulus Configuration	Constant Hub Diameter
Mean Blade Thickness (% of Chord)	1.5
Casing Thickness (m)	0.005
Material	Inconel 718

Based on the design point data and geometry data, two engine model are created. One is based on the Turbomatch, and the other is based on the GasTurb. All the input data were controlled the same.

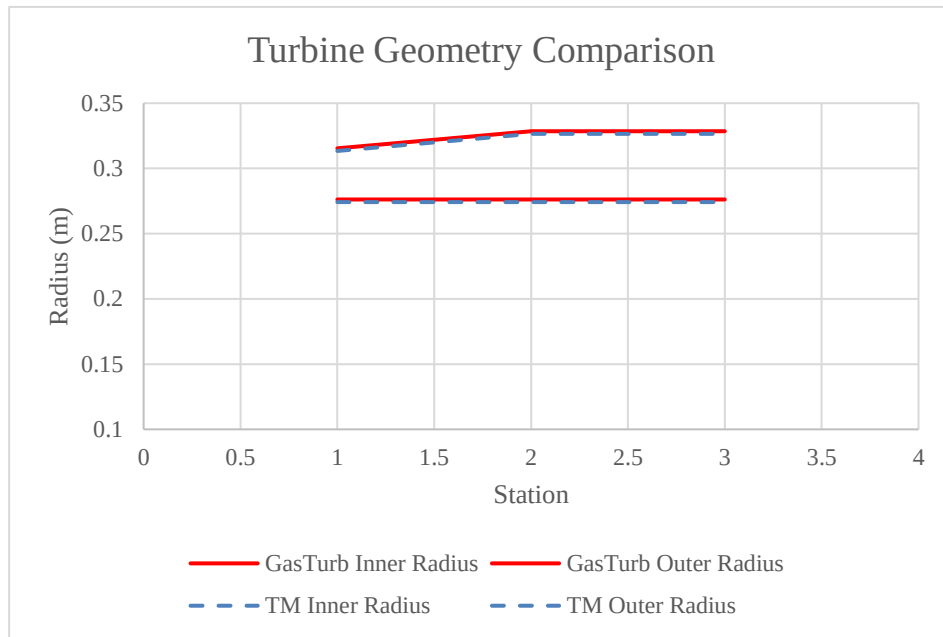
According to the geometry identification outcome, the maximum difference between the two software regarding the compressor's inner radius is 4.3%, while the difference in blade number is only 1, occurring at stator 1. For the turbine, the maximum difference for geometry is 1.3%, with no difference in blade number. The design outcomes by GasTurb and TM are illustrated in Figure 4-15. The reason why discrepancies occur is due to the different applied Mach number distribution model. For the proposed model, the constant axial velocity is assumed for each stage Mach number estimation. However, in GasTurb, the Mach number distribution is based on a liner interpolation based on the user defined inlet and outlet Mach number. With different Mach number distribution model, the flow function distribution will be different and lead to a difference in flow area and geometry of component.



(a)



(b)



(c)

Figure 4-15 Geometry comparison between Turbomatch and GasTurb

To ensure consistency, the wetted area and mass of each component are also compared. While dividing the compressor into stages ensures similarity in each stage's mass and wetted area, maintaining overall consistency across components is challenging. Therefore, the total mass and wetted area of the components are compared to align both software at the same level. The detailed data are presented in Table 4-9. The highest difference is found in compressor blade mass, which is overestimated by 4.82%. With differences limited to within 5%, the geometry design outcome is reasonably good for the current study.

Table 4-9 Wetted area and mass of component for Turbomatch and GasTurb

	GasTurb	TM	Difference (%)
COMPRESSOR Blade Mass	53.87 (kg)	53.79 (kg)	-0.14
COMPRESSOR Blade Area	4.855 (m ²)	5.101 (m ²)	4.82

COMPRESSOR Casing Mass	13.28 (kg)	13.27 (kg)	-0.06
COMPRESSOR Casing Area	1.055 (m ²)	1.048 (m ²)	-0.71
COMPRESSOR Disc Mass	43.47 (kg)	42.34 (kg)	-2.60
COMPRESSOR Disc Area	0.257 (m ²)	0.256 (m ²)	-0.08
COMBUSTOR Can Mass	33.34 (kg)	33.54 (kg)	0.6
COMBUSTOR Can Area	1.667 (m ²)	1.687 (m ²)	1.21
TURBINE Blade Mass	10.93 (kg)	10.93 (kg)	-0.01
TURBINE Blade Area	1.041 (m ²)	1.046 (m ²)	0.55
TURBINE Casing Mass	4.333(kg)	4.246 (kg)	-2.00
TURBINE Casing Area	0.108 (m ²)	0.106 (m ²)	-2.00
TURBINE Disc Mass	23.19 (kg)	22.83 (kg)	-1.55
TURBINE Disc Area	0.045 (m ²)	0.044 (m ²)	-2.99

Overall, the proposed geometry identification model shows an approximately 20% maximum difference with the actual engine data and it is mainly caused by

the limitation of the preliminary design (the geometry is limited within constant tip diameter, constant hub diameter, and constant mean diameter). When comparing the simulation outcome with GasTurb, where the geometry model is also based on preliminary design, the difference is within 5%. This indicates that the accuracy of the proposed model is trustable based on the criteria in other research publication [150,162].

4.4 Heat Soakage Model Validation

When validating the heat soakage model, two parts need to be validated. The first part is the heat flow rate prediction accuracy during transient manoeuvre. And the second part is the transient performance prediction accuracy with the consideration of heat soakage effect. However, in the field of heat soakage effect on gas turbine transient performance, no suitable validation material is provided in the public domain. Due to this reason, GasTurb is chosen to validate the proposed heat soakage model because it includes a simple heat soakage model.

4.4.1 Heat Flow Rate Estimation Model Validation

During the heat flow rate prediction accuracy validation, the same gas turbine structure as described in Section 4.3 is selected during the heat soakage model validation. According to the geometry identification outcome discussed in Section 4.3, the quasi-2D thermal network is established.

As mentioned before, the heat soakage model used in GasTurb is different with the proposed heat soakage model. For GasTurb, the heat soakage model is based the conventional lumped parameter method, which treat the whole component as a lumped node. Convective heat transfer is neglected. However, the proposed heat soakage model is a quasi-2D model which establish the thermal network based on each stage of compressor and turbine elements (blade, casing and disc) and combustor zone, where convective heat transfer is considered. When estimating the heat flow rate of the combustor, the mean gas temperature based on the inlet and outlet condition of the combustor is used by GasTurb's heat soakage model. In the proposed model, combustor zone temperature is applied to have a more realistic temperature distribution. Besides,

the radiative heat transfer is considered. Moreover, the proposed heat soakage model considered the cooling technologies including film cooling, internal cooling and thermal barrier coating. These are not included in GasTurb's heat soakage model.

When estimating the heat flow rate, the mentioned differences will lead to different heat transfer process. The convective heat transfer will lead to a delay in heat transfer process. The combustor temperature distribution will lead to a higher estimated combustor temperature compared with the outcome from mean temperature. The radiative heat transfer will increase the heat flow rate. The film cooling and thermal barrier coating will reduce the wall temperature. The internal cooling will move part of the heat. To keep the consistency of the heat transfer process, modification has to be applied to the boundary condition of the proposed heat soakage model.

Based on the difference between GasTurb's heat soakage model and proposed heat soakage model, the combustor temperature profile is set as the mean temperature based on combustor inlet and outlet condition during the estimation to keep the consistency. The radiative heat flow rate is removed. The convective heat transfer coefficient is set as 0 and the nodes temperature is updated based on the convective heat flow rate. This neglect the effect coming from convective heat transfer. Film cooling efficiency is set to 0 to neglect the effect coming from film cooling. The internal cooling boundary is set as adiabatic boundary condition to neglect internal cooling. The thermal barrier coating thickness is set to 0 to avoid the effect coming from thermal barrier coating. By setting these boundary condition, heat transfer process of the proposed heat soakage model is kept consistent as the heat soakage model used by GasTurb.

Based on the gas turbine model discussed in 4.2.1 and transient maneuverer 4.2.5.1, a comparison of the heat flow rate in each engine component over time between TM and GasTurb is presented in Figure 4-16. It can be found that the magnitude of the estimated heat flow rate from two software show a good agreement with each other through the entire transient manoeuvre. Based on the

comparison, the maximum heat flow rate differences for the compressor, combustor, and turbine are -5.88%, 7.94%, and -3.08%, respectively.

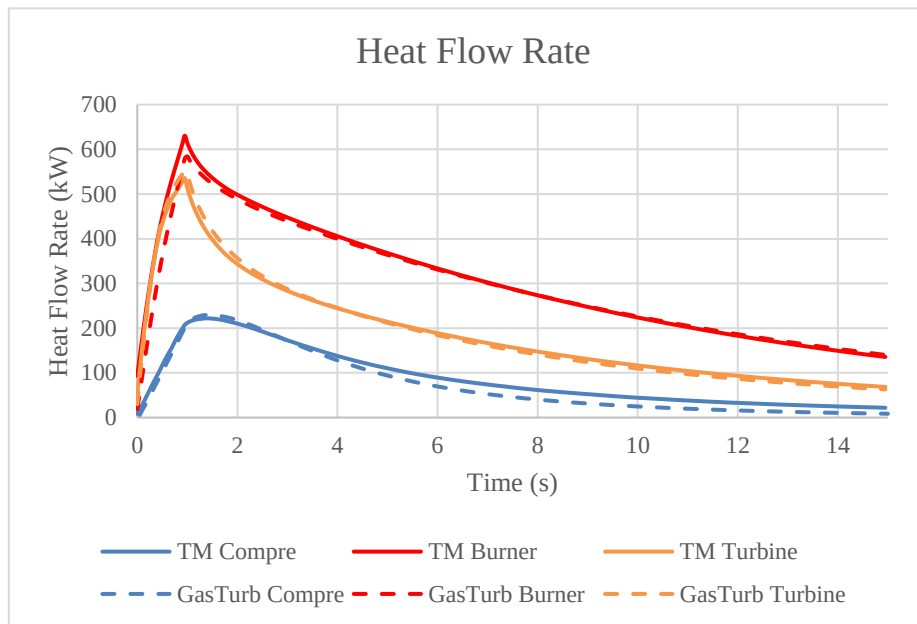


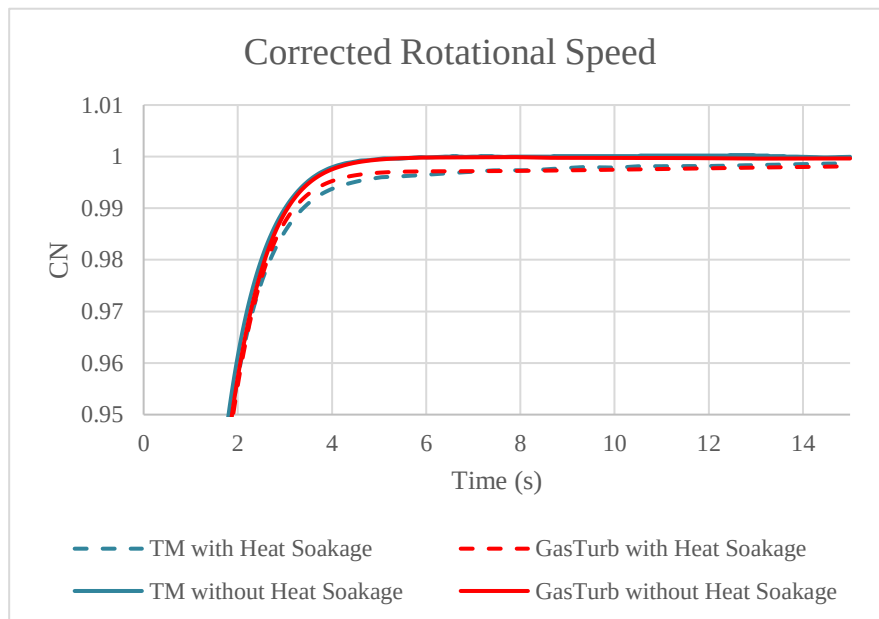
Figure 4-16 Heat flow rate comparison between TM and GasTurb

Due to the validation on heat flow rate estimation is based on the comparison with GasTurb and the boundary condition is modified to keep the heat transfer consistency. A sensitivity analysis on the model input effect on heat flow rate estimation outcome is discussed in Section 5.1.

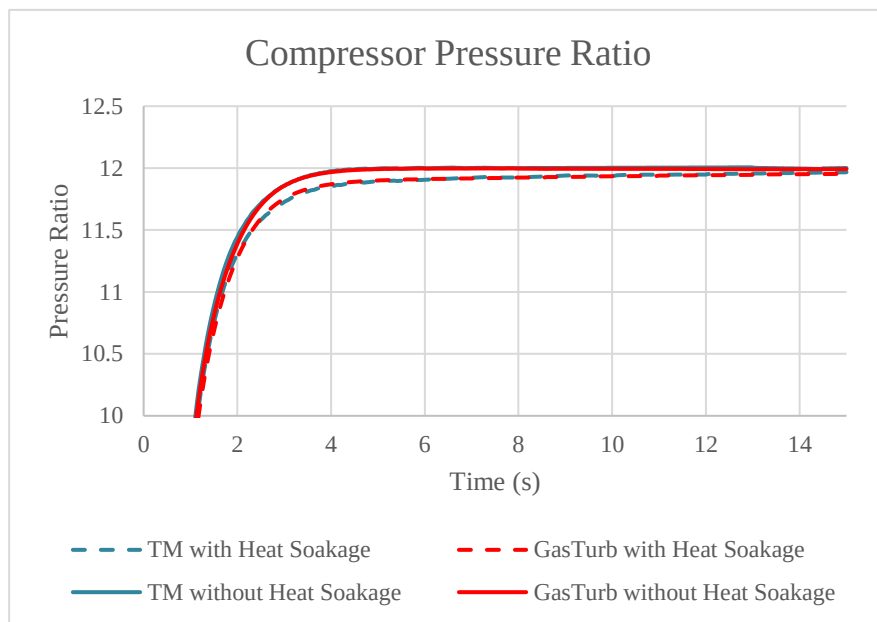
4.4.2 Heat Soakage Effect on Transient Performance Model Validation

The heat flow rate estimation accuracy is important. However, it is more important to analysis the heat soakage effect on gas turbine transient performance. the parameter chosen for validation are shaft corrected rotational speed, compressor pressure ratio, compressor outlet temperature, combustor outlet temperature and exhaust gas temperature. Comparison between Turbomatch and GasTurb with and without heat soakage are shown in Figure 4-17, where the solid lines represent the condition without considering heat soakage, while the dashed lines represent the condition with the consideration of heat soakage.

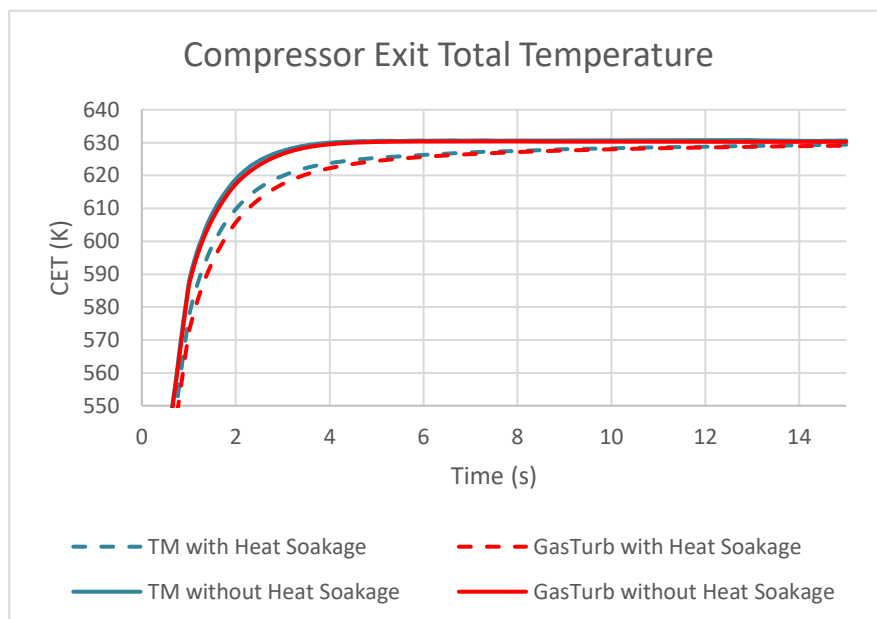
For CN, the maximum difference observed is 0.57% at 1.45 simulation seconds in the transient condition and 0.411% at 1.3 simulation seconds in the heat soakage condition. The maximum differences for the compressor pressure ratio are 0.065 in the transient condition and 0.073 in the heat soakage condition, both within 0.7%. For compressor outlet temperature, the maximum difference is 0.29% at 1.5 simulation seconds in transient condition and 0.84% at 1.35 simulation seconds in heat soakage condition. For combustor outlet temperature, the maximum difference is -0.5% at 1.1 simulation seconds in transient condition and 0.4% at 3.2 simulation seconds in heat soakage condition. The maximum difference for the exhaust gas temperature is 12.25K in the transient condition and 23.37K in the heat soakage condition, within 2.5%. Based on the above discussion, the engine heat soakage component performance modification method has been validated.



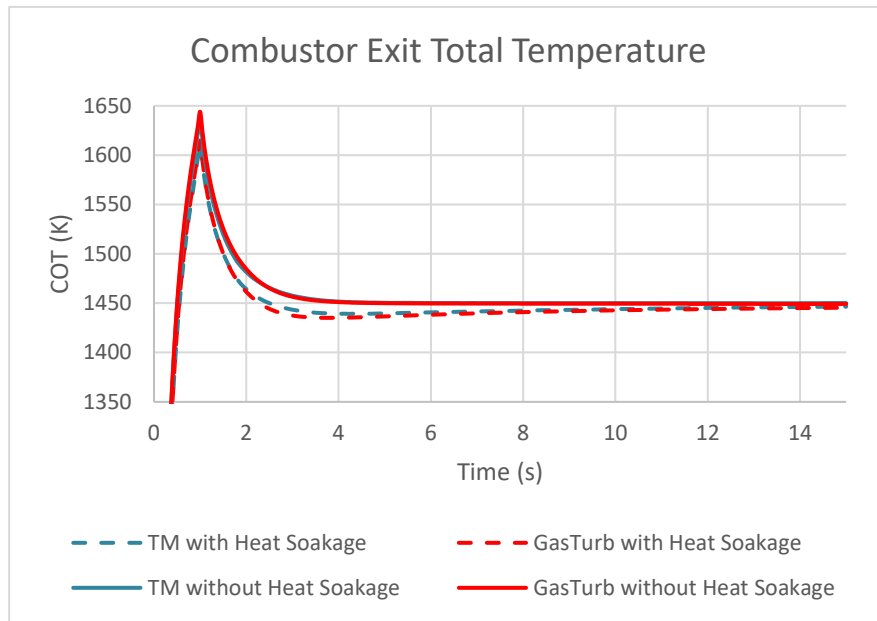
(a)



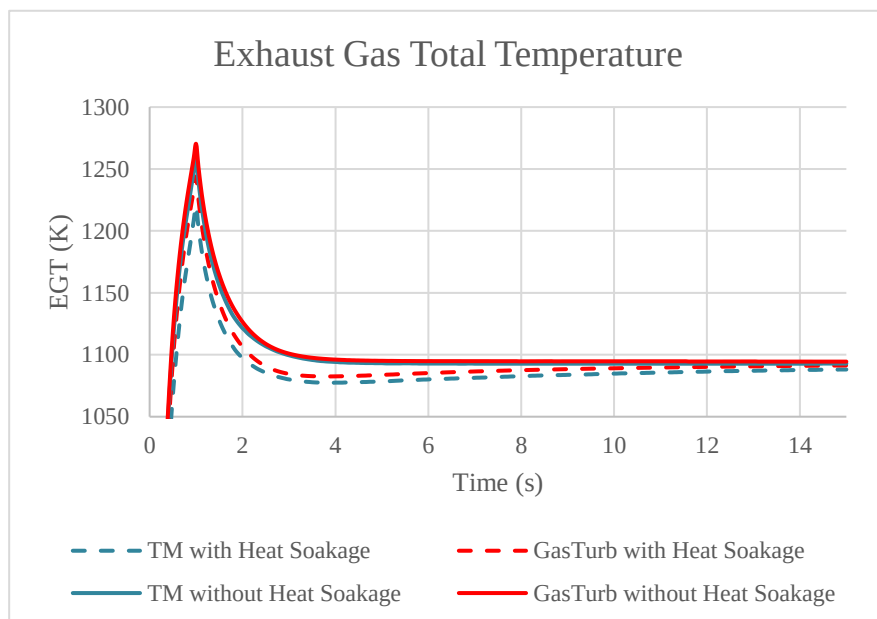
(b)



(c)



(d)



(e)

Figure 4-17 Gas turbine transient performance comparison between TM and GasTurb with and without heat soakage effect

4.5 Tip Clearance Model Validation

GasTurb was initially selected as the validation benchmark due to its inclusion of tip clearance estimation during transient simulation. The gas turbine model discussed in Section 4.2.1 is selected as the simulation object. However, the tip clearance datum condition in GasTurb and Turbomatch is different. For GasTurb, the datum condition is based on the last off-design condition which is also the starting point of transient simulation. For Turbomatch, the datum condition is chosen as design point condition. In order to keep the datum condition, the same, the transient manoeuvre should start from the design point. Since the design point of the gas turbine model describe in Section 4.2.5 is based on full rotational speed, a deceleration from the design point to idle condition was selected as the transient manoeuvre. By using this way, the datum condition of tip clearance and transient manoeuvre can be kept the same. The fuel flow schedule is shown in Figure 4-18, similar to the schedule discussed in Section 4.2.5.1, based on time for control purposes. The tip turbine clearance change over time is shown in Figure 4-19, demonstrating good agreement between the two software throughout the duration. The maximum difference is 0.18% of the blade length. However, towards the end of the transient simulation, the tip clearance reduction rate in GasTurb is smaller. This is because GasTurb disregards the thermal expansion of the disc during short transient simulation spans. The live disc radius is only affected by centrifugal force. This approach is reasonable for short-duration transient simulations, as disc heat transfer is relatively slow. Nonetheless, long-duration transient simulations remain necessary.

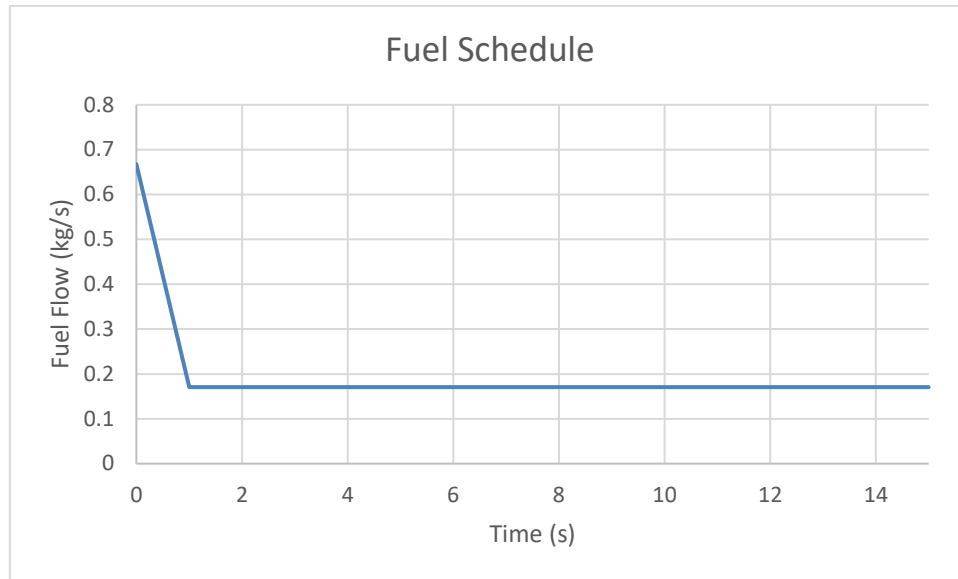


Figure 4-18 Fuel schedule for tip clearance validation against GasTurb

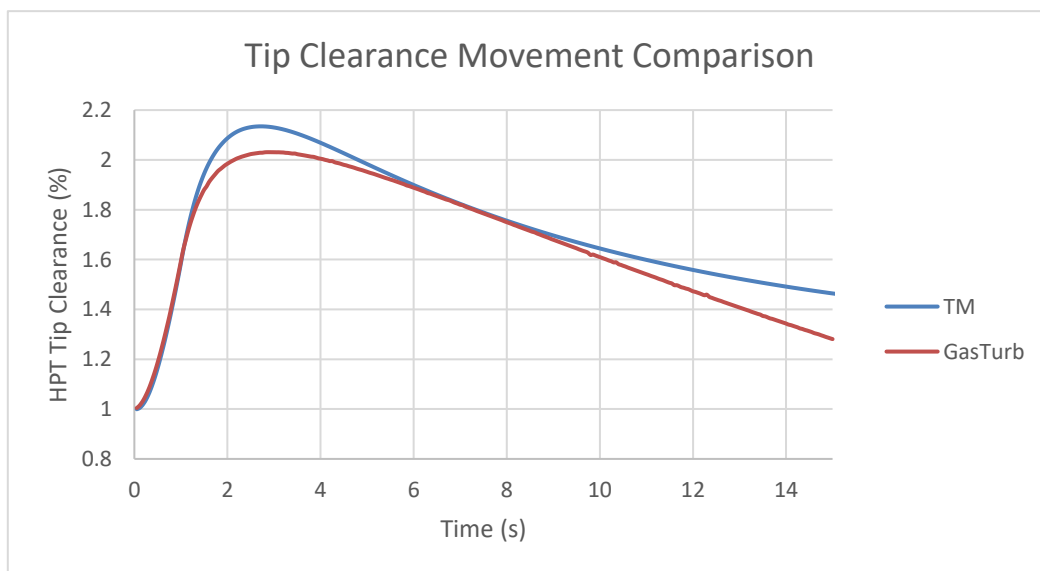


Figure 4-19 Turbine tip clearance change comparison between GasTurb and Turbomatch

Unfortunately, detailed geometry data for tip clearance models are not provided in open literature by most investigators. As a result, when comparing tip clearance movement during a long-duration transient manoeuvre, only qualitative analysis can be conducted, rather than quantitative analysis.

For the long-duration transient comparison, the research findings from Kumar et al. [163] are chosen. In Kumar's study, a full 3D turbine stage in a small gas

turbine engine was established. The geometry of the turbine rotor established by Kumar et al. is shown in Figure 4-20.

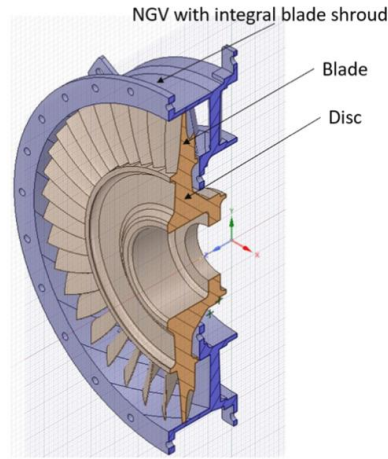
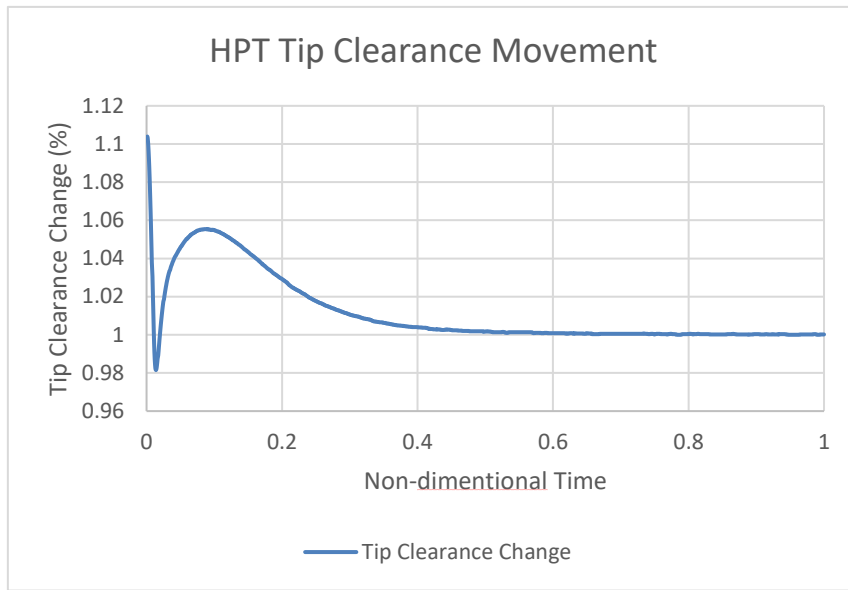
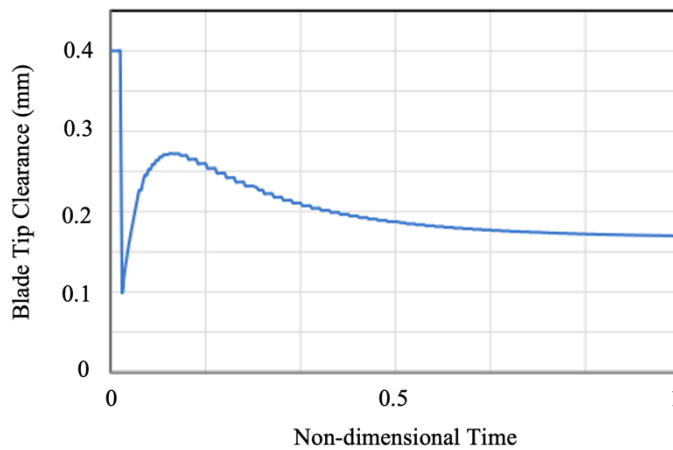


Figure 4-20 The geometry of the turbine rotor in Kumar's research [163]

An acceleration transient was simulated. This transient manoeuvre based on gas turbine model discussed in Section 4.2.1 and the fuel flow schedule discussed in Section 4.2.5.1 is simulated. The tip clearance movement of the HPT is shown in Figure 4-21 (a), and the variation of tip clearance from Kumar's work is shown in Figure 4-21 (b). From the comparison, it can be observed that both models capture the same trend of tip clearance movement. Initially, there is a sudden reduction in tip clearance due to the rapid change in blade length and disc radius caused by increasing in rotational speed. Subsequently, the tip clearance increases due to casing thermal expansion, followed by a reduction to the equilibrium condition. During this stage, the dominant factor is the change in turbine thermal displacement. Based on these findings, it can be concluded that the results produced by the proposed tip clearance model are reliable.



(a)



(b)

Figure 4-21 Tip clearance variation comparison

By comparing Figure 4-19 and Figure 4-21, the impact of the different choice of tip clearance condition can be investigated. In the condition of GasTurb, where the datum condition is selected as the starting point of transient manoeuvre, the tip clearance will always start from the user setting tip clearance value (the value that will not affect component efficiency). However, this means that the datum condition will always change under different transient manoeuvres. For example,

for the acceleration process starting with CN equals to 0.7 and deceleration process starting with CN equals to 1.0, the tip clearance will both start from user setting value. This is not in line with reality. But if you set the datum condition based on the design point condition, the tip clearance at the beginning of transient manoeuvre will be dependent on the selection of transient manoeuvre, which is closer to reality.

4.6 Mean Line Model Validation

In order to create the map modification model that described in Section 3.5.3, the accuracy of the proposed mean line model that discussed in Section 3.5.1 and 3.5.2 should be validated. In this section, the adiabatic mean line model (without consideration of heat transfer effect) and the non-adiabatic mean line model (with consideration of heat transfer effect) will be discussed.

4.6.1 Adiabatic Mean Line Model Validation

The mean line model accuracy is firstly compared with the NASA stage 35 experiment data [164]. NASA stage 35 is a single stage compressor with design point pressure ratio of 1.82. The flow passage of NASA stage 35 is shown in Figure 4-22.

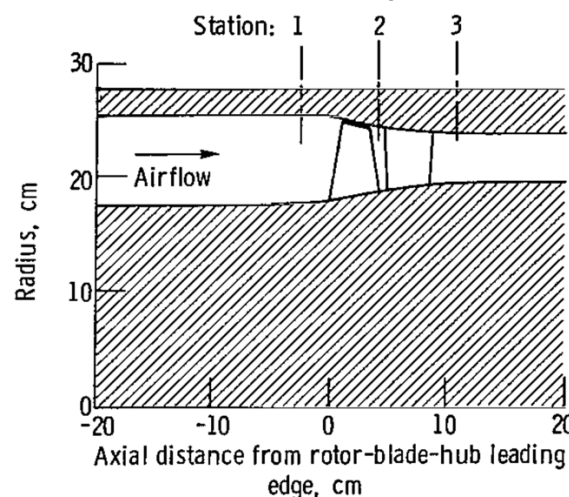


Figure 4-22 Flow passage for NASA stage 35 [164]

By using the proposed adiabatic mean line model described above, several speed lines of the single stage compressor have been simulated, including 100%,

90%, 80%, and 70% speed lines. The comparison between the experiment data and simulation outcome is shown in Figure 4-23. The red dots represent the NASA experiment data and the dashed blue line represent the mean line model outcome. It can be found that the mean line model has a good agreement with the experiment data. For speed line 100% and 90%, the differences are controlled within 2%. For speed line 70%, there is a relatively larger difference in the speed line, however, the difference is still controlled within 6%. Based on the comparison outcome, the accuracy of the developed adiabatic mean line model is validated.

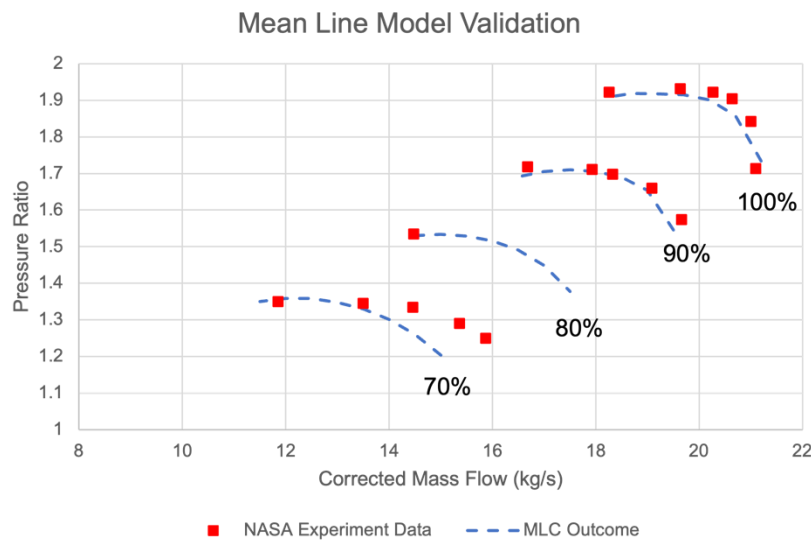


Figure 4-23 Comparison between the developed mean line model and NASA experiment data

4.6.2 Non-Adiabatic Mean Line Model Validation

Based on the developed adiabatic mean line model, the effect coming from heat transfer is also analysed and the simulation outcome is compared with the 3D CFD model of NASA rotor 35 developed by Shah [165]. The 3D CFD model is shown in Figure 4-24.

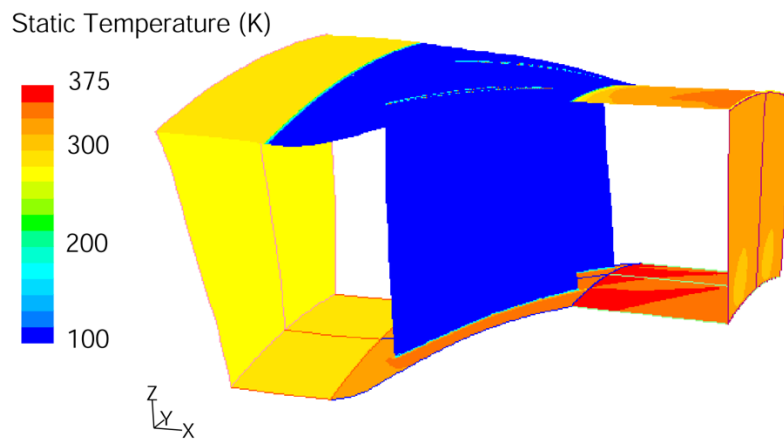


Figure 4-24 3D CFD model of NASA rotor 35 developed by Shah [165]

The rotational speed was set as 93.7% of the design rotational speed and the q^* value provided by Shah was -0.008. The comparison is shown in Figure 4-25, where the red solid and dashed line represent the outcome under adiabatic condition. The blue solid and dashed line represent the outcome with the consideration of heat transfer effect. The reason why the shape of the speed line that calculate from CFD is different from what calculate from mean line code is that the pressure ratio computed in the CFD cases is based on a mass-average of the total pressure in the absolute frame of reference at the exit plane of the computational domain. Although the shape of the speed line various with each other, the deviation of the speed line due to heat transfer effect has a good agreement. At a corrected flow of 19.27 kg/s, the pressure ratio of the compressor increased by 2.84% based on Shah's model. In the developed mean line model, the pressure ratio was increased by 3.76%, which is very close to Shah's finding. For the maximum mass flow capacity, it was increased by 1.4%, which is similar to Shah's finding. Based on this comparison, the accuracy of the map modification model is validated, and is good enough for further investigation.

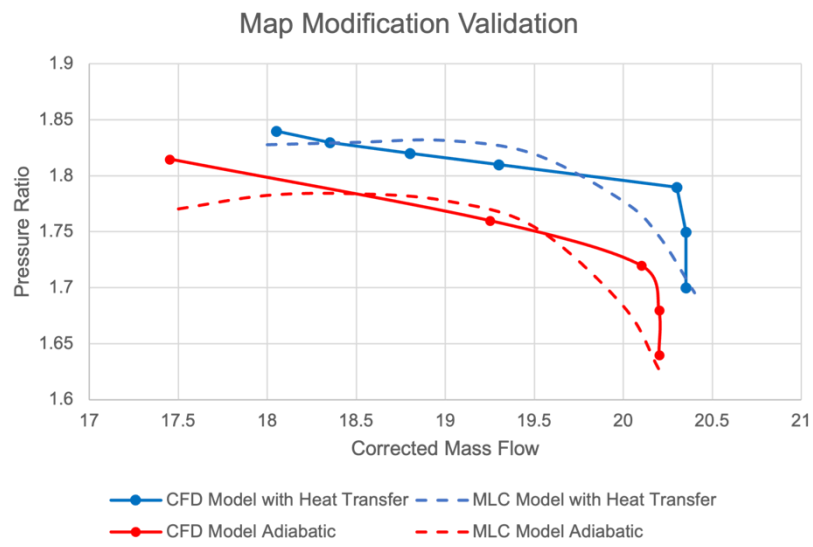


Figure 4-25 Comparison of the heat transfer effect on compressor map

5 Case Study

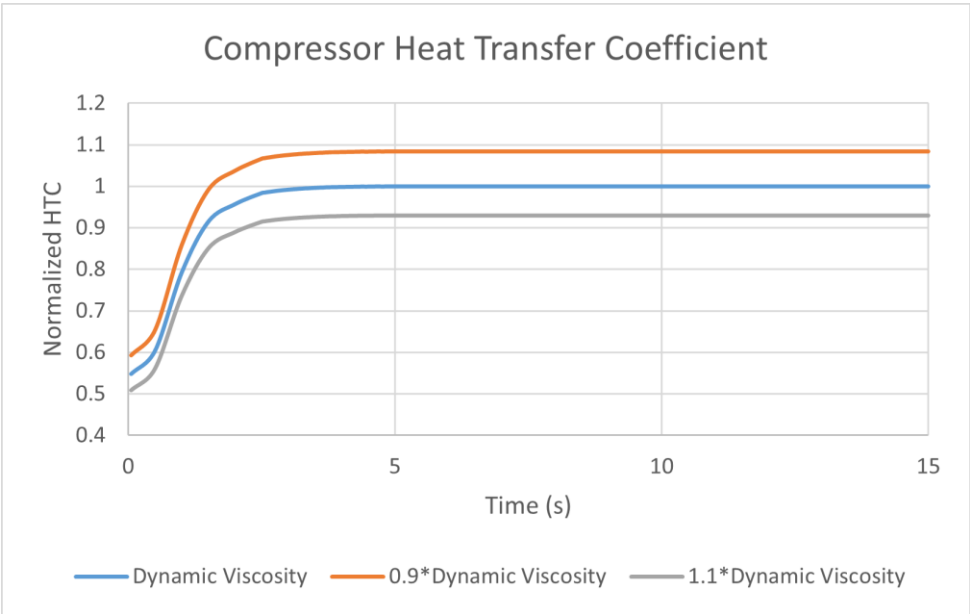
In this Chapter, several case studies based on the proposed heat transfer effect model mentioned above will be discussed.

5.1 Quantifying Heat Flow Rate Sensitivity to Model Inputs

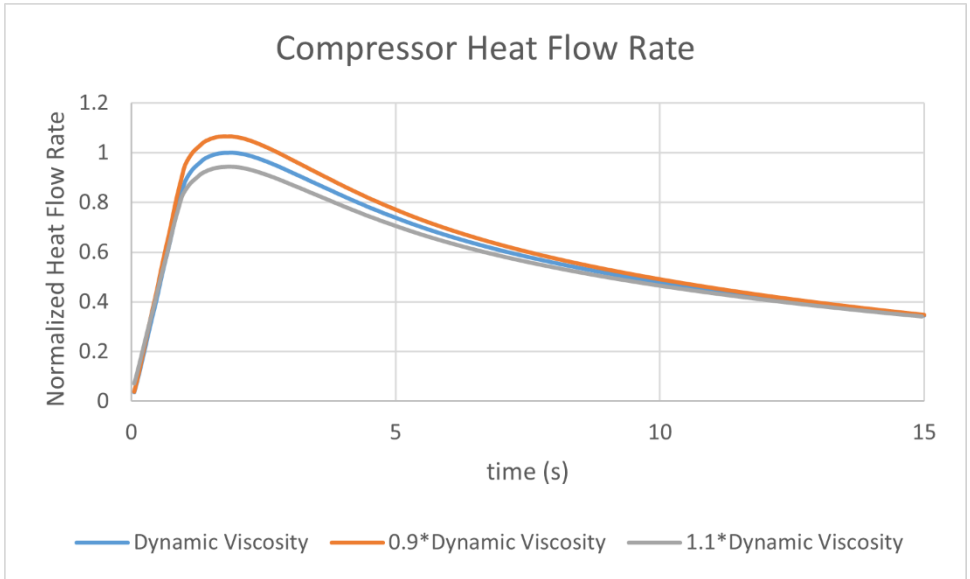
The accuracy of the proposed heat soakage model was compared with GasTurb in Section 4.4. During this comparison, the component performance modification model remains unchanged, but the boundary conditions of the heat soakage model are adjusted to ensure consistent heat transfer processes in both GasTurb and the proposed heat soakage model. Since experimental data is unavailable for comparison due to a lack of necessary data in public domain, it is important to assess the sensitivity of heat flow rates to various model assumptions.

The first assumption involves the estimation of the heat transfer coefficient, which is based on the Nusselt number. Dynamic viscosity plays a critical role in determining the Nusselt number, and it is estimated using Sutherland's law, with reference temperature, Sutherland constant, and reference viscosity all based on air. The reference temperature selected by different researcher varies from 273K to 288K, which lead to approximately 4-7% differences in estimated dynamic viscosity based on temperature. To assess the impact of potential errors in the dynamic viscosity estimation, a variation is applied to the calculated dynamic viscosity. In our case, a small change from -10% to 10% is applied to the estimated dynamic viscosity. The outcome is shown in Figure 5-1 (a) to (d). It was found that a 10% decrease in dynamic viscosity leads to approximately an 8.4% increase in the heat transfer coefficient, while a 10% increase in dynamic viscosity results in about a 7.05% decrease in the heat transfer coefficient. The change in the heat transfer coefficient over time is illustrated using the compressor as an example. Changes in the heat transfer coefficient impact the estimation of heat flow rates, as they directly affect the heat transfer process—a higher heat transfer coefficient results in faster heat transfer, and vice versa. By comparing the heat flow rates in three components, it was observed that a 10% decrease in dynamic viscosity leads to a maximum increase of approximately

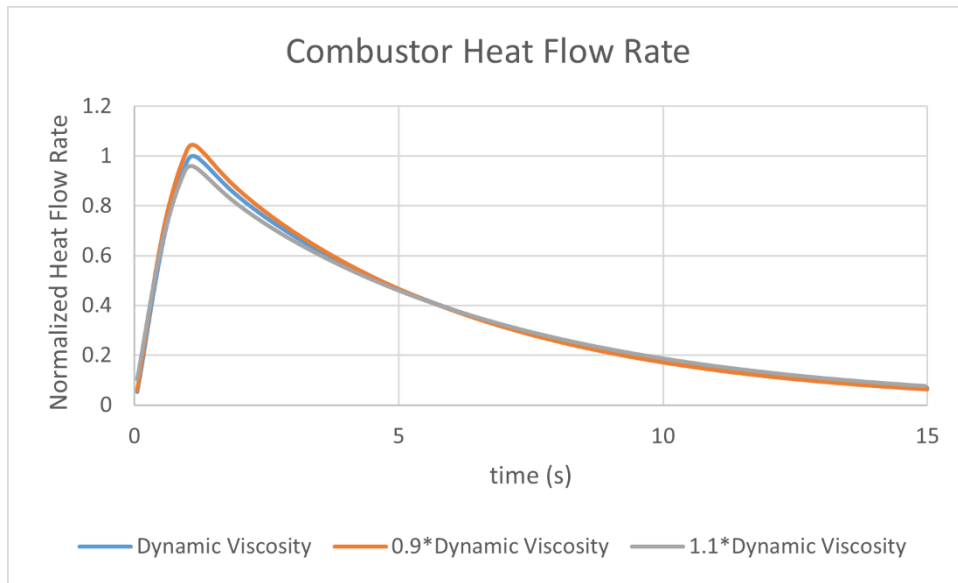
6.5% in compressor heat flow rate, 4% in combustor heat flow rate, and 7% in turbine heat flow rate.



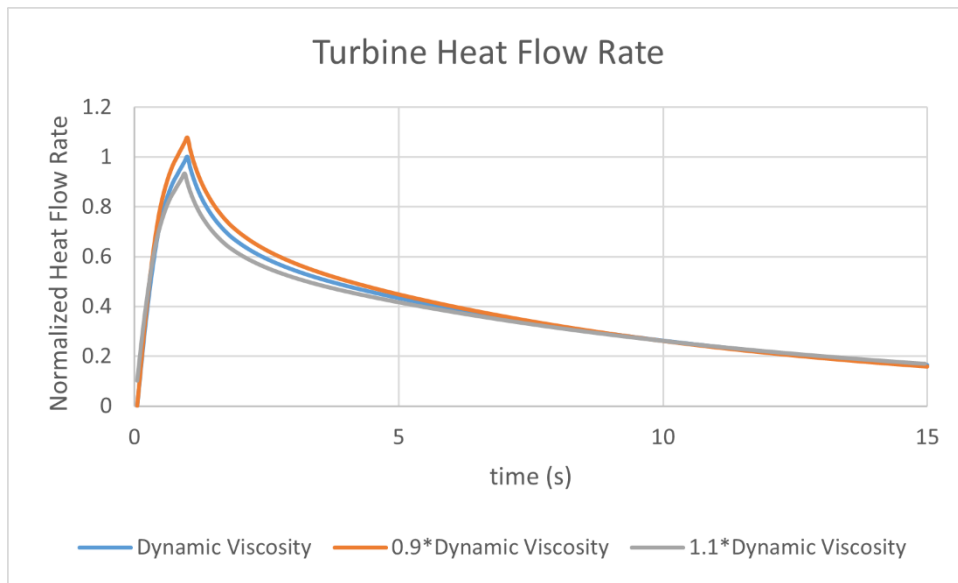
(a)



(b)



(c)

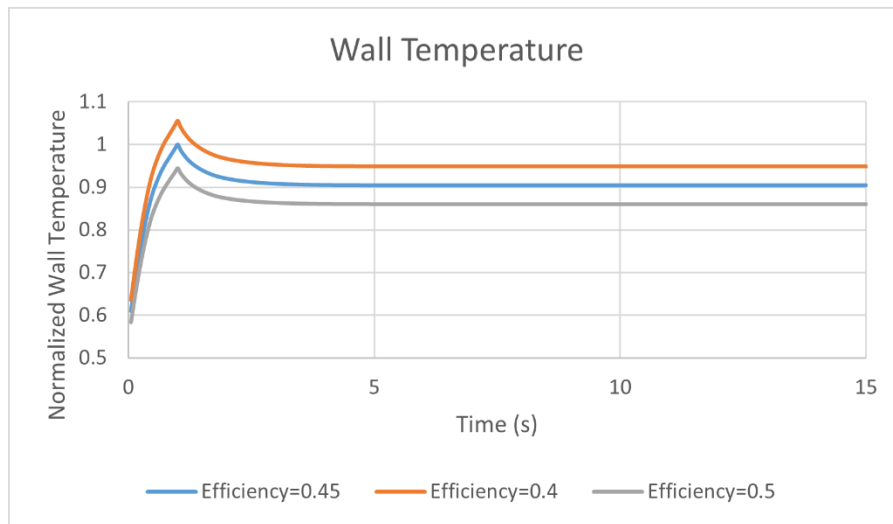


(d)

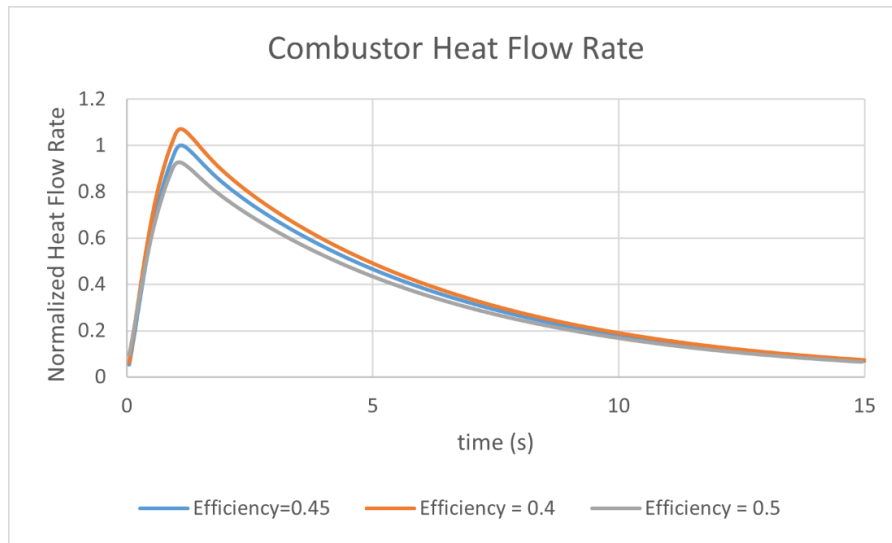
Figure 5-1 Sensitivity analysis of dynamic viscosity on heat flow rate

Another assumption is made during the development of the film cooling model. The wall temperature of the metal is estimated using the film cooling equation, with a typical film cooling efficiency value of 0.45. Variations in film cooling efficiency will alter the wall temperature of the metal, thereby affecting the heat flow rate. To assess the impact of potential errors in selecting the film cooling

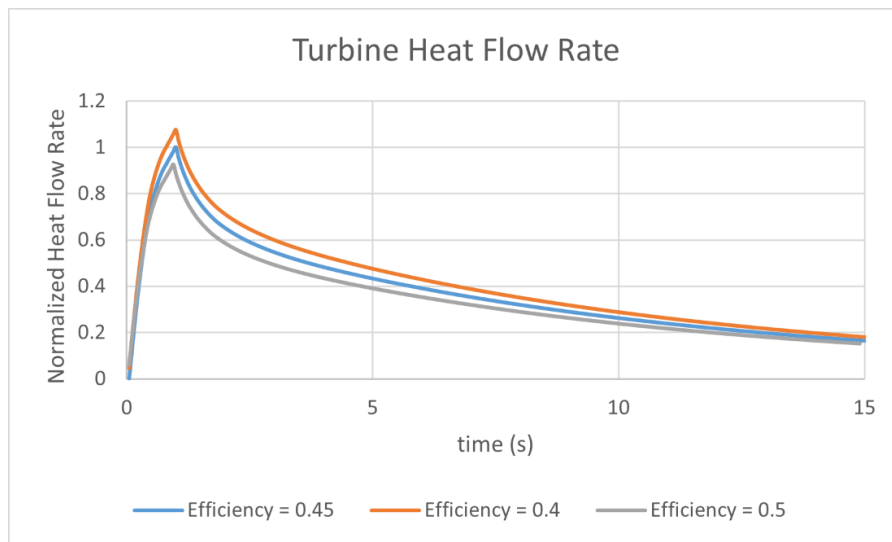
efficiency, a variation is applied to the film cooling efficiency. Since the film cooling efficiency of different cooling design varies with each other, in this case, small change in film cooling efficiency value from 0.4 to 0.5 is investigated. The simulation outcome is shown in Figure 5-2. The magnitude of wall temperature change due to variations in film cooling efficiency is influenced not only by the efficiency value itself but also by the temperature difference between the gas and cooling flow. In the current case, a 0.05 reduction in film cooling efficiency results in a maximum increase of 5.556% in the combustor primary zone wall temperature, 4.62% in the secondary zone, and 4.6% in the dilution zone. For the turbine, the blade wall temperature increases by 3.92%. Conversely, a 0.05 increase in film cooling efficiency leads to a maximum decrease of 5.557% in the combustor primary zone wall temperature, 4.62% in the secondary zone, and 4.61% in the dilution zone, with the turbine blade wall temperature decreasing by 3.92%. This change in wall temperature further affects the heat flow rate. A 0.05 reduction in film cooling efficiency results in a maximum increase of 7% in combustor heat flow rate and 7.5% in turbine heat flow rate. Conversely, a 0.05 increase in film cooling efficiency reduces the maximum combustor heat flow rate by 7.7% and the turbine heat flow rate by 7.5%.



(a)



(b)



(c)

Figure 5-2 Sensitivity analysis of film cooling efficiency on heat flow rate

The thermal barrier coating thickness applied in combustor is assumed to be a typical value of 500 μm . The thickness of thermal barrier coating can vary widely based on the specific design. In our case, a small range from -10% to 10% is added to thermal barrier coating to evaluate the affect from potential error caused by the thickness of thermal barrier coating. The simulation outcome is shown in Figure 5-3. It was found that 10% increase in thermal barrier coating will lead to

4.08% decrease combustor heat flow rate. And 10% decrease in thermal barrier coating will lead to 4.46% increase combustor heat flow rate.

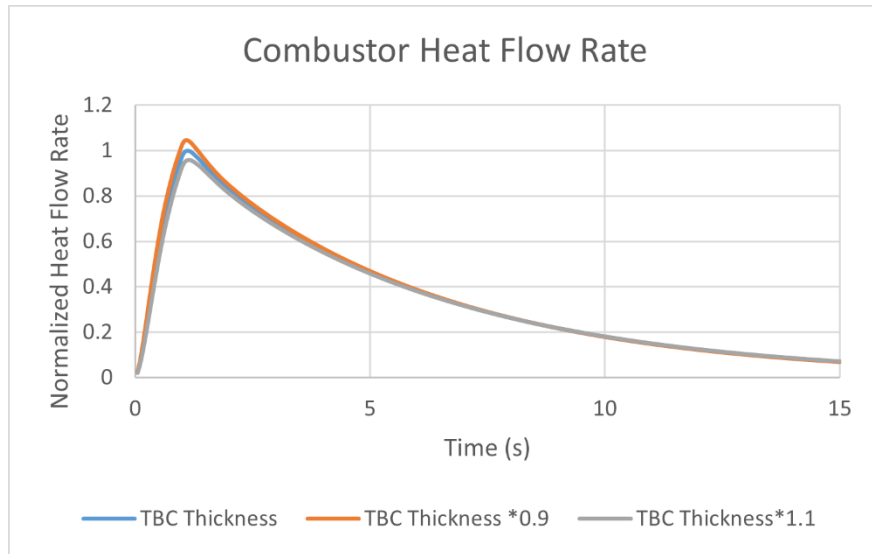
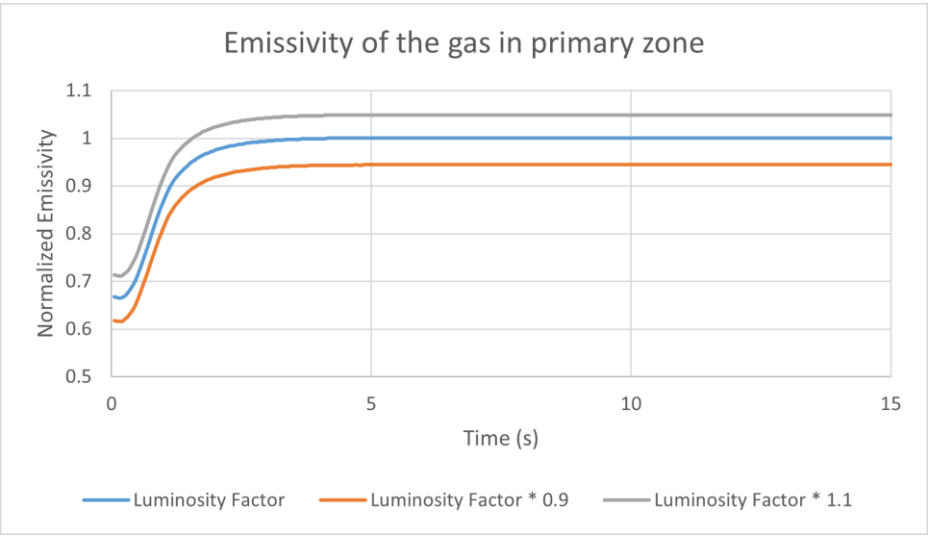


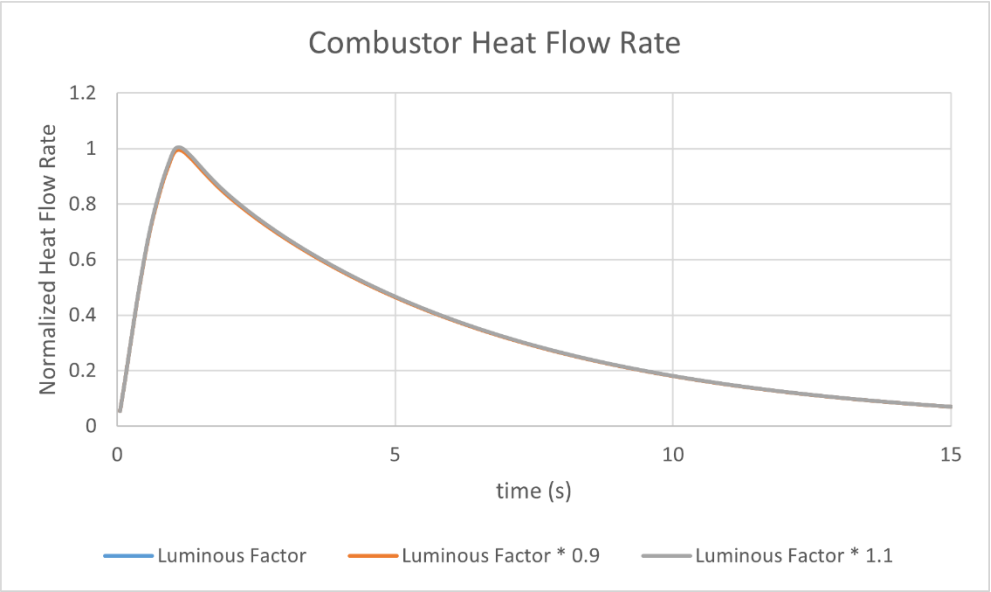
Figure 5-3 Sensitivity analysis of TBC thickness on heat flow rate

The luminosity factor is chosen as a typical value for each combustor zone based on other researcher's finding. For primary zone, the luminosity factor is assumed to be 4. For secondary zone, the luminosity factor is assumed to be 3. For dilution zone, the luminosity factor is assumed to be 2. Cause the luminosity factor value is highly dependent on the combustor design, in our case, a small range from -10% to 10% is added to the luminosity factor to evaluate the effect from potential error caused by luminosity factor setting. Simulation outcome is shown in Figure 5-4. For primary zone, the 10% increase in luminosity factor leads to 4.88% increase in emissivity of gas. The 10% decrease in luminosity factor leads to 5.46% decrease in emissivity of gas. For secondary zone, the 10% increase in luminosity factor leads to 7.83% increase in emissivity of gas. The 10% decrease in luminosity factor leads to 8.44% decrease in emissivity of gas. For primary zone, the 10% increase in luminosity factor leads to 8.09% increase in emissivity of gas. The 10% decrease in luminosity factor leads to 8.42% decrease in emissivity of gas. However, when compare the combustor heat flow rate, 0.44% increase in heat flow rate was found when luminosity factor increased by 10% and 0.47% decrease in heat flow rate was found when luminosity factor decreased by 10%. This is because the heat flow rate caused by radiation is far

smaller than what caused by convention. So small change in radiative heat flow rate will not have a huge impact on the total heat flow rate.



(a)



(b)

Figure 5-4 Sensitivity analysis of luminosity factor on heat flow rate

During radiative heat transfer calculation, the value of emissivity of the wall is assumed to be 0.7, which is based on Nimonic material. However, for stainless material, the value of emissivity of the wall is about 0.8. To evaluate the affect from potential error caused by the choice of emissivity of the wall value, the

current value is changed with the range of 0.6 to 0.8. Based on the investigation, 0.1 increase in the value of emissivity of the wall will lead to 0.26% increase in burner heat flow and 0.1 decrease in the value of emissivity of the wall will lead to 0.27% decrease in burner heat flow rate.

The effect of each assumption on heat transfer coefficient, emissivity of the gas, heat flow rate in compressor, combustor, and turbine is summarized in Table 5-1.

Table 5-1 Sensitivity analysis: impact of assumed variables on estimation outcome

VARIABLE	h	ε_g	q_{compre}	$q_{combustor}$	$q_{turbine}$
μ	◆	N/A	◆	▲	◆
$\eta_{cooling}$	N/A	N/A	N/A	◆	◆
Th_{TBC}	N/A	N/A	N/A	▲	N/A
L	N/A	◆	N/A	–	N/A
ε_w	N/A	N/A	N/A	–	N/A

NB: – Blow 1%, ▲ 1-5%, ◆ 5-10%, ● over 10%

μ Dynamic viscosity, Factor range 0.9-1.1

$\eta_{cooling}$ Cooling efficiency, Value Range 0.4-0.5

Th_{TBC} Thickness of thermal barrier coating, Factor range 0.9-1.1

L Luminosity factor, Factor range 0.9-1.1

ε_w Emissivity of the wall, Value range 0.6-0.8

5.2 Quantifying Combustor Temperature Distribution and Cooling Technologies Effect on Heat Flow Rate and Transient Performance

As described in Section, the combustor temperature distribution, cooling technologies like film cooling, internal cooling and thermal barrier coating are not considered by other researchers during heat soakage analysis. In order to analysis the effect coming from the consideration of combustor temperature distribution, cooling technologies like film cooling, internal cooling and thermal barrier coating, a case study is investigated. Models with the combination of mentioned combustor temperature profile and cooling technologies are compared each other to quantify their impact on heat flow rate estimation and further effect on gas turbine transient performance. Here, the combustor is selected as the investigation object because it include all the features mentioned above.

5.2.1 Effect of combustor temperature profile and cooling technologies on combustor heat flow rate

The simulation and comparison of the combustor temperature profile and different cooling technologies are investigated. The heat flow rate estimations of the combustor are estimated using six combinations:

Combination 1: LPM based model;

Combination 2: Quasi-2D thermal network based model; No cooling technologies; Considering combustor temperature profile (CTD);

Combination 3: Quasi-2D thermal network based model; Only consider internal cooling (IC); Considering combustor temperature profile;

Combination 4: Quasi-2D thermal network based model; Considering internal cooling and TBC; Considering combustor temperature profile;

Combination 5: Quasi-2D thermal network based model; Considering internal cooling and Film Cooling (FC); Considering combustor temperature profile;

Combination 6: Quasi-2D thermal network based model; Considering internal cooling, Film Cooling, and TBC; Considering combustor temperature profile.

The gas turbine model and transient fuel flow is kept the same as what discussed in Section 4.4. The estimated heat flow rate values based on these models are illustrated in Figure 5-5. A noticeable increase in heat flow rate is observed when comparing the results of the combination 1 and combination 2. During the acceleration, the combination 2's peak heat flow rate value is approximately four times that of Model 1. This significant increase in heat flow rate can be attributed to the high temperatures in the primary and secondary zones, leading to a larger temperature difference and subsequent heat flow rate. By implementing the internal cooling, the heat flow rate of combination 3 is 14% lower than combination 2. An additional reduction of 9% heat flow rate is found when adding TBC (combination 4). Implementing film cooling and internal cooling technology proves to be highly effective in reducing the heat flow rate. With a cooling efficiency set at 0.45 and the cooling temperature based on the combustor's inlet temperature, the simulation shows a 40% decrease in peak heat flow rate compared to combination 1. However, the heat flow rate is still 2.3 times higher than that of the combination 1. The application of TBC also contributes to reducing the heat flow rate (as shown in combination 6), owing to its low thermal conductivity. It is important to note that TBC also slows down the heat transfer process and results in a smaller temperature gradient. When film cooling, internal cooling and TBC are combined, a further reduction in heat flow rate is observed (48.97%). Nevertheless, the heat flow rate is still twice as high as that of combination 1. Based on these findings, it becomes evident that considering the combustor temperature profile and incorporating cooling technologies are crucial aspects in heat soakage analysis. Neglecting these factors may lead to inaccurate assessments of heat soakage within the gas turbine components.

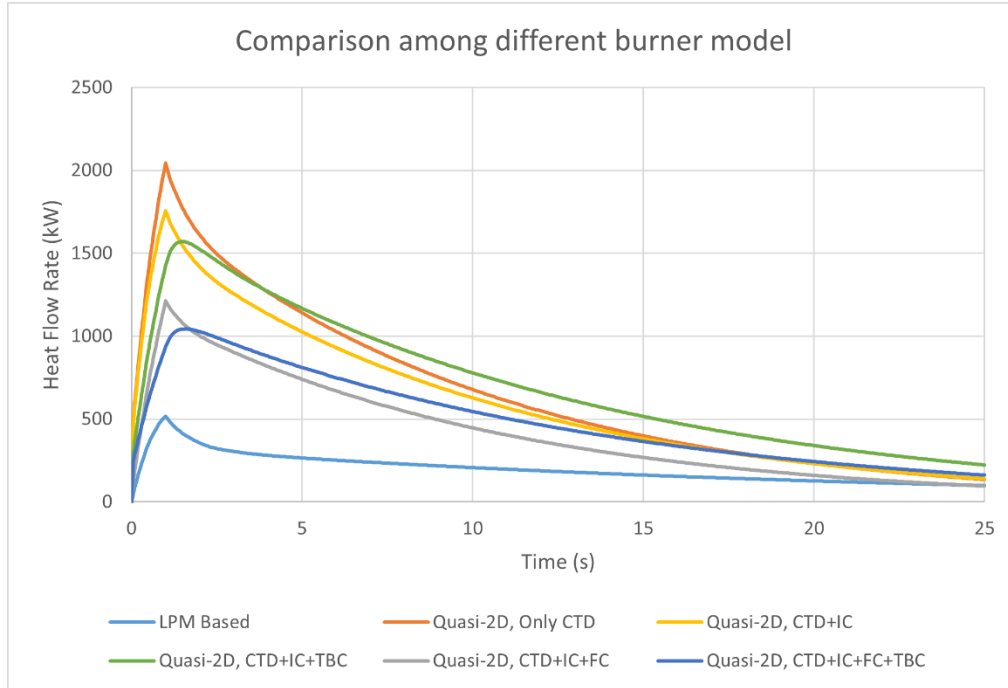


Figure 5-5 Combustor temperature profile distribution and film cooling effect on heat transfer process

5.2.2 Heat flow rate variation effect on gas turbine transient performance

The effect of different combination of combustor temperature distribution, film cooling, internal cooling and thermal barrier coating on combustor heat flow rate is discussed in Section 5.2.1. The further impact of change in combustor heat flow rate on gas turbine transient performance is investigated here.

The combustor outlet temperature during transient manoeuvre based on different combination discussed in Section 5.2.1 is shown in Figure 5-6. The change in heat flow rate will have a direct impact on the combustor outlet temperature. The higher the heat flow rate, the higher the heat loss caused by heat soakage and the higher in temperature reduction. For combination 2, which only consider the combustor temperature distribution, the heat flow rate is about 4 times of what based on 0D lumped parameter method. The maximum combustor temperature is reduced by 31 K. However, for combination 6, which consider the temperature

distribution, film cooling, internal cooling, and thermal barrier coating, the reduction in maximum combustor outlet temperature is 10 K.

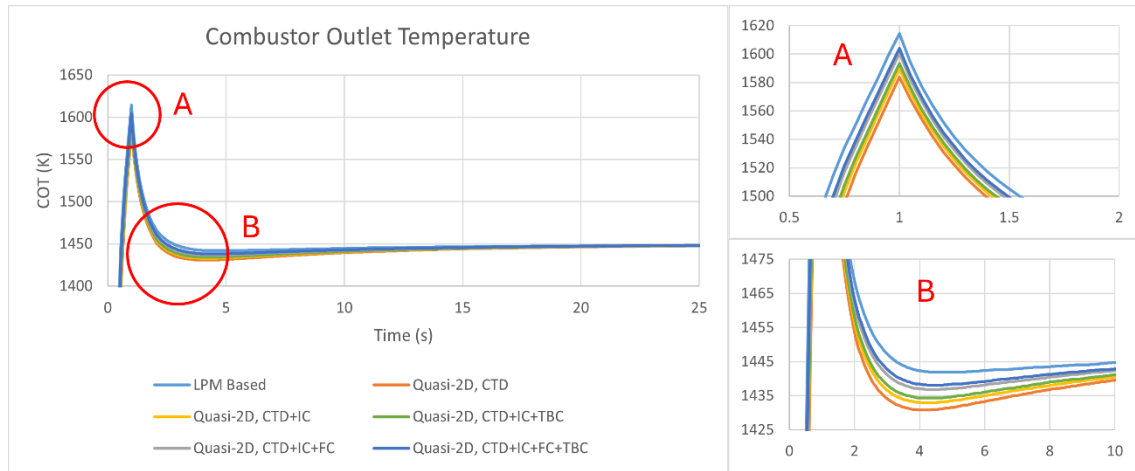
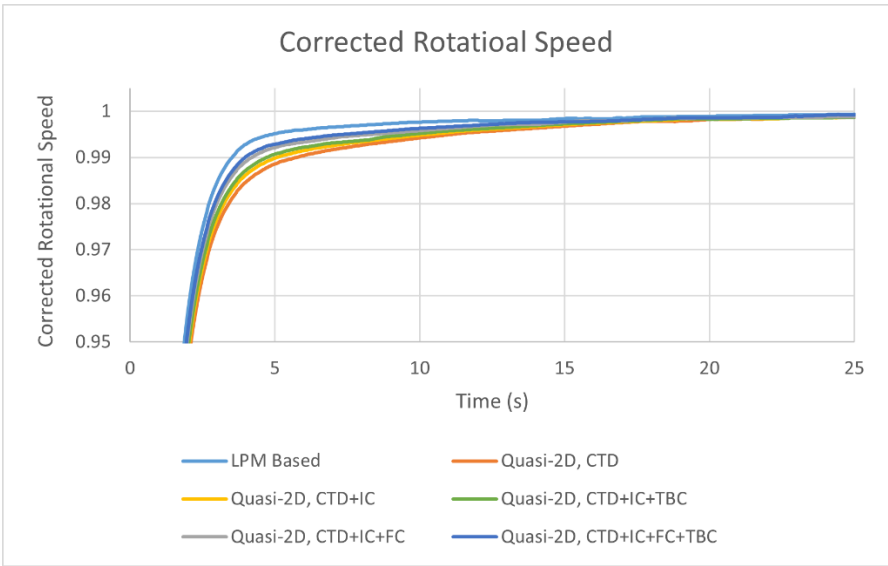


Figure 5-6 Heat flow rate change effect on combustor outlet temperature

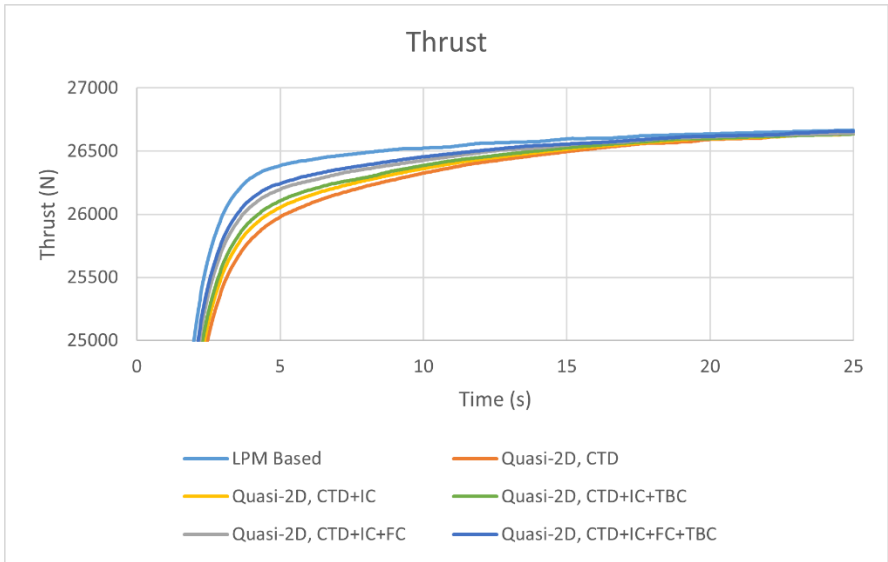
The comparisons of transient performance based on different combination discussed in Section 5.2.1 is shown in Figure 5-7 (a) to (d). Shaft corrected rotational speed, thrust, mass flow, and compressor pressure ratio are investigated.

The corrected rotational speed is shown in Figure 5-7 (a). Based on the corrected rotational speed change rate, the difference in combustor heat flow rate will lead to a different magnitude of delay in shaft response. If we assume gas turbine reach its full working condition when corrected rotational speed reaches 0.995, the time for each combination model varies with each other. For the base line model (combination 1), the time needed is 4.9 seconds. For combination 2, the time to reach full working condition takes 11.1 seconds, which increase the acceleration time by 126%. For combination 6, the time to reach full working condition takes 7.35 seconds, which increase the acceleration time by 50%. The gas turbine thrust is shown in Figure 5-7 (a). Based on the thrust change rate, the difference in combustor heat flow rate will also lead to a different magnitude of delay in thrust response. At simulation time 4.9 seconds, the gas turbine model based on baseline heat soakage model is assumed to reach the full working condition. The thrust level is 26.378 kN. For combination 2, the thrust level at 4.9 second is 25.965 kN, which is 1.56% lower than baseline condition. For

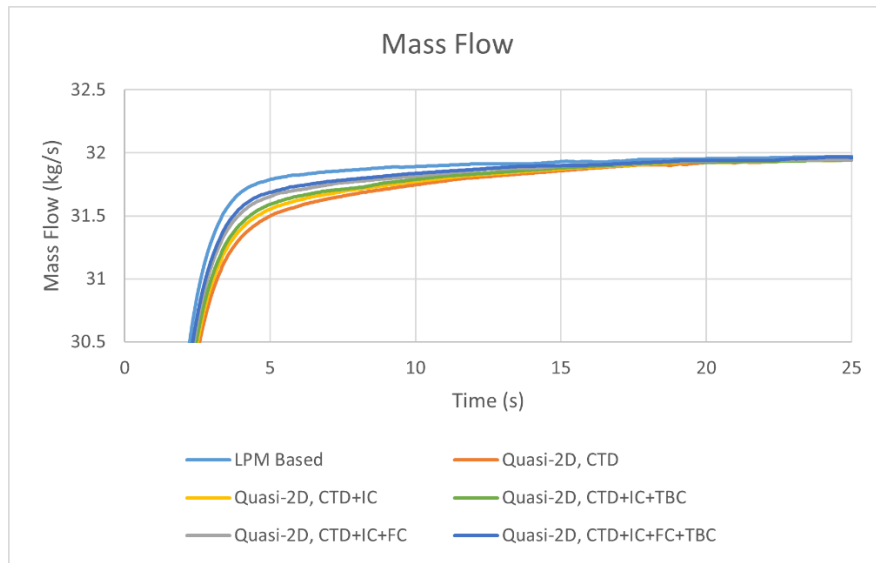
combination 6, the thrust level at 4.9 second is 26.236 kN, which is 0.53% lower than baseline condition. The gas turbine mass flow is shown in Figure 5-7 (c). Based on the comparison with baseline model, the mass flow rate based on combination 2 is 1% lower at 4.9 second. The compressor pressure ratio is shown in Figure 5-7 (d). The pressure ratio of combination 2 is 1.2% lower than based line condition at 4.9 second.



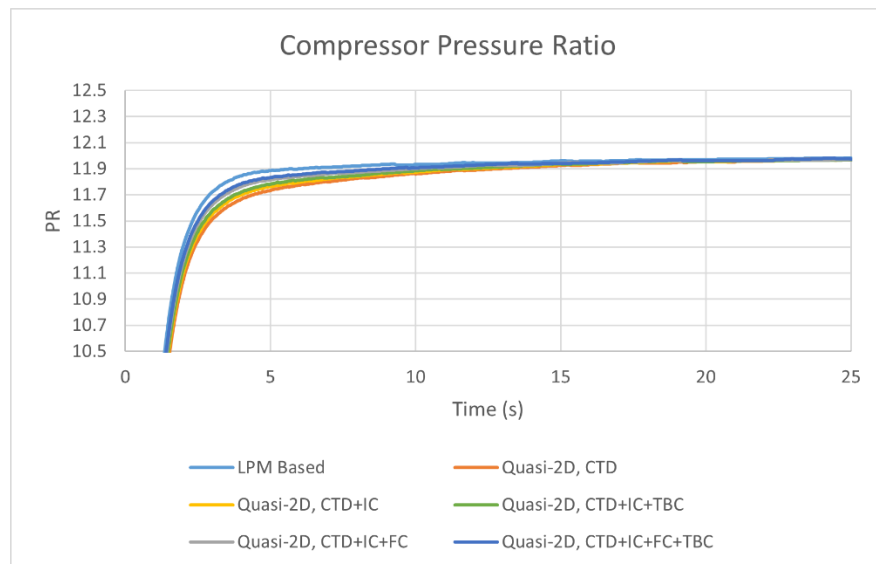
(a)



(b)



(c)



(d)

Figure 5-7 Heat flow change effect on gas turbine transient performance

Based on the discussion, it can be found that the consideration of combustor temperature distribution, film cooling, internal cooling and thermal barrier coating is important during the heat soakage effect investigation because they will change the combustor heat flow rate significantly and further effect the gas turbine transient performance. Change in heat flow rate in combustor will lead to a different magnitude of delay in gas turbine response. Compared with the

conventional heat soakage mode, the proposed heat soakage model can estimate the heat flow rate in combustor during engine transients in a more realistic way. By incorporating the combustor temperature profile and cooling technologies, a more accurate representation of the heat transfer process is achieved, resulting in a notable variation in heat flow rate estimation. The observed disparities in gas turbine transient performance emphasize the significance of including these features in contemporary heat soakage models. The novel heat soakage model offers an improved understanding of heat transfer dynamics and its impact on engine performance during transients.

5.3 Heat Soakage and Tip Clearance Effect on Gas Turbine Transient Performance

In order to evaluate the heat soakage and tip clearance effect on gas turbine transient performance, one case study is investigated based on the gas turbine model with similar performance as the CFM 56-7B turbofan engine. In this section, the gas turbine engine model, geometry identification outcome, heat soakage effect and tip clearance change effect on gas turbine transient performance will be discussed.

5.3.1 Gas Turbine Engine Model and Design Point Parameters

To have a better understanding of the heat soakage effect and tip clearance change effect on gas turbine transient performance, advanced civil aircraft engine is selected as the investigation target. A gas turbine engine model with the same architecture as CFM 56-7B is developed. The cross section of CFM 56-7B is shown in Figure 5-8. The engine is a two-spool turbofan engine with a single stage fan, 3-stage booster, 9-stage high pressure compressor, single stage high pressure turbine and a 4-stage low pressure turbine. The combustor is an annular combustor.

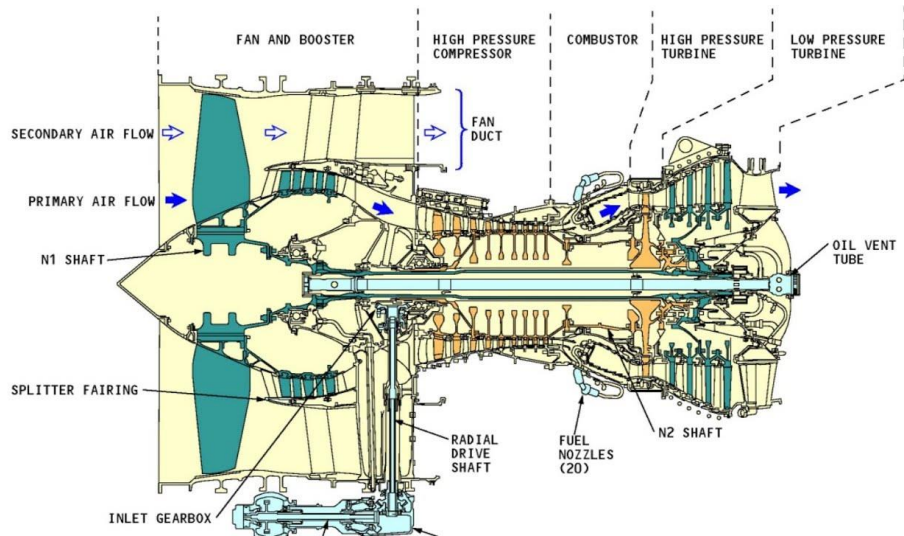


Figure 5-8 Cross section of CFM 56-7B turbofan engine [166]

The design point data for the established two spool turbofan engine model is listed in Table 5-2.

Table 5-2 Design point data for the two-spool turbofan engine

Parameters	Value
Bypass Ratio	5.1
Corrected Mass Flow (kg/s)	680
Fan Pressure Ratio	1.7
Fan Efficiency (%)	88.5
Booster Pressure Ratio	1.794
Booster Efficiency (%)	0.87
Compressor - Pressure Ratio	10.155
Compressor Efficiency (%)	0.87
Combustor Outlet Temperature (K)	1498
Combustor Efficiency (%)	99.9

Combustor Pressure Loss (%)	5
HPT Efficiency (%)	92
LPT Efficiency (%)	91

5.3.2 Engine Transient manoeuvre Parameter

The transient simulation of the two-spool turbofan engine is considered. For control purposes, the fuel schedule was set as a function of time, and the internal component volume method is used. For the rotor moment of inertia and the internal component volume, they are listed in Table 5-3. The transient manoeuvre is selected as an acceleration manoeuvre. It takes 2s to reach design point (HP PCN = 1) fuel flow form 1kg/s, and the total transient time is set as 150s. The fuel flow schedule is shown in Figure 5-9.

Table 5-3 Rotor moment of inertia and the internal component volume parameter

Parameters	Value
High Pressure Spool Moment of Inertia ($\text{kg}\cdot\text{m}^2$)	12.5
Low Pressure Spool Moment of Inertia ($\text{kg}\cdot\text{m}^2$)	146.8
Fan Volume (m^3)	1.5
Booster Volume (m^3)	0.1
High Pressure Compressor Volume (m^3)	0.5
Combustor Volume (m^3)	0.05
High Pressure Turbine Volume (m^3)	0.1
Low Pressure Turbine Volume (m^3)	0.15

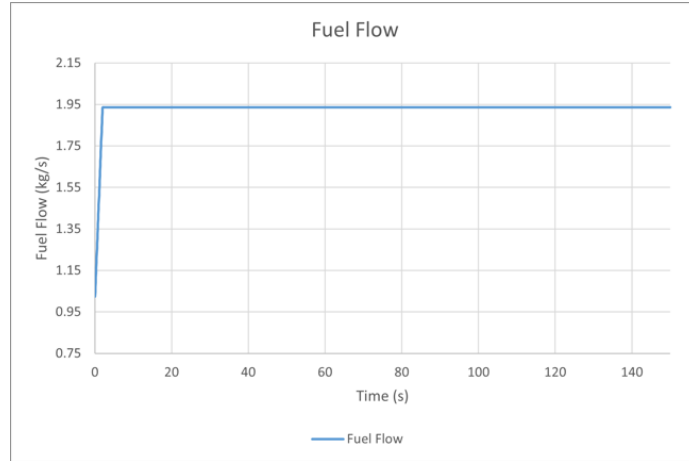


Figure 5-9 Fuel flow schedule for the two spool turbofan engine

5.3.3 Geometry Identification of the Aircraft Engine Model

Based on the geometry identification model developed above, the design point gas turbine component parameters and the geometry data are used as the input. The outcome of the geometry of each component is shown in Figure 5-10.

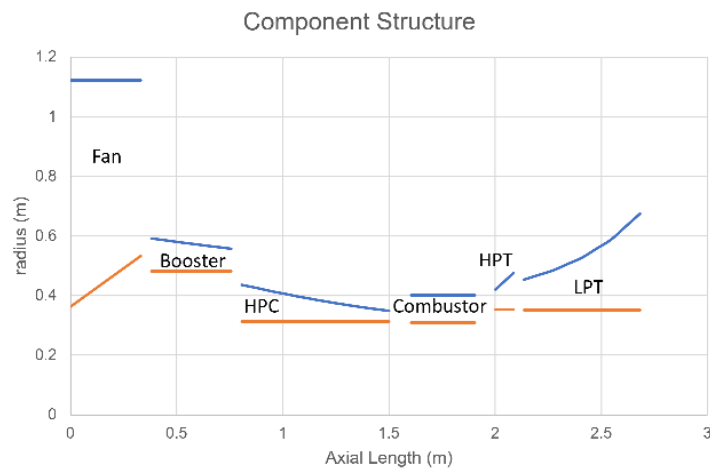


Figure 5-10 Component flow path geometry outcome

The wetted area and mass for each component parts are shown in Table 5-4, where the R and S represent the rotor and stator. For the stator of compressor and turbine, the disc mass and wetted area are set as 0.

Table 5-4 Component mass and wetted area outcome

Fan	R1	S1								
Blade Mass	424.19	262.85								
Blade Area	18.84	15.04								
Casing Mass	93.32	72.41								
Casing Area	2.33	1.81								
Disc Mass	461.77									
Disc Area	0.93									
Booster	R1	S1	R2	S2	R3	S3				
Blade Mass	18.31	15.733	13.89	11.99	10.62	9.34				
Blade Area	1.46	1.347	1.276	1.173	1.11	1.03				
Casing Mass	10.85	10.03	9.24	8.599	7.969	7.44				
Casing Area	0.27	0.25	0.23	0.214	0.199	0.186				
Disc Mass	66.59		58.063		51.025					
Disc Area	0.222		0.193		0.17					
HPC	R1	S1	R2	S2	R3	S3	R4	S4	R5	S5
Blade Mass	17.7	14.11	11.29	9.093	7.452	6.139	5.069	4.254	3.57	3.017
Blade Area	1.375	1.21	1.089	0.964	0.877	0.788	0.716	0.652	0.599	0.546
Casing Mass	7.049	6.1739	5.36	4.7624	4.199	3.77	3.37	3.059	2.76	2.525
Casing Area	0.176	0.1543	0.134	0.119	0.1049	0.09437	0.0842	0.07648	0.06901	0.06314
Disc Mass	52.58		42.35		34.716		28.89		24.36	
Disc Area	0.124		0.1		0.0824		0.0686		0.0578	
HPC	R6	S6	R7	S7	R8	S8	R9	S9		
Blade Mass	2.573	2.202	1.902	1.6468	1.4273	1.2484	1.092	0.9588		

Blade Area	0.506	0.465	0.434	0.402	0.374	0.349	0.327	0.305		
Casing Mass	2.298	2.118	1.942	1.799	1.66	1.547	1.435	1.343		
Casing Area	0.05747	0.05295	0.04855	0.04499	0.0415	0.0386	0.03588	0.03357		
Disc Mass	20.77		17.89		15.55		13.63			
Disc Area	0.04933		0.04249		0.03694		0.03236			
Combustor										
Mass	62.19									
Area	1.55									
HPT	S1	R1								
Blade Mass	12.52	21.1								
Blade Area	1.231	2.075								
Casing Mass	19.079	21.407								
Casing Area	0.2384	0.2675								
Disc Mass		256.83								
Disc Area		0.187								
LPT	S1	R1	S2	R2	S3	R3	S4	R4		
Blade Mass	12.08	14.144	16.428	19.36	22.948	27.5648	33.31	40.73		
Blade Area	1.473	1.7252	2.0038	2.361	2.799	3.362	4.06	4.968		
Casing Mass	15.835	16.36	16.994	17.72	18.61	19.64	20.907	22.39		
Casing Area	0.1979	0.2045	0.212	0.221	0.232	0.2455	0.2613	0.2799		
Disc Mass		98.977		98.977		98.977		98.977		
Disc Area		0.1506		0.1506		0.1506		0.1506		

5.3.4 Heat Flow Rate and Tip Clearance Change

Based on the fuel flow schedule described in Figure 5-9, the transient simulation has been investigated. The heat flow rate in each component are shown in Figure 5-11. It can be found that the highest heat flow occurs in the LPT. This is due to the relative high temperature and the high component mass and wetted area. Although combustor and HPT experiences the highest temperature, the combustor's mass and wetted area is small and there is only one stage in HPT. Also, the cooling technologies applied to combustor and HPT reduce the heat flow rate. Which further decrease the heat flow. For the fan, booster and HPC, their heat flow rate is small is due to the low temperature. The fan's heat flow is even higher than the booster's heat flow. This is due to the extremely high mass and wetted area in Fan blade.

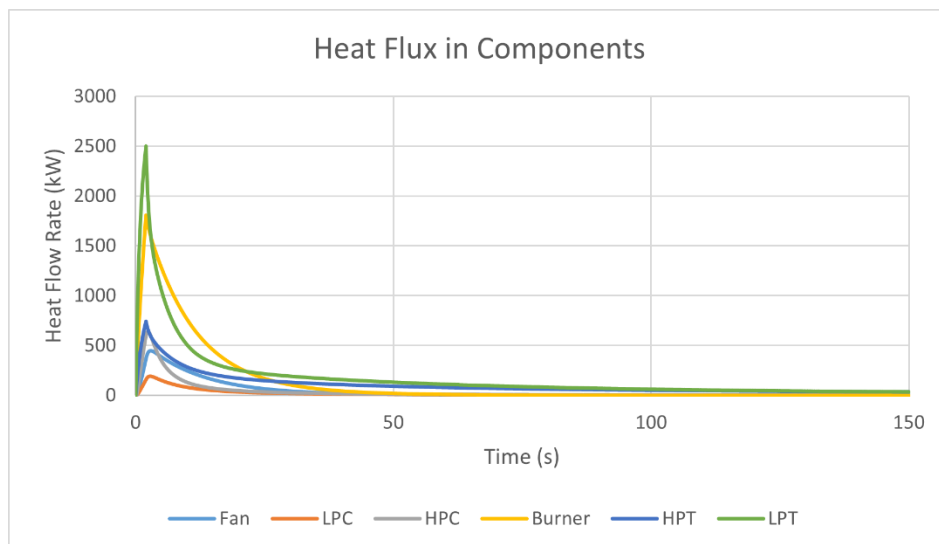


Figure 5-11 Heat flow rate in several components

The root means square tip clearance change and efficiency scalar for each component are shown in (a) to (e). The detailed tip clearance changes for every stages are listed in 7.8Appendix E.

The fan tip clearance change is shown in Figure 5-12 (a) and (b). it can be found that the initial tip clearance is higher than 1%. This is because the fan tip clearance is dominated by the fan blade length. At the beginning of transient, the blade mechanical displacement is lower than the datum condition due to low

rotational speed, leading to an increase in tip clearance and a decrease in fan efficiency. When engine accelerating, the rotational speed increases and the mechanical displacement of fan respond quick, leading to a fast decrease in tip clearance. In the fan, the stage 2 and stage 3 of tip clearance change is not obvious cause the heat transfer process is low in fan. It is mainly affected by the mechanical displacement coming from change in rotational speed.

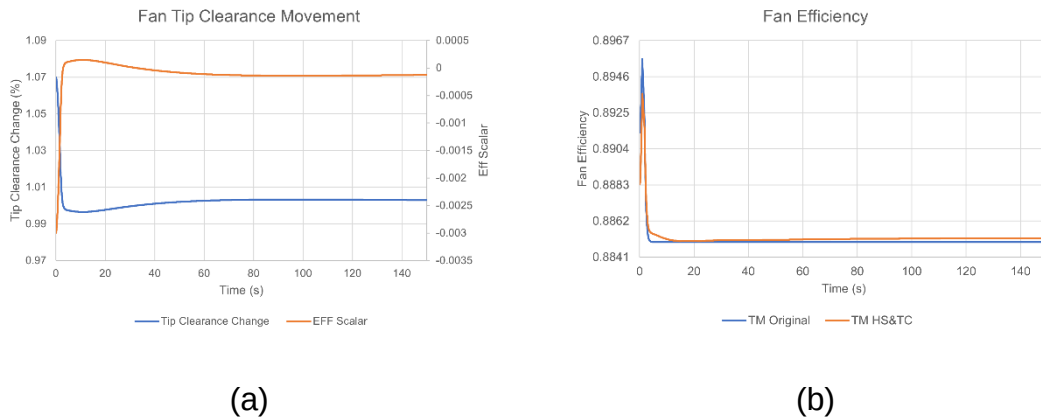
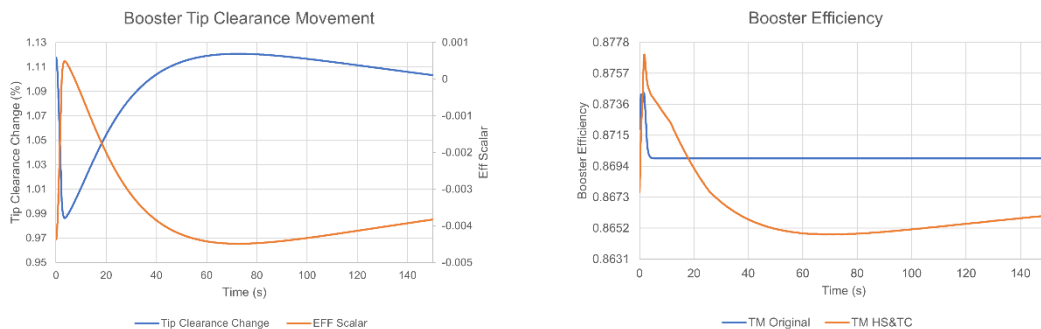


Figure 5-12 Fan tip clearance change and efficiency change

The booster tip clearance change is shown in Figure 5-13 (a) and (b). It can be found the initial tip clearance is still higher than 1%. Different from the fan, the first stage of booster is dominated by the mechanical displacement of the disc cause the blade length is relative shorter. Compared with the fan, booster will have a clearer stage 2 and stage 3. In stage 2, the thermal displacement of casing will dominant. For stage 3, the disc thermal displacement will dominant. However, due to the long disc radius, the conductive heat transfer in disc is slow and after 150s it has not reach the equilibrium condition.

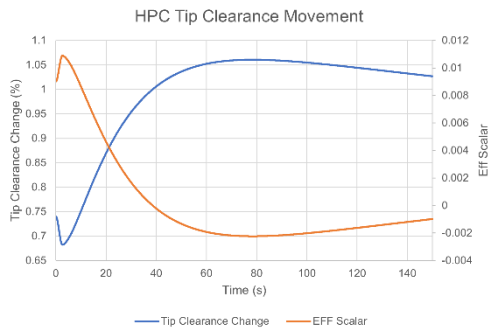


(a)

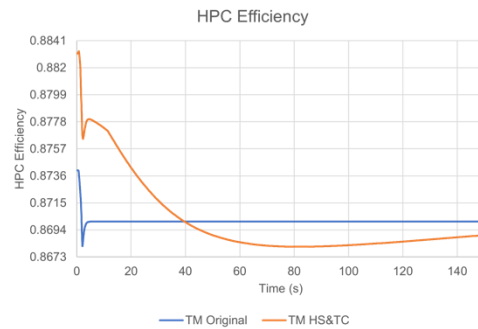
(b)

Figure 5-13 Booster tip clearance change and efficiency change

The HPC tip clearance change is shown in Figure 5-14 (a) to (b). It can be found that the initial tip clearance is smaller than 1%. This is due to that the casing thermal displacement dominant the initial condition. Because in HPC, blade length is short, and disc radius is small, the mechanical displacements of blade and disc are small. However, the thermal resistant in casing is smaller than disc so casing will have a more sensitive response to temperature change. So, in the beginning, the thermal displacement of casing is greater, especially in the rare stage of HPC. That is the reason why initial tip clearance is smaller than 1%. During the acceleration, the change in stage 1 is small due to smaller change in mechanical displacement. Stage 2 is large due to the dominant of casing thermal expansion.



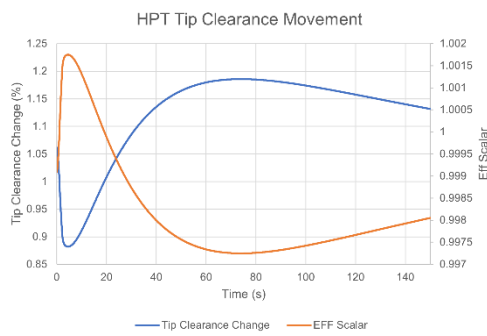
(a)



(b)

Figure 5-14 HPC tip clearance change and efficiency change

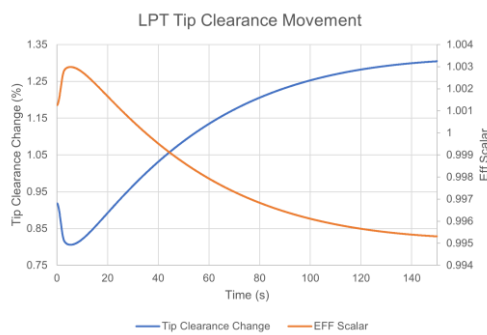
The HPT and LPT tip clearance change is shown in Figure 5-15 (a) to (d). For the HPT a standard change in tip clearance can be observed. However, stage 3 for LPT is not observed. This is due to the slow casing thermal displacement. The temperature in LPT is lower than that in HPT, leading to a slower heat transfer. At 150s, the casing reach its equilibrium condition. That's why stage 3 is not found in this transient manoeuvre.



(a)



(b)



(c)



(d)

Figure 5-15 HPT and LPT tip clearance change and efficiency change

5.3.5 Heat Soakage and Tip Clearance Change Effect on Transient Performance

Based on the estimated heat transfer rate and tip clearance change, the transient performance of the gas turbine engine is investigated. Three models are considered: the transient manoeuvre without heat soakage and tip clearance change effect, the transient manoeuvre with heat soakage effect, and the transient manoeuvre with heat soakage and tip clearance change effect.

The low and high pressure spools' corrected rotational speed are compared with each other first, as shown in Figure 5-16. The blue line represents the corrected rotational speed change without heat transfer effect. The yellow line represents the corrected rotational speed change with heat soakage effect. The orange line represents the corrected rotational speed change with heat soakage and tip clearance change effect. From the yellow line and blue line in Figure 5-16, it can

be found that heat soakage causes a delay in LP spool CN change rate. This is due to the heat loss during the acceleration. However, the HP spool CN is higher than the original one. This is caused by the temperature drop at the inlet of HPC. When considering the tip clearance change effect, a different trend will be found. in the first 20 seconds, the LP and HP spools' CN are higher than the original one. This is caused by the change in component efficiency. As shown in Section 5.3.4, the tip clearance will experience a reduction stage in the beginning due to the mechanical displacement. An increase in component efficiency is found. Since the fuel schedule is kept the constant, the surplus power of each spool will increase and lead to an increase in CN change rate. However, when tip clearance moves into stage 2 and 3, the efficiency of component decrease to a lower value than design point condition, and the CN will come into a value lower than the design point condition. In this case, a maximum 0.11% increase in CN is found in LP spool in the initial 20s. and a -0.256% decrease is found after 20s transient simulation. For the HP CN, CN increases by 1.36% in initial 20s, and decreases -0.4% after 20s.

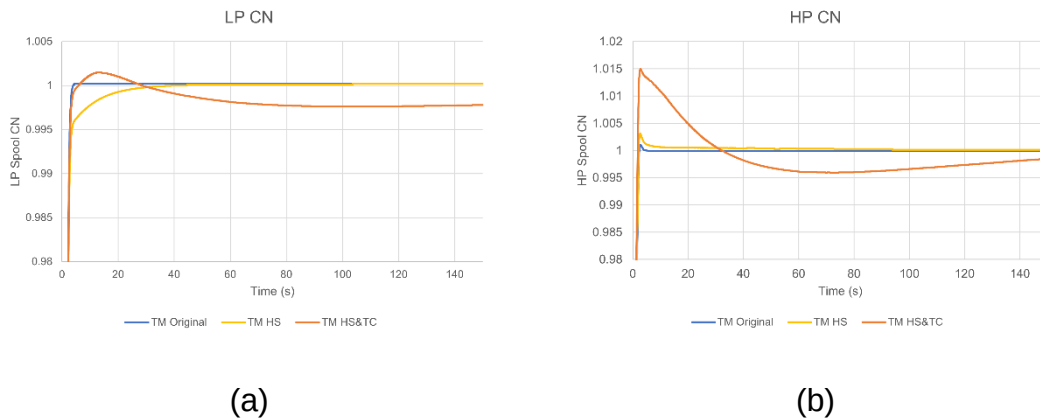


Figure 5-16 Spool corrected rotational speed change

The heat soakage and tip clearance change effects on turbofan thrust and SFC are shown in Figure 5-17. When only consider the heat soakage effect on gas turbine transient performance, the maximum difference between the original transient and heat soakage effect is -2.37%. When considering the tip clearance effect, the thrust difference is reduced to 1.81% in the initial 20s. However, the thrust was decreased by 0.67% after 60s. The SFC is affected by the change in

thrust when the fuel flow is kept the same. In the beginning of the acceleration, an increase in SFC is found due to the decrease in thrust. After 60s, the SFC of TC& HS will be higher caused the decrease in thrust.

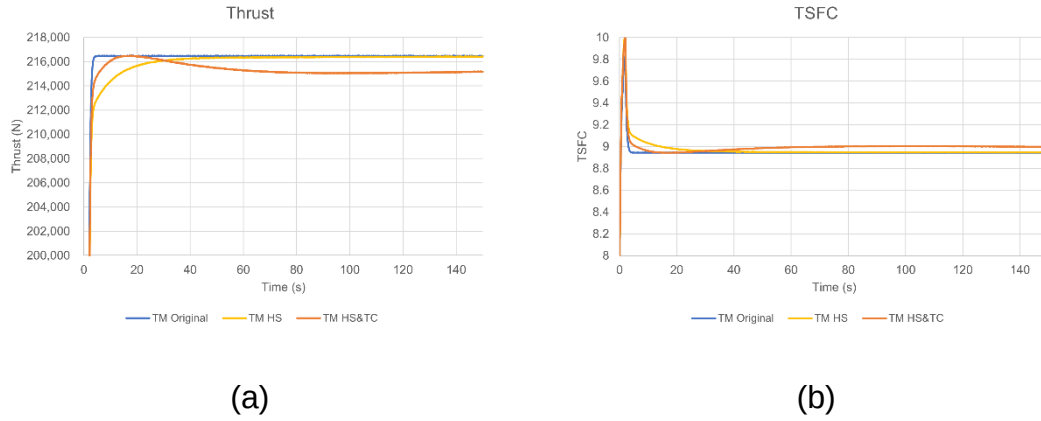


Figure 5-17 Engine thrust and SFC change

Except the gas turbine performance. It can be found that tip clearance change will affect the engine performance longer than the heat soakage effect. In this case study, the heat soakage effect is negligible after 40s. However, the tip clearance effect last to the end of the transient manoeuvre. This reveals the importance of considering heat soakage and tip clearance effect during the gas turbine transient simulation.

5.4 Map Modification Case Study

The proposed compressor map modification model has been applied to the high-pressure compressor with similar parameter as NASA Energy Efficiency Engine (E3). In this section, the mean line model outcome, heat transfer effect on compressor map, and numerical correlations will be discussed.

5.4.1 Mean Line Model Outcome

The targeted compressor is a 10-stage compressor with the pressure ratio of 25 under 100% rotational speed. The design point of the compressor is selected at the maximum climb condition, and the inlet total temperature and the inlet total pressure of the compressor is 304.4K and 59641.8Pa respectively. The detailed compressor design point information is listed in Table 5-5. Moreover, the variable

inlet guide vane and stator is applied to the mean line model based on the schedule in Figure 5-18. The mean line model simulation outcome is shown in Figure 5-19, where the red line represents the NASA experiment data and blue dashed line represents the mean line model outcome. It can be found that the mean line model has a good agreement with the NASA experiment data. The maximum difference in mass flow occurred under 95% speed line and the difference is 3%. Based on this finding, the mean line model is good enough for our further research.

Table 5-5 Parameters of NASA E3 Engine High Pressure Compressor

Parameter	Value
Design Point Pressure Ratio	25
Number of Stage	10
Rotational Speed (rpm)	12303
Corrected Mass Flow (kg/s)	54.4
Inlet Total Temperature (K)	304.4
Inlet Total Pressure (Pa)	59641.8
Isentropic Efficiency (%)	85.7

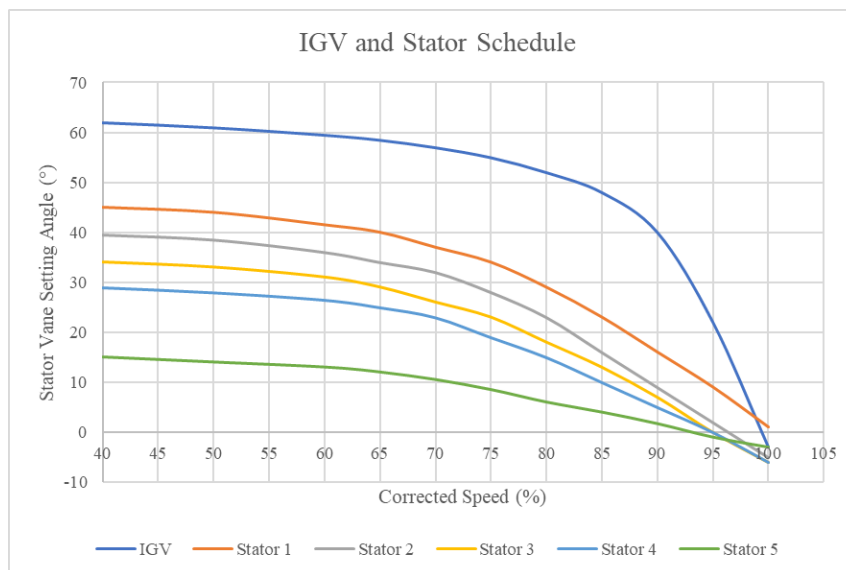


Figure 5-18 E3 Engine HPC IGV and Stator Schedule

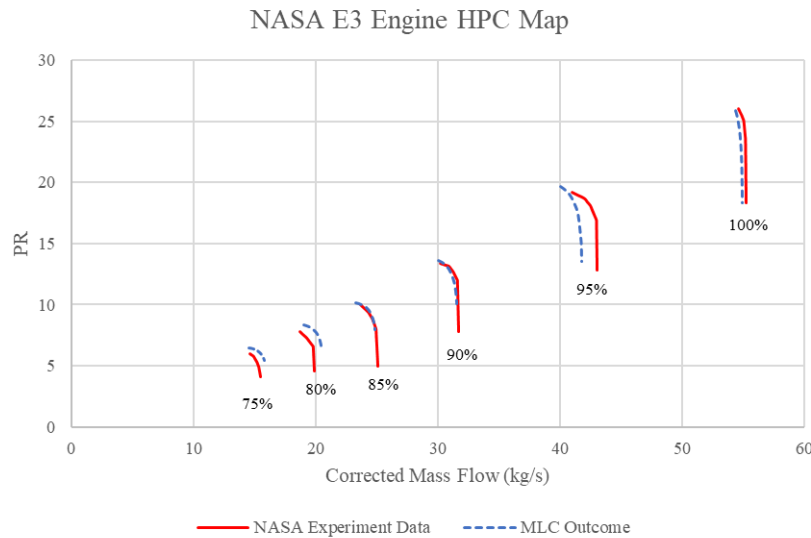


Figure 5-19 E3 Engine HPC Mean Line Model Outcome

5.4.2 NASA E3 HPC Map Modification

In order to capture the distribution of the normalized heat transfer rate, a transient simulation with the thermal network described in 3.3.2 is launched first. The time averaged q^* in each station is shown in Figure 5-20. It can be found that the q^* distribution is not average among stages, which is different from Shah and Kiss's assumption. The front stage will have a higher heat transfer process due to its high metal mass and wetted area compared with rear stages. Also, the rotor q^* will be higher than stator q^* . It is because the rotor has the disc part, which will increase the heat transfer.

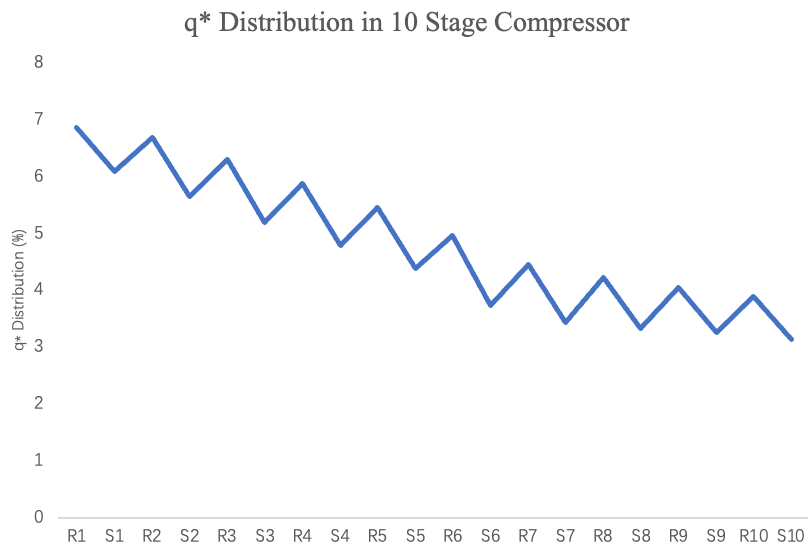


Figure 5-20 Normalized heat transfer rate distribution in the compressor mean line model

With the normalized heat transfer rate distribution, the movement of each speed line under various total q^* were studied, as shown Figure 5-21. When q^* is lower than zero, heat was extracted from the gas path fluid. the speed line moves to higher corrected flows due to mis-matching. The speed line also shifts to higher pressure ratios due to the decreased loss and reduction in flow turning from decreased deviation. The opposite effect occurs when q^* is higher than zero.

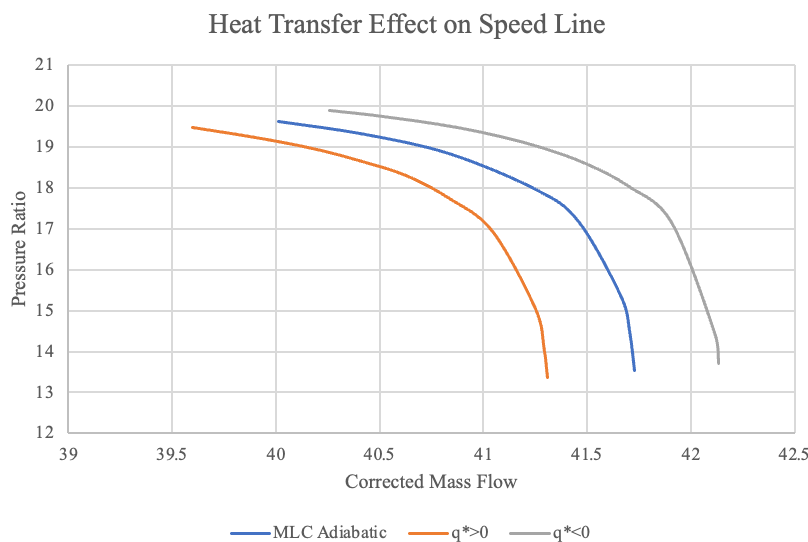
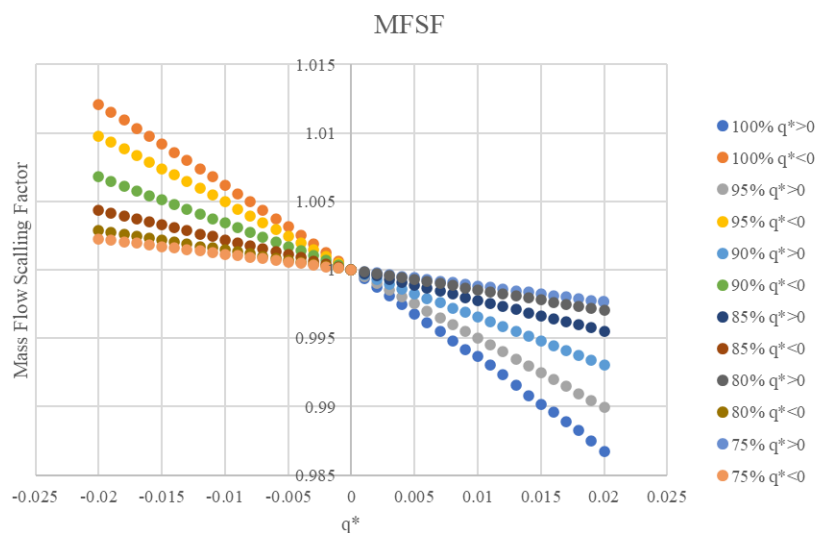
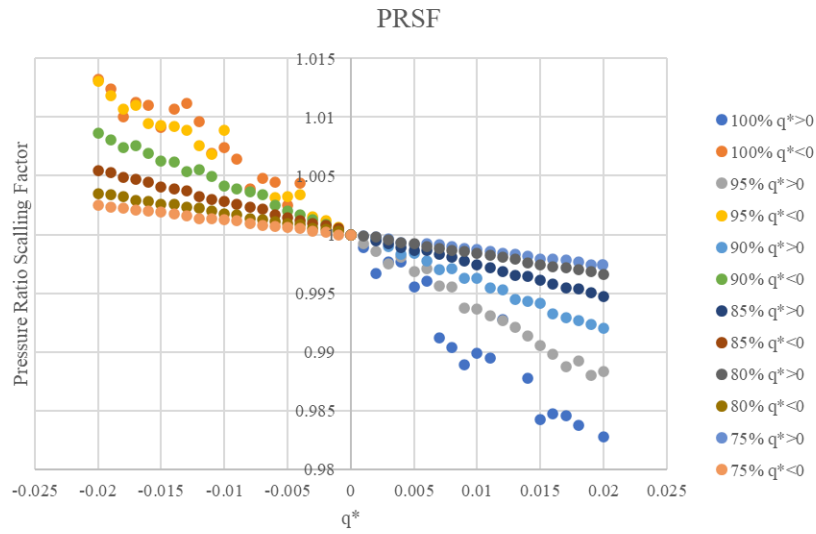


Figure 5-21 Heat Transfer Effect on Speed Line

Since the movement of the speed line is estimated, the change in pressure ratio and corrected mass flow can be calculated by using the scaling factor idea described in Section 3.5.3. The scaling factor outcome for corrected mass flow and pressure ratio under different q^* are shown in Figure 5-22. Although the pressure ratio scaling factor for 100% and 95% speed lines are not smooth, a liner relation between the nondimensional heat flow rate and mass flow and pressure ratio scaling factor is found. Besides, with the reduction in rotational speed, the slope of each scaling factor line is reducing. The reason why pressure ratio scaling factor for 100% and 95% speed lines are not smooth is caused by the convergence of the mean line code. When compressor enter the choking area at high rotational speed, a very small change in mass flow rate will lead to a great change in pressure ratio. Since the mean line code was applied during the investigation, the accuracy of mean line code might cause the non-smooth outcome. Moreover, when the speed line reduced, the relative Mach number through the compressor is reduced, which lead to a smaller deviation angle change and pressure loss. Under the same nondimensional heat flow rate, the change in mass flow and pressure ratio at low rotational speed line will be smaller than what at high rotational speed line. This explains the reduction in the slope of each scaling factor line when speed line reduced.



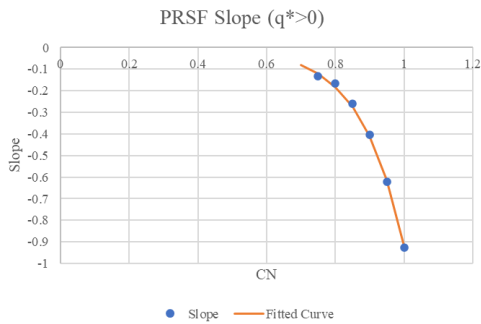
(a)



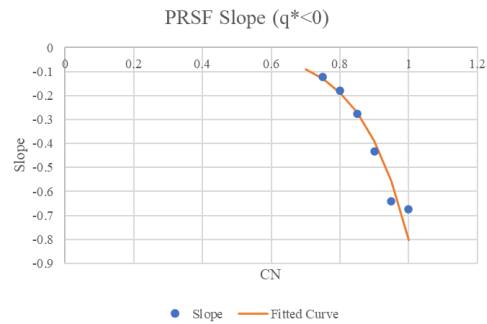
(b)

Figure 5-22 Scaling factor for mass flow and pressure ratio under various q^*

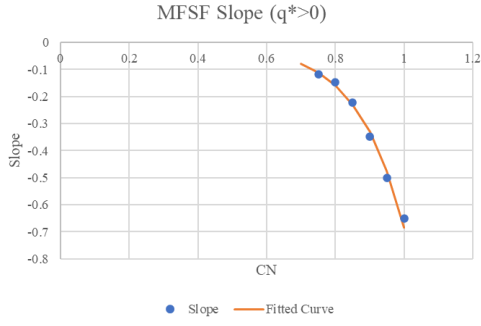
After capturing the slope of each scaling factor line, an exponential relationship between scaling factor and CN are found based on Figure 5-23. The blue dots represent the slope for different CN and the orange line is the fitted curve.



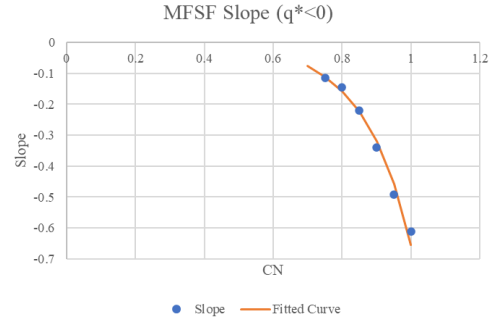
(a)



(b)



(c)



(d)

Figure 5-23 Relation between slope of scaling factor lines and rotational speed

The raised numerical correlation between the q^* and SF is shown by Eqs. (5-1) and (5-2):

$$SF_{PR} = \begin{cases} (-0.00029 \cdot e^{8.0591 \cdot CN}) \cdot q^* + 1 & (q^* \geq 0) \\ (-0.00055 \cdot e^{7.2863 \cdot CN}) \cdot q^* + 1 & (q^* < 0) \end{cases} \quad (5-1)$$

$$SF_{MF} = \begin{cases} (-0.00048 \cdot e^{7.2638 \cdot CN}) \cdot q^* + 1 & (q^* \geq 0) \\ (-0.00052 \cdot e^{7.138 \cdot CN}) \cdot q^* + 1 & (q^* < 0) \end{cases} \quad (5-2)$$

5.4.3 Transient Performance Comparison

The developed component map modification method has been integrated into Turbomatch to provide further insight into the map modification effect on gas turbine transient performance.

5.4.3.1 Engine Structure and Design Point Parameters

A two-spool turbofan engine model similar to NASA E3 engine was developed in TM. The structure of the gas turbine engine is shown in Figure 5-24. The specific engine data and performance parameters are listed in Table 5-6.

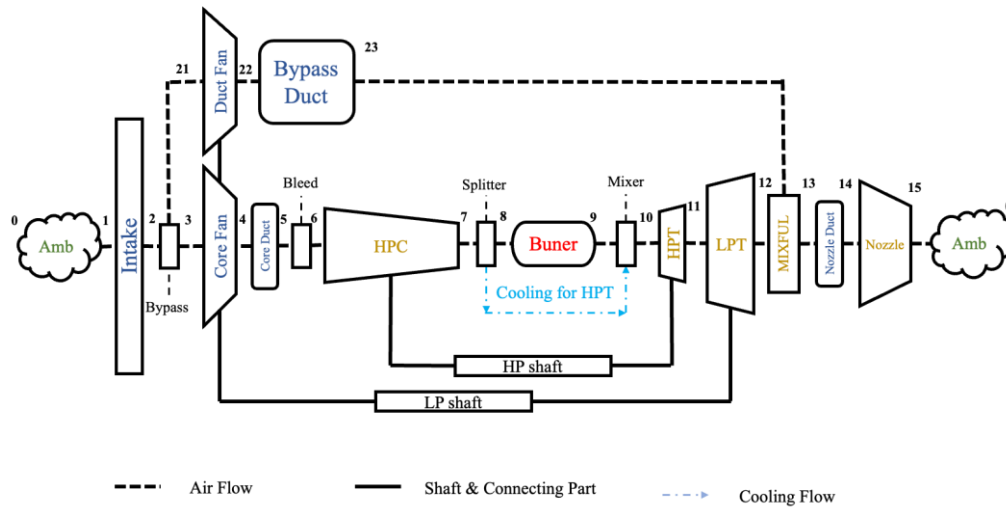


Figure 5-24 Schematic of the two-spool turbofan engine

Table 5-6 two spool turbofan engine parameters (ISA+10, 10668m, 0.8 Mach)

Parameters	Value
Net Thrust (Uninstalled) (kN)	40.211
TSFC (Uninstalled) (g/kN/s)	15.0278
Bypass Ratio	6.7
Overall Pressure Ratio	38.4
Corrected Mass Flow (kg/s)	646
Fan - Duct - Pressure Ratio	1.68
Fan - Core - Pressure Ratio	1.7
Fan - Duct - Efficiency (%)	88.8
Fan - Core - Efficiency (%)	90.5
Fan - Duct - Pressure Loss (%)	1.7
Fan - Core - Pressure Loss (%)	1.4
Compressor Inlet Temperature (K)	304.8

Compressor Inlet Pressure (N/m ² or kPa)	60469
Compressor Corrected Inlet Mass Flow (kg/s)	54.4
Compressor - Pressure Ratio	23.1
Compressor – Isentropic Efficiency (%)	0.86
CPD Nonchargeable Cooling (%W25)	16.15
Combustor Efficiency (%)	99.9
Combustor Pressure Loss (%)	5
HPT Efficiency (%)	92.7
LPT Efficiency (%)	92.7

5.4.3.2 Engine Transient manoeuvre Parameters

The transient simulation of the two-spool turbofan engine is considered. For control purposes, the fuel schedule was set as a function of time, and the internal component volume method is used. Additionally, component performance maps are captured from NASA report. The transient manoeuvre is selected as the hot reslam or Bodie manoeuvre. The engine the engine is held at full power condition in the beginning, a slam deceleration to approach condition is followed by a reslam back to full power. The fuel schedule is illustrated in Figure 5-25. The entire transient manoeuvre lasts for 30 seconds.

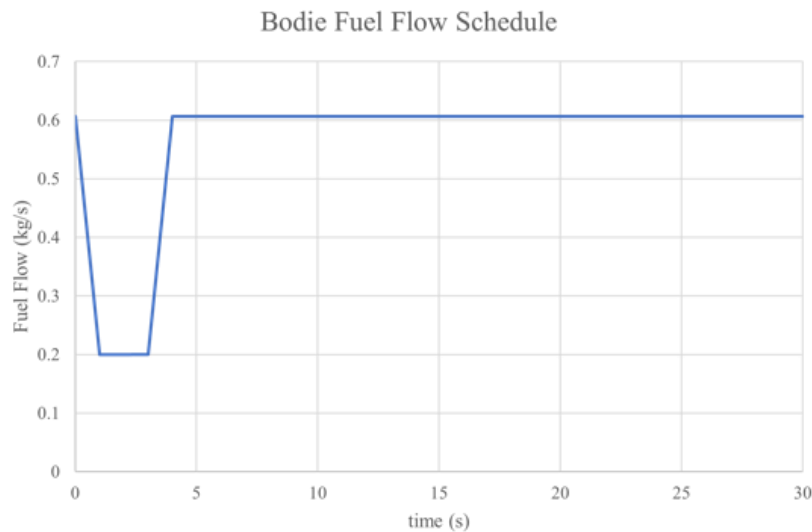


Figure 5-25 Transient Fuel Schedule for Two Spool Turbofan Engine Transient Performance

5.4.3.3 Engine Transient Performance with and without map modification

Based on the two-spool turbofan engine model and transient fuel flow schedule, the transient simulation with and without map modification are simulated. The change of non-dimensional heat transfer rate is illustrated in Figure 5-26. During the slam deceleration, the metal temperature drop rate is lower than the fluid temperature drop rate. This led to a heat transfer from the hot metal to the cold fluid and causing a heating effect. During the reslam acceleration, the temperature of the fluid rise quickly and the temperature difference between the metal and fluid drops. This led to a decrease in the heat transfer rate. After engine reach the full power condition, the metal temperature is lower than the fluid temperature and cause a negative value on the heat transfer rate. The high-pressure compressor working lines with and without map modification were compared with each other and is shown in Figure 5-27. At the end of deceleration, the working point moves toward left. During the re-accelerating, the working line shifts and cause a reduction in surge margin. When engine reach the full power condition, the working point moves right. In this case, the pressure ratio and corrected mass flow were reduced by 0.7% and 0.9% at the end of deceleration

due to the map modification effect. The surge margin was reduced by 4% during the reacceleration. When engine reach the full power condition, the pressure ratio and corrected mass flow were increased by 0.4% and 0.5%.

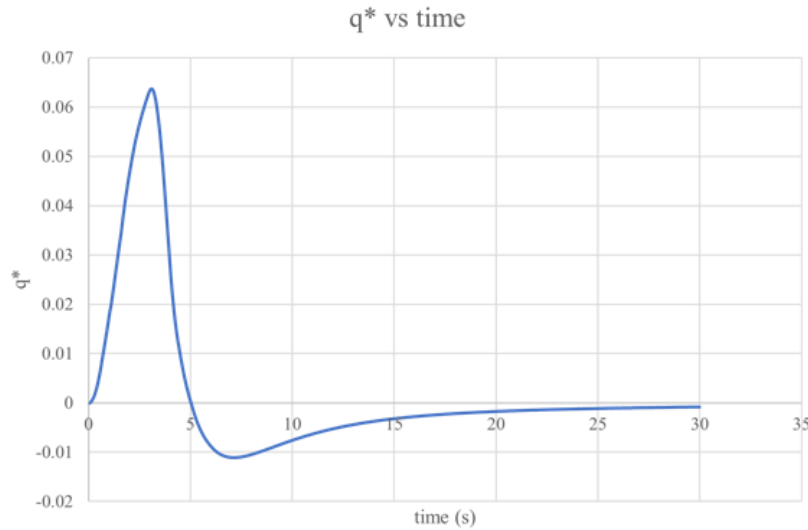


Figure 5-26 Non-dimensional heat transfer rate during transient manoeuvre

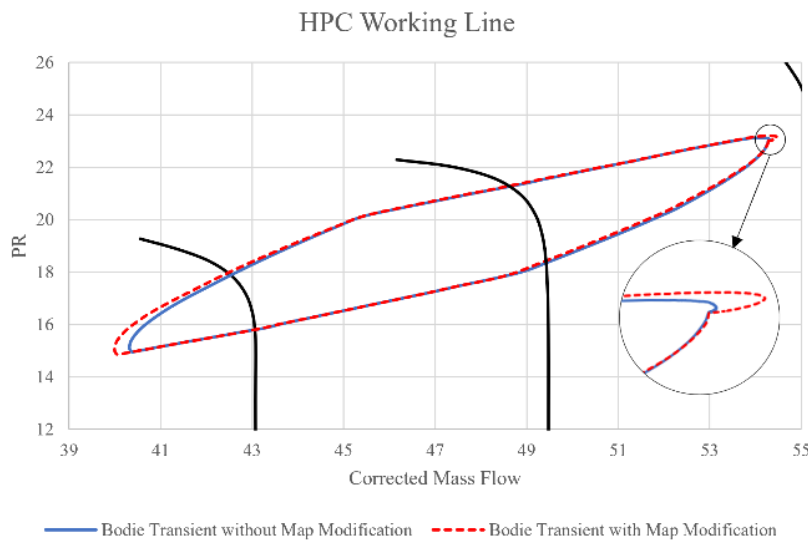


Figure 5-27 High pressure compressor working line comparison

In Kiss's investigation, the reduction in stall margin was around 8%. Compared with this research, the surge margin is smaller than Kiss's finding. One possible reason is the selection of the transient manoeuvre. In current case study, the change in CN is from 90% to 100%. The range is smaller than Kiss's

investigation, which is from 78% to 100%. However, the case study still captures the phenomena caused by the map changing during the transient manoeuvre. The change in pressure ratio, corrected mass flow and surge margin might be bigger in a more severe transient manoeuvre.

6 Conclusion

The development of a comprehensive heat transfer effect model for gas turbine transient performance modification as well as investigation of heat transfer effect on gas turbine transient performance has been presented here.

The developed heat transfer effect model includes 4 major parts. The first is the component geometry identification model, which provides a preliminary design for the compressor, combustor, and turbine, estimating component geometry, metal mass, and wetted area for each blade, casing, disc, and flame tube. The second part is the heat soakage model, which establishes a quasi-2D thermal network based on the geometry identification model's output and estimates the heat flow rate between the main gas and component metal. The third component is the tip clearance model, which predicts the movement of each compressor and turbine stage and adjusts component efficiency accordingly. Finally, the compressor map modification model predicts the movement of the compressor speed line using a non-adiabatic mean line model. Numerical correlations are derived to modify the adiabatic compressor map into a non-adiabatic one using map scaling. Together, these parts enable a systematic analysis of the heat transfer effect on gas turbine transient performance.

During the establishment of heat transfer model, a novel quasi-2D thermal network based heat soakage model and a novel compressor map modification model are developed. The heat soakage model accounts for combustor temperature distribution, film cooling, internal cooling, and thermal barrier coatings. By incorporating these features, the developed heat soakage model improves the current state-of-the-art work by making the heat transfer process more closely resemble real-world conditions. For the compressor map modification model, a new model is introduced. The investigation revealed a linear relationship between the movement of the compressor speed line and the non-dimensional heat flow rate. Further, by applying the concept of a scaling factor, an exponential relationship was identified between the pressure ratio scaling factor, the mass flow scaling factor, and the slope of the linear relationship between the compressor speed line movement and the non-dimensional heat

flow rate. These findings allow the changes in the compressor to be represented as a function of the scaling factor and non-dimensional heat flow rate. This method improves the flexibility of the current state-of-the-art compressor map modification models while maintaining accuracy.

The developed heat transfer model has been successfully integrated to Turbomatch, which improves Turbomatch's capability on simulating heat transfer effect on gas turbine performance during transient manoeuvre.

To evaluate the accuracy of the proposed model, comparisons are made using data from public domain, simulation software, and experimental results. For the component geometry identification model, the simulation outcomes are first compared with actual compressor geometry. Due to limitations in compressor configuration, the maximum difference in stage radius can reach up to 10%. In the absence of detailed data on mass and wetted area, the proposed model is further compared with simulation results from GasTurb using the same input data, with a maximum difference of within 5%. For the heat soakage model, validation is performed against GasTurb simulation results due to a lack of public domain validation materials. The boundary conditions of the proposed heat soakage model are controlled to ensure consistency in the heat transfer process. The maximum difference in heat flow rate is within 8%, and the maximum difference in gas turbine transient performance is within 2.5%. The tip clearance model is compared with publicly available data, showing a consistent trend in tip clearance movement. For the compressor map modification model, the simulated adiabatic and non-adiabatic compressor maps are compared with NASA experimental data and a high-fidelity model, with differences within 6%.

Since the proposed heat soakage model is simplified during validation, a sensitivity analysis is performed to examine the effect of small changes in assumptions on the estimated outcomes. The simulation time for initial temperature identification does not impact the estimated heat flow rate in the compressor, combustor, and turbine. A 10% change in dynamic viscosity results in approximately an 8% change in the heat transfer coefficient, about a 7% change in compressor and turbine heat flow rates, and around a 4% change in

combustor heat flow rate. A 10% change in cooling efficiency leads to approximately a 5% change in wall temperature and about a 7% change in both combustor and turbine heat flow rates. A 10% change in the luminosity factor results in approximately an 8% change in the emissivity of the gas and a 0.5% change in combustor heat flow rate, affecting convective heat transfer within the combustor. Finally, a 10% change in the emissivity of the wall leads to an approximate 0.45% change in combustor heat flow rate.

Based on the proposed heat transfer model, three case studies have been done to investigate the heat transfer effect on gas turbine transient model.

The first case study consist of the comparison between the proposed novel quasi-2D thermal network with the traditional LPM. It is found that:

1. The combustor temperature profile model can offer a more realistic heat flow rate estimation in the combustor. The findings revealed a significant increase (four times that of the LPM based model) in heat flow rate within the combustor due to the high temperature in the primary zone and secondary zone.
2. Cooling technologies implementation effectively decreases the rate at which heat flows. In comparison to the combustor temperature profile model: internal cooling reduces the heat flow rate by 14%; internal cooling and TBC reduce the heat flow by 23%; film cooling and internal cooling reduces the heat flow rate by 40%; film cooling, internal cooling and TBC reduces it by 48.97%. However, even with these measures, the heat flow rate remains twice as high as the LPM model.
3. The proposed heat soakage model provided a more realistic estimation of heat soakage effect, further reducing the gas turbine's transient response. During the acceleration process, the acceleration time increases by 126% when combustor temperature profile is applied. The outcome reveals the importance of considering combustor temperature profile, film cooling and TBC in heat soakage analysis.

The second case study is related to the heat soakage and tip clearance change effect on a 2-spool turbofan engine.

1. Compared with the heat soakage effect, the tip clearance change effect will last longer. In the current case, the heat soakage effect lasts for 60 seconds. However, for the tip clearance, it lasts for more than 150s.
2. During the acceleration manoeuvre, the corrected rotational speed change trend of spools will be different between the transient with consideration of heat soakage effect and the transient with consideration of heat soakage and tip clearance. In initial 20 seconds, the spools' acceleration rate of the simulation with tip clearance change will be faster due to the decrease in tip clearance and increase in component efficiency. However, after 40 seconds, the spools' CN will lower than design point due to increase in tip clearance and decrease in efficiency.
3. Heat soakage effect leads to a maximum 2.37% decrease in engine thrust during the transient manoeuvre. When considering both heat soakage and tip clearance, the thrust is increased by maximum 1.81% in initial 20 seconds and decreased by maximum 0.67% after 60 seconds.

The third case study is related to the compressor map change based on a modern 10 stage compressor model that has the similar performance with the HPC of NASA E3 engine. It is found that:

1. Heat transfer will affect the compressor speed line. When heat is added to the main gas path fluid, the speed line will move toward lower corrected flow due to the miss match and blockage effect. The opposite effect occurs when heat is extracted from the fluid.
2. Based on the idea of scaling factor, the relation between the scaling factor of pressure ratio and corrected mass flow and the non-dimensional heat transfer rate is found. When the rotational speed is kept constant, the scaling factor shows a linear relation with the non-dimensional heat transfer rate. Besides, an exponential relationship between the slopes of scaling factor lines and rotational speed is found.
3. According to the case study of a two-spool turbo fan engine, the effect of high-pressure compressor map change on gas turbine transient performance is investigated. During a Bodie process, the HPC pressure ratio and corrected mass flow are decreased by 0.7% and 0.9%

respectively during the deceleration comparing with the adiabatic process. The HPC pressure ratio and corrected mass flow are increased by 0.4% and 0.5% respectively during the re-acceleration.

Based on the developed heat transfer effect model, the research gaps defined in Section 2.4 are fulfilled. The proposed model offers a comprehensive view on the heat transfer effect on gas turbine transient performance. For researcher developing gas turbine fuel control logic, they can take the heat transfer effect into consideration. For the researchers in gas turbine degradation analysis and gas turbine mechanical design, they can also benefit from the proposed model. However, limitations exist in the proposed model. They are discussed in Section 7.

7 Recommendation for Future Work

In this Chapter, recommendations for future work are proposed to further improve the models.

7.1 Burner Time Constant

As described in Section 4.2.5.2 and 7.8Appendix D, Turbomatch does not consider combustor delay during transient simulation. The combustor delay will cause a small delay in transient response. In the sensitivity study, a 0.1 second delay was found when setting burner time constant as 0.1. It is recommend to added combustor delay in Turbomatch to improve the transient simulation accuracy.

7.2 Model Validation

Due to the lack of appropriate experimental data in the public domain, the validation of gas turbine transient performance simulation and heat flow rate during transient maneuvers is conducted by comparing the results with simulation data from the commercial software GasTurb. This comparison uses the same gas turbine engine model, transient maneuvers, and heat transfer processes. Sensitivity analysis is performed to investigate the impact of small changes in assumptions on the predicted outcomes.

If available, comparisons with actual gas turbine experimental data would enhance confidence in the proposed model. During the comparison, metrics such as shaft rotational speed, mass flow rate, thrust, component outlet temperature, and compressor pressure ratio changes can be measured and compared. However, since heat flow rate is not a directly measurable value, validation of the heat soakage model should be based on comparing transient simulation outcomes with both a simplified heat soakage model and a comprehensive heat soakage model, against available experimental data. If the results from the comprehensive heat soakage model are closer to the experimental data than those from the simplified model, then the proposed model can be considered validated.

7.3 Geometry Identification Model

For compressor and turbine geometry estimation, it is assumed the constant stage temperature change and constant axial velocity. However, in real compressor and turbine, the stage temperature will change for a better stage loading and the stage inlet axial velocity will change. Moreover, the blade is assumed as a flat plate without the setting angle. In real compressor design, the blade will have a more complex shape with certain blade inlet and outlet angle as well as the setting angle.

In order to have a better estimation on the compressor and turbine geometry estimation, Falck's method is recommended [159]. The calculation flow logic of Falck's method is shown in Figure 7-1. In his method, two main input parameters must be satisfied: the total pressure ratio π , and the stage reaction R . Several input data are required for the calculation, they are listed in Table 7-1. The main idea of Falck's method is to determine the flow area of each stage. Based on the estimated flow area, calculate the local velocity triangle as well as the stage performance. During this process, the blade inlet and outlet flow angle will be estimated based on the minimum loss incidence and deviation angle empirical correlation.

Table 7-1 Specific Parameters for Falck's Method

Input Parameters
Tip clearance/Chord ratio for Each Stage
Aspect Ratio for Each Stage
Blade Maximum Thickness/Chord Ratio for Each Stage
Axial Velocity Ratio for Each Stage
Blockage Factor for Each Stage
Diffusion Factor for Each Stage

Stage Loading Distribution

Compared with the proposed method above, Falck's method will offer a more realistic simulation outcome with the definition of stage parameters as described in Table 7-1. However, it requires very detailed data which are commonly not available from the open literature. Besides, his method is still based on the three compressor geometries as shown in Figure 3-4, which is not the real case when design the compressor.

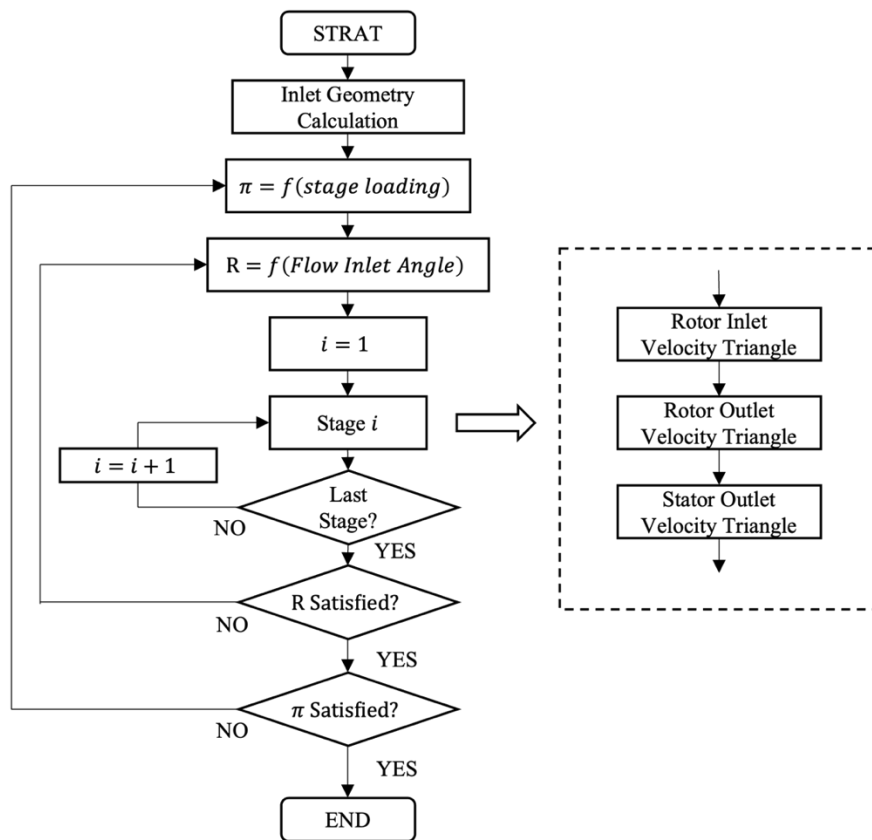


Figure 7-1 Logic flow of Falck's compressor geometry identification method

Another limitation is the constant thickness assumption of the compressor and turbine disc. In reality, disc will have a more complicate shape. To have a better prediction on disc mass and wetted area, a more complex disc model is developed, as shown in Figure 7-2. 10 parameters are required to define the disc model, including inner bore radius, outer bore radius, inner web radius, outer web radius, inner rim radius, outer rim radius, half bore width, half inner web width, half outer web width, and half rim width. By using this disc model, a more realistic

disc mass can be estimated by using Eqs. (7-1). However, this model requires too many user defined parameters. For a multistage compressor or turbine, massive parameters are required. Thus, this model is only used for self-research but not integrated into Turbomatch.

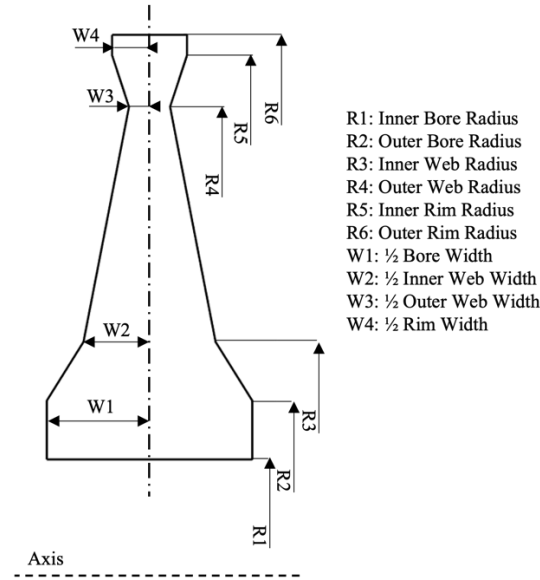


Figure 7-2 Parameterized disc model

$$M_d = f(Radius, Width, \rho_d)$$

(7-1)

7.4 Combustor System Architecture

There are several limitations on the developed combustor heat soakage model.

The first limitation is related to combustor zone temperature identification, the combustor zone temperature is based on the adiabatic condition. The heat loss caused by mass diffusion, chemical reaction and dissociation are not considered. During gas turbine acceleration, high temperature will occur in combustor due to over fuelling. When temperature is over 1800K [149], dissociation will happen and lead to heat loss. Based on the literature, the heat loss caused by dissociation is around 7% and the heat loss caused by heat soakage is around 9% [167]. Since dissociation only happens at extreme high temperature, so the magnitude of heat loss caused by dissociation during entire transient will be lower

than 7%. However, it is still recommended to consider the heat loss caused by dissociation during temperature identification.

The second limitation concerns the configuration of the proposed combustor model. The research utilizes a general low-fidelity combustor model based on a conventional rich-burn combustor, which divides the combustor into primary, secondary, and dilution zones. In this model, it is assumed that 40% of the air enters the primary zone, 50% more air enters the secondary zone, and the remaining 10% of air goes to the dilution zone. However, modern combustors, such as double annular combustors, rich-burn quick-quench lean-burn combustors, axially staged combustors, and twin annular premixing swirler combustors, incorporate advanced technologies like fuel staging and partial premixing. These technologies result in different temperature distributions along the liner compared to conventional rich-burn combustors. Although the fuel-to-air ratio in each zone can be adjusted to better approximate modern combustors, it is advisable to develop a more accurate combustor temperature distribution model specifically for modern lean-burn combustors. Here, the combustor liner temperature estimation model developed by Liu et. al. is recommended [162].

The third limitation is related to the consideration of cooling technologies. In the developed model, film cooling, internal cooling and thermal barrier coating are considered. However, in modern combustor, more cooling technologies like double skin, transpiration cooling and tiles are considered. In order to have a more comprehensive view on combustor cooling, additional heat transfer models for the motioned cooling technologies are recommended.

7.5 Heat Soakage Model Sensitivity Analysis

During the heat soakage model sensitivity analysis, the effect coming from change in model assumption on heat flow rate estimation is investigated. Because the applied value of film cooling efficiency, thermal barrier coating thickness, and luminosity factor are highly depended on specific design or working condition, the sensitivity analysis only investigate the effect from a small change in the assumed value. A more comprehensive parametric investigation based on specific gas turbine design is recommended.

7.6 Heat Soakage Model

One limitation of the heat soakage model is linked with the selected geometry model. As discussed in Section 7.3, a complex disc model is established to obtain a more realistic mass and wetted area estimation. In order to investigate the disc shape effect on the simulation outcome of the heat soakage model, a comparison between the uniform disc and non-uniform disc has been conducted. The inner radius, outer radius and total mass of the two discs are kept the same. Subsequently, a 72-second heat-up process is simulated while controlling the initial temperature to be the same. The final temperature distributions of the discs are presented in Figure 7-3, where the uniform disc temperature distribution is shown in (a), and the non-uniform disc temperature distribution is shown in (b). The comparison of the temperature distribution is shown in Figure 7-4. The heat flow rate of two disc is shown in Figure 7-5. It can be observed that there is no significant difference in temperature distribution and heat flow rate when the heat-up time is short. But the difference will be bigger when the heat up time is longer. A potential expansion of a user-friendly disc shape definition program for each stage disc with adjustable parameters is suggested.

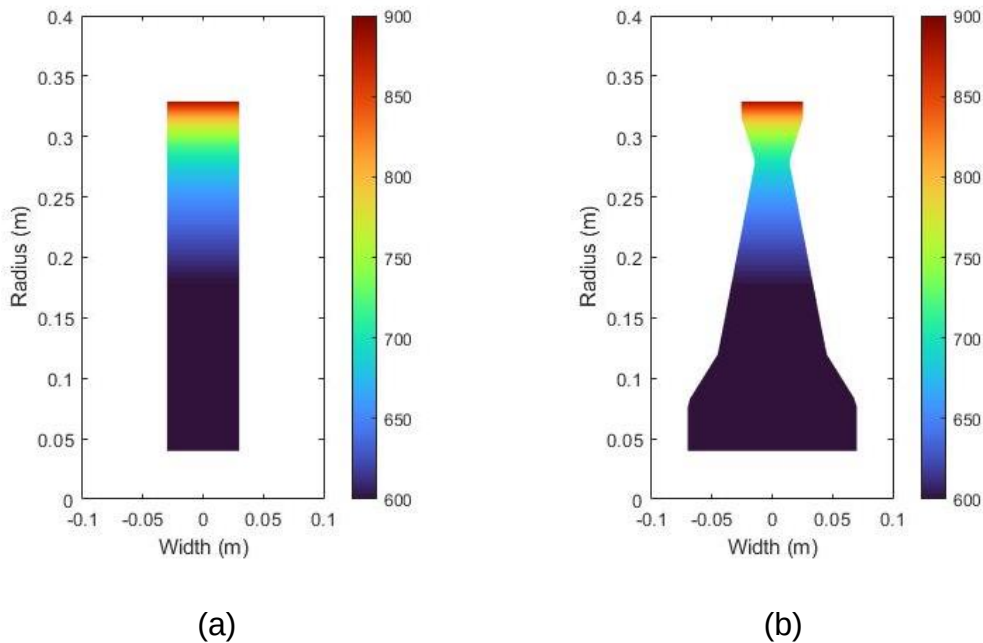


Figure 7-3 Temperature distribution of the uniform and non-uniform disc

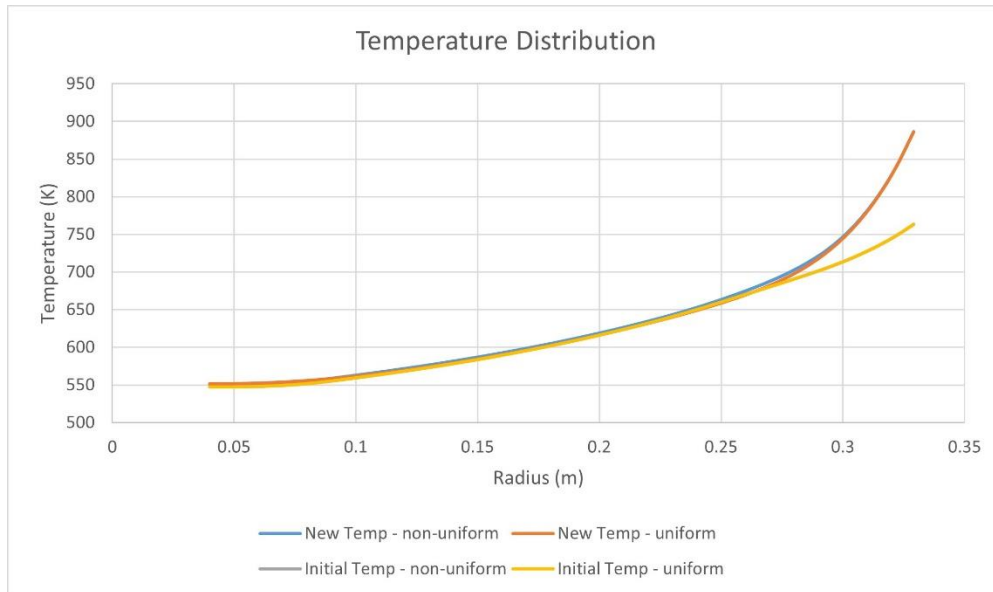


Figure 7-4 Temperature distribution along the radius direction

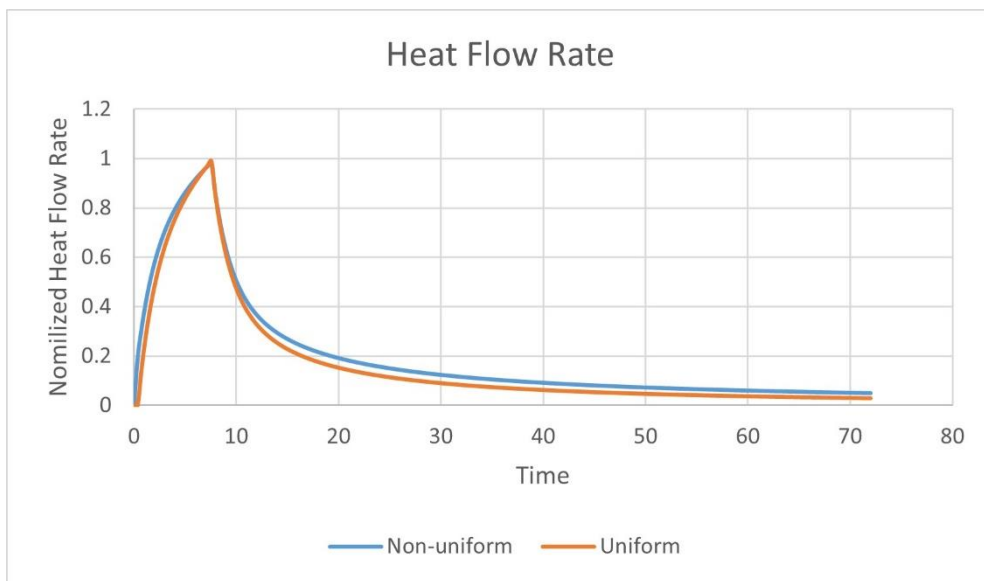


Figure 7-5 Heat flow rate comparison between uniform and non-uniform disc

7.7 Tip Clearance Movement Model

Same as the heat soakage model's limitation discussed in Section 7.6, the thermal and elastic displacement model for disc is derived based on the assumption of constant thickness disc. In order to have a better estimation of the disc radius change, the ring method proposed by Nalyor can be applied to

estimate the thermal and elastic displacement of the disc based on the heat soakage estimation outcome that discussed in Section 7.6.

7.8 Compressor Map Modification Model

For the compressor map modification, one model is developed to derive the numerical correlation between heat flow rate, CN, pressure ratio scaling factor, and corrected mass flow scaling factor. However, the derived numerical correlation is only suitable for the specific compressor model. When the compressor changes, the index in the correlation will change. The potential improvement can be the development of the correlation which takes the configuration of the compressor into consideration. With that type of correlation, it can be easily and widely applied to different gas turbine model during the transient simulation. Besides, the proposed routine can be applied to turbine map modification to have a more complete heat transfer effect investigation.

REFERENCES

- [1] H.I.H. Saravanamuttoo, G.F.C. Rogers, H. Cohen, Gas turbine theory, Sixth ed., Pearson Prentice Hall, 2009.
- [2] A.J. Fawke, H.I.H. Saravanamuttoo, Digital Computer Methods for Prediction of Gas Turbine Dynamic Response, in: SAE Technical Papers, 1971. <https://doi.org/10.4271/710550>.
- [3] J. SZUCH, Advancements in real-time engine simulation technology, in: 18th Joint Propulsion Conference, American Institute of Aeronautics and Astronautics, Reston, Virginia, 1982. <https://doi.org/10.2514/6.1982-1075>.
- [4] C. Evans, Testing and modelling aircraft gas turbines: an introduction and overview, in: UKACC International Conference on Control (CONTROL '98), IEE, 1998: pp. 1361–1366. <https://doi.org/10.1049/cp:19980428>.
- [5] V. Sanghi, B.K. Lakshmanan, V. Sundararajan, Survey of Advancements in Jet-Engine Thermodynamic Simulation, J Propuls Power 16 (2000) 797–807. <https://doi.org/10.2514/2.5644>.
- [6] H. Asgari, X. Chen, M.B. Menhaj, R. Sainudiin, Artificial Neural Network–Based System Identification for a Single-Shaft Gas Turbine, J Eng Gas Turbine Power 135 (2013). <https://doi.org/10.1115/1.4024735>.
- [7] M. Jelali, A. Kroll, Hydraulic servo-systems: modelling, identification, and control, First ed., Springer, 2003. <https://doi.org/10.1007/978-1-4471-0099-7>.
- [8] L.L. Jaw, S. Garg, Propulsion Control Technology Development in the United States A Historical Perspective, NASA Technical Memorandum, NASA/TM-2005-213978 (2005).
- [9] E.W. Otto, I. Burt L. Taylor, Dynamics of a turbojet engine considered as a quasi-static system, NACA Technical Notes 2091 (1950).
- [10] H. Gold, S. Rosenzweig, A method for estimating speed response of gas-turbine engines, NACA Research Memorandum E51K21 (1952).

- [11] J.O.N. Lawrence, R.D. Powell, The Application of Servo-Mechanism Analysis to Fuel Control Problems, *Proceedings of the Institution of Mechanical Engineers* 172 (1958) 439–469. https://doi.org/10.1243/PIME_PROC_1958_172_039_02.
- [12] D. Novik, Some linear dynamics of two-spool turbojet engines, *NACA Technical Notes* 3274 (1956).
- [13] J.F. Dugan, Component Operating Trends during Acceleration and Deceleration of Two Hypothetical Two Spool Engines, *NACA Research Memorandum* E54L28 (1955).
- [14] R.E. Filippi, J.F. Dugan, Effect of Design Over all Compressor Pressure Ratio Division on Acceleration Characteristics of Three Hypothetical Two-Spool Turbojet Engines, *NACA Research Memorandum* E56D13 (1956).
- [15] V.L. Larowe, M.M. Spencer, M. Tribus, A Dynamic Performance Computer for Gas-Turbine Engines, *J Fluids Eng* 79 (1957) 1707–1714. <https://doi.org/10.1115/1.4013457>.
- [16] C. Kong, J. Ki, K. Koh, Steady-State and Transient Performance Simulation of a Turboshaft Engine With Free Power Turbine, in: *Volume 2: Coal, Biomass and Alternative Fuels; Combustion and Fuels; Oil and Gas Applications; Cycle Innovations*, American Society of Mechanical Engineers, 1999. <https://doi.org/10.1115/99-GT-375>.
- [17] W.J. Tong, S.J. Tang, Geared turbofan engine transient modeling research based on CMF method, *Hangkong Dongli Xuebao/Journal of Aerospace Power* 26 (2011) 1377–1383. <https://doi.org/10.13224/j.cnki.jasp.2011.06.029>.
- [18] G. Wortmann, O. Schmitz, M. Hornung, Comparative assessment of transient characteristics of conventional and hybrid gas turbine engine, *CEAS Aeronaut J* 5 (2014) 209–223. <https://doi.org/10.1007/s13272-014-0101-8>.

- [19] M. Montero Carrero, M.L. Ferrari, W. De Paepe, A. Parente, S. Bram, F. Contino, Transient Simulations of a T100 Micro Gas Turbine Converted Into a Micro Humid Air Turbine, in: Volume 3: Coal, Biomass and Alternative Fuels; Cycle Innovations; Electric Power; Industrial and Cogeneration, American Society of Mechanical Engineers, 2015: pp. 1–9. <https://doi.org/10.1115/GT2015-43277>.
- [20] M. Raggio, D. Bellotti, M.L. Ferrari, Transient Analysis of a Micro Gas Turbine With Fuel Composition Change, in: Volume 4: Cycle Innovations; Cycle Innovations: Energy Storage, American Society of Mechanical Engineers, 2022: pp. 1–10. <https://doi.org/10.1115/GT2022-81300>.
- [21] M.J. Kim, J.H. Kim, T.S. Kim, Program development and simulation of dynamic operation of micro gas turbines, *Appl Therm Eng* 108 (2016) 122–130. <https://doi.org/10.1016/j.applthermaleng.2016.07.103>.
- [22] F. Mueller, R. Gaynor, A.E. Auld, J. Brouwer, F. Jabbari, G.S. Samuelsen, Synergistic integration of a gas turbine and solid oxide fuel cell for improved transient capability, *J Power Sources* 176 (2008) 229–239. <https://doi.org/10.1016/j.jpowsour.2007.10.081>.
- [23] V. Singh, L.-U. Axelsson, W.P.J. Visser, Transient Performance Analysis of an Industrial Gas Turbine Operating on Low-Calorific Fuels, in: Volume 8: Microturbines, Turbochargers and Small Turbomachines; Steam Turbines, American Society of Mechanical Engineers, 2016: pp. 1–9. <https://doi.org/10.1115/GT2016-57523>.
- [24] A.J. Fawke, H.I.H. Saravanamuttoo, Experimental Investigation of Methods for Improving the Dynamic Response of a Twin-Spool Turbojet Engine, *Journal of Engineering for Power* 93 (1971) 418–424. <https://doi.org/10.1115/1.3445601>.
- [25] L. Chuankai, Q. Tian, D. Shuiting, Transient analysis of volume packing effects on turbofan engine, *Procedia Eng* 17 (2011) 549–558. <https://doi.org/10.1016/j.proeng.2011.10.068>.

- [26] K. CHUNG, K. LEAMY, T. COLLINS, A turbine engine aerodynamic model for in-install transient simulation, in: 21st Joint Propulsion Conference, American Institute of Aeronautics and Astronautics, Reston, Virginia, 1985. <https://doi.org/10.2514/6.1985-1429>.
- [27] T. HAYKIN, S. MURTHY, Transient engine performance with water ingestion, in: 22nd Joint Propulsion Conference, American Institute of Aeronautics and Astronautics, Reston, Virginia, 1986. <https://doi.org/10.2514/6.1986-1621>.
- [28] R. Yadav, Y. Kapadi, A. Pashilkar, Aero-Thermodynamic Model for Digital Simulation of Turbofan Engine, in: Volume 1: Turbo Expo 2005, ASMEDC, 2005: pp. 63–70. <https://doi.org/10.1115/GT2005-68248>.
- [29] A. Traverso, F. Calzolari, A. Massardo, Transient Analysis of and Control System for Advanced Cycles Based on Micro Gas Turbine Technology, in: Volume 3: Turbo Expo 2003, ASMEDC, 2003: pp. 201–209. <https://doi.org/10.1115/GT2003-38269>.
- [30] A. Traverso, R. Scarpellini, A. Massardo, Experimental Results and Transient Model Validation of an Externally Fired Micro Gas Turbine, in: Volume 5: Turbo Expo 2005, ASMEDC, 2005: pp. 35–43. <https://doi.org/10.1115/GT2005-68100>.
- [31] C. Wang, Y.G. Li, B.Y. Yang, Transient performance simulation of aircraft engine integrated with fuel and control systems, *Appl Therm Eng* 114 (2017) 1029–1037. <https://doi.org/10.1016/j.applthermaleng.2016.12.036>.
- [32] S. Shamekhi Amiri, J. Al-Zaili, A.I. Sayma, Development of a Dynamic Model for Simulating the Transient Behaviour of a Solar-Powered Micro Gas Turbine, in: Volume 4: Cycle Innovations; Cycle Innovations: Energy Storage, American Society of Mechanical Engineers, 2022: pp. 1–10. <https://doi.org/10.1115/GT2022-79059>.
- [33] A.J. Fawke, H.I.H. Saravanamuttoo, M. Holmes, Experimental Verification of a Digital Computer Simulation Method for Predicting Gas Turbine

- Dynamic Behaviour, Proceedings of the Institution of Mechanical Engineers 186 (1972) 323–329.
https://doi.org/10.1243/PIME_PROC_1972_186_035_02.
- [34] A. Nishi, T. Sawada, Digital Simulation of a Variable Geometry Gas Turbine, Bulletin of University of Osaka Prefecture. Series A, Engineering and Natural Sciences 23 (1974) 111–125.
<https://doi.org/10.24729/00008728>.
- [35] P. Pilidis, Digital simulation of Gas Turbine Performance. Ph.D. thesis, University of Glasgow, 1983.
- [36] P.H. Naylor, Gas turbine transient performance: heat soakage modelling. Ph.D thesis, Cranfield University, 2004.
- [37] N.U. Rahman, J.F. Whidborne, Real-Time Transient Three Spool Turbofan Engine Simulation: A Hybrid Approach, J Eng Gas Turbine Power 131 (2009) 1–8. <https://doi.org/10.1115/1.3079611>.
- [38] Y.G. Li, N.R.L. Maccallum, P. Pilidis, Pressure Waves in Volume Effect in Gas-Turbine Transient-Performance Models, J Propuls Power 17 (2001) 706–710. <https://doi.org/10.2514/2.5799>.
- [39] E. Tsoutsanis, N. Meskin, M. Benammar, K. Khorasani, Dynamic performance simulation of an aeroderivative gas turbine using the matlab simulink environment, ASME International Mechanical Engineering Congress and Exposition, Proceedings (IMECE) 4 A (2013) 1–10.
<https://doi.org/10.1115/IMECE2013-64102>.
- [40] Z. Li, Aircraft engine transient performance modelling with heat soakage effects. Ph.D thesis, Cranfield University, 2019.
- [41] H. Asgari, X. Chen, Gas Turbines Modeling, Simulation, and Control, First ed., CRC Press, 2015. <https://doi.org/10.1201/b18956>.
- [42] H. Asgari, X. Chen, R. Sainudiin, M. Morini, M. Pinelli, P.R. Spina, M. Venturini, Modeling and Simulation of the Start-Up Operation of a Heavy-

- Duty Gas Turbine by Using NARX Models, in: Volume 3A: Coal, Biomass and Alternative Fuels; Cycle Innovations; Electric Power; Industrial and Cogeneration, American Society of Mechanical Engineers, 2014: pp. 1–10. <https://doi.org/10.1115/GT2014-25056>.
- [43] N. Chiras, C. Evans, D. Rees, Global Nonlinear Modeling of Gas Turbine Dynamics Using NARMAX Structures, *J Eng Gas Turbine Power* 124 (2002) 817–826. <https://doi.org/10.1115/1.1470483>.
- [44] A.E. Ruano, P.J. Fleming, C. Teixeira, K. Rodríguez-Vázquez, C.M. Fonseca, Nonlinear identification of aircraft gas-turbine dynamics, *Neurocomputing* 55 (2003) 551–579. [https://doi.org/10.1016/S0925-2312\(03\)00393-X](https://doi.org/10.1016/S0925-2312(03)00393-X).
- [45] A. Gambarotta, I. Vaja, A real time dynamic model of a micro - Gas turbine CHP system with regeneration, *Proceedings of the ASME Power Conference 2007* (2007) 575–586. <https://doi.org/10.1115/POWER2007-22098>.
- [46] R. Rezvani, M. Ozcan, B. Kestner, J. Tai, D.N. Mavris, R. Meisner, S. Sirica, A Gas Turbine Engine Model of Transient Operation Across the Flight Envelope, in: Volume 3: Controls, Diagnostics and Instrumentation; Education; Electric Power; Microturbines and Small Turbomachinery; Solar Brayton and Rankine Cycle, *ASME*, 2011: pp. 133–140. <https://doi.org/10.1115/GT2011-45565>.
- [47] M. Rahn timer, H. Ghorbani, A. Montazeri, Nonlinear identification of a gas turbine system in transient operation mode using neural network, in: *4th Conference on Thermal Power Plants*, IEEE, 2012: pp. 1–6.
- [48] R. Khalili, M. Karrari, Modeling and identification of an industrial gas turbine using classical and non-classical approaches, in: *2017 Iranian Conference on Electrical Engineering (ICEE)*, IEEE, 2017: pp. 667–672. <https://doi.org/10.1109/IranianCEE.2017.7985123>.

- [49] I. Koleini, A. Roudbari, V. Marefat, EGT prediction of a micro gas turbine using statistical and artificial intelligence approach, *IEEE Aerospace and Electronic Systems Magazine* 33 (2018) 4–13. <https://doi.org/10.1109/MAES.2018.170045>.
- [50] S. Kim, K. Kim, C. Son, Transient system simulation for an aircraft engine using a data-driven model, *Energy* 196 (2020) 117046. <https://doi.org/10.1016/j.energy.2020.117046>.
- [51] B. Wu, Dynamic performance simulation analysis method of split shaft gas turbine based on RBF neural network, *Energy Reports* 7 (2021) 947–958. <https://doi.org/10.1016/j.egyr.2021.09.178>.
- [52] I. Ibrahim, O. Akhrif, H. Moustapha, M. Staniszewski, Neural Networks Modelling of Aero-derivative Gas Turbine Engine: A Comparison Study, in: *Proceedings of the 16th International Conference on Informatics in Control, Automation and Robotics*, SCITEPRESS - Science and Technology Publications, 2019: pp. 738–745. <https://doi.org/10.5220/0007928907380745>.
- [53] I.M.A. Ibrahim, O. Akhrif, H. Moustapha, M. Staniszewski, An Ensemble of Recurrent Neural Networks for Real Time Performance Modeling of Three-Spool Aero-Derivative Gas Turbine Engine, *J Eng Gas Turbine Power* 143 (2021) 1–10. <https://doi.org/10.1115/1.4051112>.
- [54] H. Asgari, M. Venturini, X. Chen, R. Sainudiin, Modeling and Simulation of the Transient Behavior of an Industrial Power Plant Gas Turbine, *J Eng Gas Turbine Power* 136 (2014) 1–10. <https://doi.org/10.1115/1.4026215>.
- [55] F. Jurado, Non-linear modeling of micro-turbines using NARX structures on the distribution feeder, *Energy Convers Manag* 46 (2005) 385–401. <https://doi.org/10.1016/j.enconman.2004.03.012>.
- [56] H. Asgari, X. Chen, M. Morini, M. Pinelli, R. Sainudiin, P.R. Spina, M. Venturini, NARX models for simulation of the start-up operation of a single-

- shaft gas turbine, *Appl Therm Eng* 93 (2016) 368–376.
<https://doi.org/10.1016/j.applthermaleng.2015.09.074>.
- [57] H. Asgari, E. Ory, Prediction of Dynamic Behavior of a Single Shaft Gas Turbine Using NARX Models, in: *Volume 6: Ceramics and Ceramic Composites; Coal, Biomass, Hydrogen, and Alternative Fuels; Microturbines, Turbochargers, and Small Turbomachines*, American Society of Mechanical Engineers, 2021: pp. 1–10.
<https://doi.org/10.1115/GT2021-58960>.
- [58] B. Yu, W. Shu, Research on Turbofan Engine Model above Idle State Based on NARX Modeling Approach, *IOP Conf Ser Mater Sci Eng* 187 (2017) 012002. <https://doi.org/10.1088/1757-899X/187/1/012002>.
- [59] H. Bahlawan, M. Morini, M. Pinelli, P. Ruggero Spina, M. Venturini, Development of Reliable NARX Models of Gas Turbine Cold, Warm, and Hot Start-Up, *J Eng Gas Turbine Power* 140 (2018).
<https://doi.org/10.1115/1.4038838>.
- [60] E. Ory, H. Asgari, Machine learning approaches for modelling a single shaft gas turbine, *Int J Model Identif Control* 37 (2021) 275.
<https://doi.org/10.1504/IJMIC.2021.10045910>.
- [61] M. Alsarayreh, O. Mohamed, M. Matar, Modeling a Practical Dual-Fuel Gas Turbine Power Generation System Using Dynamic Neural Network and Deep Learning, *Sustainability* 14 (2022) 870.
<https://doi.org/10.3390/su14020870>.
- [62] M.B. Hashmi, T.A. Lemma, S. Ahsan, S. Rahman, Transient Behavior in Variable Geometry Industrial Gas Turbines: A Comprehensive Overview of Pertinent Modeling Techniques, *Entropy* 23 (2021) 250.
<https://doi.org/10.3390/e23020250>.
- [63] N.R.L. MacCallum, The Performance of Turbojet Engines during the ‘Thermal Soak’ Transient,’ *Proceedings of the Institution of Mechanical*

- Engineers, Conference Proceedings 184 (1969) 23–29.
https://doi.org/10.1243/PIME_CONF_1969_184_179_02.
- [64] C. Daguang, P. Yongquan, Thermal effects on transient process of a turbojet engine, *Hangkong Dongli Xuebao/Journal of Aerospace Power* 2 (1987) 85–86.
- [65] P.P. Walsh, P. Fletcher, *Gas turbine performance*, second ed., Oxford Blackwell Science, 2004.
- [66] K. Bauerfeind, Die exakte Bestimmung des Uebertragungsverhaltens von Turbostrahltriebwerken unter Beruecksichtigung des instationaeren Verhaltens seiner Komponenten, PhD thesis in German, Fakultae fuer Maschinenwesen und Elektrotechnik der Technischen Hochschule Muench, 1968.
- [67] A.J. Fawke, H.I.H. Saravanamuttoo, Digital Computer Simulation of the Dynamic Response of a Twin-Spool Turbofan With Mixed Exhausts., *Aeronautical Journal* 77 (1973) 471–478.
<https://doi.org/https://doi.org/10.1017/S0001924000041567>.
- [68] S. KHALID, R. HEARNE, Enhancing dynamic model fidelity for improved prediction of turbofanengine transient performance, in: 16th Joint Propulsion Conference, American Institute of Aeronautics and Astronautics, Reston, Virigina, 1980. <https://doi.org/10.2514/6.1980-1083>.
- [69] P.H. Naylor, *Gas turbine transient performance: heat soakage modelling*. Ph.D thesis, Cranfield University, 2004.
- [70] W.P.J. Visser, I.D. Dountchev, Modeling Thermal Effects on Performance of Small Gas Turbines, in: *Volume 1: Aircraft Engine; Fans and Blowers; Marine*, American Society of Mechanical Engineers, 2015: pp. 1–12.
<https://doi.org/10.1115/GT2015-42744>.
- [71] J.W. Chapman, T.-H. Guo, J.L. Kratz, J.S. Litt, Integrated Turbine Tip Clearance and Gas Turbine Engine Simulation, in: 52nd AIAA/SAE/ASEE Joint Propulsion Conference, American Institute of Aeronautics and

- Astronautics, Reston, Virginia, 2016: pp. 1–16.
<https://doi.org/10.2514/6.2016-5047>.
- [72] J.L. Kratz, D.E. Culley, J.W. Chapman, Approximation of Engine Casing Temperature Constraints for Casing Mounted Electronics, in: 52nd AIAA/SAE/ASEE Joint Propulsion Conference, American Institute of Aeronautics and Astronautics, Reston, Virginia, 2016: pp. 1–23.
<https://doi.org/10.2514/6.2016-4858>.
- [73] F. Chen, Y. Chen, K. Song, Y. Li, Study on the Influence of Heat Transfer Effect on Performance Simulation of Engine Transition State, in: 2019 IEEE 10th International Conference on Mechanical and Aerospace Engineering (ICMAE), IEEE, 2019: pp. 443–448.
<https://doi.org/10.1109/ICMAE.2019.8880972>.
- [74] R.S. Merkler, S. Staudacher, Modeling of Heat Transfer and Clearance Changes in Transient Performance Calculations: A Comparison, in: Volume 2: Aircraft Engine; Ceramics; Coal, Biomass and Alternative Fuels; Controls, Diagnostics and Instrumentation; Environmental and Regulatory Affairs, ASMEDC, 2006: pp. 37–45. <https://doi.org/10.1115/GT2006-90041>.
- [75] C. Riegler, Correlations to include heat transfer in gas turbine performance calculations, *Aerosp Sci Technol* 3 (1999) 281–292.
[https://doi.org/10.1016/S1270-9638\(00\)86964-3](https://doi.org/10.1016/S1270-9638(00)86964-3).
- [76] M. Vieweg, F. Wolters, R.-G. Becker, Comparison of a Heat Soakage Model With Turbofan Transient Engine Data, in: Volume 1: Aircraft Engine; Fans and Blowers; Marine; Honors and Awards, American Society of Mechanical Engineers, 2017: pp. 1–12. <https://doi.org/10.1115/GT2017-63461>.
- [77] T. Qiuye, J. Rui, X. Xuan, Z. Heng, Y. Hongming, Effects of heat soakage on transient performance of gas turbine engine, *Journal of Aerospace Power* 32 (2017) 630–636.
<https://doi.org/10.13224/j.cnki.jasp.2017.03.015>.

- [78] J. Kurzke, H. Jan., Propulsion and power: an exploration of gas turbine performance modeling, First ed., Springer, 2018.
- [79] Z. Zhili, L. Kuo, Analysis of heat transfer impacting on gas turbine engine transients, *Journal of Propulsion Technology* 17 (1996) 10–15.
- [80] A. Stamatis, K. Mathioudakis, J. Ruiz, B. Curnock, Real Time Engine Model Implementation for Adaptive Control and Performance Monitoring of Large Civil Turbofans, in: Volume 1: Aircraft Engine; Marine; Turbomachinery; Microturbines and Small Turbomachinery, American Society of Mechanical Engineers, 2001: pp. 1–7. <https://doi.org/10.1115/2001-GT-0362>.
- [81] W. Hu, L. Xiaochun, Numerical simulation of effects of heat transfer on the transient performance of gas turbine engines, *Journal of Propulsion Technology* 23 (2002) 445–447.
- [82] P.M. Tagade, S.K. Sane, K. Sudhakar, Startup Simulation of Twin Spool Turbofan Engine, *International Journal of Turbo and Jet Engines* 27 (2010) 265–276. <https://doi.org/10.1515/TJJ.2010.27.3-4.265>.
- [83] M.A. da Cunha Alves, J.R. Barbosa, A step further in gas turbine dynamic simulation, *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy* 217 (2003) 583–592. <https://doi.org/10.1177/095765090321700606>.
- [84] C.R. Davison, A.M. Birk, Set Up and Operational Experience With a Micro-Turbine Engine for Research and Education, in: Volume 1: Turbo Expo 2004, ASMEDE, 2004: pp. 849–858. <https://doi.org/10.1115/GT2004-53377>.
- [85] C.R. Davison, A.M. Birk, Comparison of Transient Modeling Techniques for a Micro Turbine Engine, in: Volume 5: Marine; Microturbines and Small Turbomachinery; Oil and Gas Applications; Structures and Dynamics, Parts A and B, ASMEDE, 2006: pp. 449–458. <https://doi.org/10.1115/GT2006-91088>.

- [86] M. Vieweg, C. Klein, S. Reitenbach, F. Wolters, R.G. Becker, Coupling of predesign and performance tools for transient aircraft engine analyses, in: 31st Congress of the International Council of the Aeronautical Sciences, ICAS 2018, 2018.
- [87] A. Ferrand, M. Bellenoue, Y. Bertin, R. Cirligeanu, P. Marconi, F. Mercier-Calvairac, High Fidelity Modeling of the Acceleration of a Turboshift Engine During a Restart, in: Volume 1: Aircraft Engine; Fans and Blowers; Marine, American Society of Mechanical Engineers, 2018. <https://doi.org/10.1115/GT2018-76654>.
- [88] J.H. Kim, T.S. Kim, Development of a program to simulate the dynamic behavior of heavy-duty gas turbines during the entire start-up operation including very early part, Journal of Mechanical Science and Technology 33 (2019) 4495–4510. <https://doi.org/10.1007/s12206-019-0845-5>.
- [89] Z. Li, Y.G. Li, S. Sampath, Aeroengine transient performance simulation integrated with generic heat soakage and tip clearance model, Aeronautical Journal (2022) 1–23. <https://doi.org/10.1017/aer.2022.15>.
- [90] J.A. Kypuros, K.J. Melcher, A reduced model for prediction of thermal and rotational effects on turbine tip clearance, NASA Technical Memorandum, NASA/TM-2003-212226 (2003).
- [91] J.A. Kypuros, R. Colson, A. Muioz, Improved Temperature Dynamic Model of Turbine Subcomponents for Facilitation of Generalized Tip Clearance Control, NASA NAG3-2857 (2004).
- [92] P. Pilidis, N.R.L. MacCallum, A Study of the Prediction of Tip and Seal Clearances and Their Effects in Gas Turbine Transients, in: Volume 1: Turbomachinery, American Society of Mechanical Engineers, 1984. <https://doi.org/10.1115/84-GT-245>.
- [93] N.R.L. MacCallum, Transient Expansion of the Components of an Air Seal on a Gas Turbine Disc, in: SAE Technical Papers, 1977. <https://doi.org/10.4271/770974>.

- [94] J.L. Kratz, J.W. Chapman, Active Turbine Tip Clearance Control Trade Space Analysis of an Advanced Geared Turbofan Engine, in: 2018 Joint Propulsion Conference, American Institute of Aeronautics and Astronautics, Reston, Virginia, 2018. <https://doi.org/10.2514/6.2018-4822>.
- [95] H. SHENG, T. LIU, Y. ZHAO, Q. CHEN, B. YIN, R. HUANG, New model-based method for aero-engine turbine blade tip clearance measurement, Chinese Journal of Aeronautics (2022). <https://doi.org/10.1016/j.cja.2022.09.012>.
- [96] Y. Fang, Y. Liu, Y. Yu, X. He, Radial Displacement of HPT Rotor caused by the Centrifugal and Heat, Gas Turbine Experiment and Research 24 (2011) 31–35. <https://doi.org/10.3969/j.issn.1627-9730.2011.02.051>.
- [97] Y. Fang, Y. Liu, X. He, Study of the Radial Elongations of the Turbine Rotor with the Centrifugal Effect, Ship Electronic Engineering 31 (2011) 9–12. <https://doi.org/10.3969/j.issn.1627-9730.2011.02.051>.
- [98] G.A. Halls, Air Cooling of Turbine Blades and Vanes, Aircraft Engineering and Aerospace Technology 39 (1967) 4–14. <https://doi.org/10.1108/eb034284>.
- [99] B. Lakshminarayana, Methods of Predicting the Tip Clearance Effects in Axial Flow Turbomachinery, Journal of Basic Engineering 92 (1970) 467–480. <https://doi.org/10.1115/1.3425036>.
- [100] E.A. Baskharone, Principles of Turbomachinery in Air-Breathing Engines, Cambridge University Press, 2006. <https://doi.org/10.1017/CBO9780511616846>.
- [101] H. Chen, Z. Zhili, The Investigation of the Simulating Model for Transient Performance of the Turbofan Engine, Journal of Aerospace Power 17 (2002) 75–79. <https://doi.org/10.13224/j.cnki.jasp.2002.01.013>.
- [102] W. Jiamei, X. Xuan, T. Qiuye, Z. Kang, Y. Kun, The effects of tip clearance of compressors on the engine transient, in: Chinese Congress of Theoretical and Applied Mechanics, 2019.

- [103] X. Ma, Heat soakage Effects on Gas Turbine Engine Performance, Cranfield University, MSc thesis, 2021.
- [104] A.G. Hourmouziadis J., An Integrated Aero/Mechanical Performance Approach to High Technology Turbine Design, AGARD (1987). <https://www.sto.nato.int/publications/AGARD/AGARD-CP-421/AGARD-CP-421.pdf> (accessed 14 March 2023).
- [105] M.R. Shirzadi, H. Saeidi, The Effects of Tip Clearance on Performance of a Heavy Duty Multi Stages Axial Turbine, in: Volume 8: Turbomachinery, Parts A, B, and C, American Society of Mechanical Engineers, 2012: pp. 1531–1541. <https://doi.org/10.1115/GT2012-69553>.
- [106] N.R.L. Maccallum, Thermal Influences in Gas Turbine Transients: Effects of Changes in Compressor Characteristics, in: Volume 1B: Gas Turbines, American Society of Mechanical Engineers, 1979. <https://doi.org/10.1115/79-GT-143>.
- [107] N.R.L. MACCALLUM, Effect of “Bulk” Heat Transfers in Aircraft Gas Turbines on Compressor Surge Margins., (1973) 94–100. <https://www.scopus.com/record/display.uri?eid=2-s2.0-85042249153&origin=resultslist&sort=r-f&src=s&sid=4335a063ca0ba22ba29818a52c94fa0f&sot=aut&sdt=a&sl=17&s=AU-ID%286701497875%29&relpos=9&citeCnt=0&searchTerm=> (accessed 14 March 2023).
- [108] A.D. Grant, The effect of heat transfer on boundary layer stability, University of Glasgow, PhD thesis, 1973.
- [109] N.R.L. Maccallum, A.D. Grant, The Effect of Boundary Layer Changes Due to Transient Heat Transfer on the Performance of an Axial-Flow Air Compressor, in: SAE Technical Papers, 1977: pp. 1357–1364. <https://doi.org/10.4271/770284>.
- [110] R. CRAWFORD, A. BURWELL, Quantitative evaluation of transient heat transfer on axial flow compressor stability, in: 21st Joint Propulsion

Conference, American Institute of Aeronautics and Astronautics, Reston, Virginia, 1985. <https://doi.org/10.2514/6.1985-1352>.

- [111] P. Pilidis, N.R.L. Maccallum, A General Program for the Prediction of the Transient Performance of Gas Turbines, in: Volume 1: Aircraft Engine; Marine; Turbomachinery; Microturbines and Small Turbomachinery, American Society of Mechanical Engineers, 1985. <https://doi.org/10.1115/85-GT-209>.
- [112] N.R.L. Maccallum, P. Pilidis, The Prediction of Surge Margins During Gas Turbine Transients, in: Volume 1: Aircraft Engine; Marine; Turbomachinery; Microturbines and Small Turbomachinery, American Society of Mechanical Engineers, 1985. <https://doi.org/10.1115/85-GT-208>.
- [113] A.G. Stamatis, K. Mathioudakis, The Influence of Heat Transfer Effects on Turbine Performance Characteristics, in: Volume 3: Heat Transfer, Parts A and B, ASMEDC, 2006: pp. 829–838. <https://doi.org/10.1115/GT2006-91039>.
- [114] J. Larjola, Simulation of Surge Margin Changes due to Heat Transfer Effects in Gas Turbine Transients, in: Volume 2: Aircraft Engine; Marine; Microturbines and Small Turbomachinery, American Society of Mechanical Engineers, 1984: pp. 1–9. <https://doi.org/10.1115/84-GT-129>.
- [115] G. Torella, Transient performance and behaviour of gas turbine engines, Proceedings of the ASME Turbo Expo 2 (1990) 0–7. <https://doi.org/10.1115/90-GT-188>.
- [116] M. Morini, G. Cataldi, M. Pinelli, M. Venturini, A Model for the Simulation of Large-Size Single-Shaft Gas Turbine Start-Up Based on Operating Data Fitting, in: Volume 6: Turbo Expo 2007, Parts A and B, ASMEDC, 2007: pp. 1849–1856. <https://doi.org/10.1115/GT2007-27373>.
- [117] P.N. Shah, Novel Turbomachinery Concepts for Highly Integrated Airframe / Propulsion Systems, Massachusetts Institute of Technology, PhD thesis, 2007.

- [118] A. Kiss, Z. Spakovszky, Effects of Transient Heat Transfer on Compressor Stability, in: Volume 1: Aircraft Engine; Fans and Blowers; Marine, American Society of Mechanical Engineers, 2018: pp. 1–11. <https://doi.org/10.1115/GT2018-75413>.
- [119] A. Kiss, Z. Spakovszky, Effects of Transient Heat Transfer on Compressor Stability, J Turbomach 140 (2018) 1–9. <https://doi.org/10.1115/1.4041290>.
- [120] Z.-J. Li, Y.-G. Li, T. Korakianitis, Gas Turbine Transient Performance Simulation With Simplified Heat Soakage Model, in: Volume 5: Controls, Diagnostics, and Instrumentation; Cycle Innovations; Cycle Innovations: Energy Storage, American Society of Mechanical Engineers, 2020: pp. 1–12. <https://doi.org/10.1115/GT2020-14484>.
- [121] B.D. MacIsaac, H.I.H. Saravanamuttoo, A Comparison of Analog, Digital and Hybrid Computing Techniques for Simulation of Gas Turbine Performance, in: Volume 1B: General, American Society of Mechanical Engineers, 1974. <https://doi.org/10.1115/74-GT-127>.
- [122] J. Sellers, C. Daniele, Dyngen - a Program and and for Calculating Performance of Turbojet Transient, NASA Technical Note, NASA-TN-D-7901 (1975).
- [123] C.J. Daniele, S.M. Krosel, J.R. Szuch, E.J. Westerkamp, Digital Computerprogram for Generating Dynamic Turbofan Engine Models (Digtem), NASA Technical Memorandum, NASA/TM-83446 (1983).
- [124] G. SADLER, K. MELCHER, DEAN - A program for Dynamic Engine ANalysis, in: 21st Joint Propulsion Conference, American Institute of Aeronautics and Astronautics, Reston, Virigina, 1985. <https://doi.org/10.2514/6.1985-1354>.
- [125] J.R. Szuch, HYDES: A generalized hybrid computer program for studying turbojet or turbofan engine dynamics, NASA Technical Memorandum, NASA-TM-X-3014 (1974).

- [126] R.W. Koenig, L.H. Fisbbacb, GENENG—A Program for Calculating Design and Off-Design Performance for Turbojet and Turbofan Engines, NASA Technical Note, NASA-TN-D-6552 (1972).
- [127] T. Schobeiri, M. Abouelkheir, C. Lippke, Getran: A geimeraic, modularly structured computer code for simulation of dynamic behavior of aero- and power generation gas turbine engines, ASME 1993 International Gas Turbine and Aeroengine Congress and Exposition, GT 1993 3C (1993). <https://doi.org/10.1115/93-GT-388>.
- [128] J.A. Reed, A.A. Afjeh, Computational Simulation of Gas Turbines: Part I — Foundations of Component-Based Models, in: Volume 1: Aircraft Engine; Marine; Turbomachinery; Microturbines and Small Turbomachinery, American Society of Mechanical Engineers, 1999. <https://doi.org/10.1115/99-GT-346>.
- [129] J.A. Reed, A.A. Afjeh, Computational Simulation of Gas Turbines: Part II — Extensible Domain Framework, in: Volume 1: Aircraft Engine; Marine; Turbomachinery; Microturbines and Small Turbomachinery, American Society of Mechanical Engineers, 1999. <https://doi.org/10.1115/99-GT-347>.
- [130] J.K. Lytle, The Numerical Propulsion System Simulation: A Multidisciplinary Design System for Aerospace Vehicles, Nasa Technical Memorandum, NASA/TM-1999-209194 (1999).
- [131] J.K. Lytle, The Numerical Propulsion System Simulation: An Overview, NASA Technical Report (2000). <https://ntrs.nasa.gov/citations/20000063377>.
- [132] C. Argote, B.K. Kestner, D.N. Mavris, NPSS volume dynamic capability for real-time physics based engine modeling, Proceedings of the ASME Turbo Expo 1 (2011) 139–148. <https://doi.org/10.1115/GT2011-45374>.
- [133] <https://tfaws.nasa.gov/TFAWS01/NASA/12MDA%20Panel/dOWEN.PDF> (accessed 12 February 2024), (n.d.).

- [134] D.E. Hirst, Digital simulation of nonlinear dynamic systems using standard FORTRAN, Electron Lett 7 (1971) 102. <https://doi.org/10.1049/el:19710069>.
- [135] J.R. Palmer, Y. Cheng-Zhong, TurboTrans — A Programming Language for the Performance Simulation of Arbitrary Gas Turbine Engines with Arbitrary Control Systems, International Journal of Turbo and Jet Engines 2 (1985) 1–8. <https://doi.org/10.1515/TJJ.1985.2.1.19>.
- [136] A. Traverso, TRANSEO code for the dynamic performance simulation of micro gas turbine cycles, Proceedings of the ASME Turbo Expo 5 (2005) 45–54. <https://doi.org/10.1115/GT2005-68101>.
- [137] M.A. Chappell, E.G. Blevins, Advanced Turbine Engine Simulation Technique Development and Applications To Testing., AIAA Paper (1986). <https://doi.org/10.2514/6.1986-1731>.
- [138] M. CHAPPELL, P. MCLAUGHLIN, An approach to modeling continuous turbine engine operation from startup to shutdown, in: 27th Joint Propulsion Conference, American Institute of Aeronautics and Astronautics, Reston, Virginia, 1991. <https://doi.org/10.2514/6.1991-2373>.
- [139] J. Kurzke, Advanced User-Friendly Gas Turbine Performance Calculations on a Personal Computer, in: Volume 5: Manufacturing Materials and Metallurgy; Ceramics; Structures and Dynamics; Controls, Diagnostics and Instrumentation; Education; IGTI Scholar Award, American Society of Mechanical Engineers, 1995. <https://doi.org/10.1115/95-GT-147>.
- [140] (GasTurb GmbH), GasTurb 12 - Design and Off-Design Performance of Gas Turbines, GasTurb 12 - User Guide (2015).
- [141] W.P.J. Visser, M.J. Broomhead, GSP, a Generic Object-Oriented Gas Turbine Simulation Environment, in: Volume 1: Aircraft Engine; Marine; Turbomachinery; Microturbines and Small Turbomachinery, American Society of Mechanical Engineers, 2000: pp. 1–8. <https://doi.org/10.1115/2000-GT-0002>.

- [142] <https://www.gspteam.com/screenshots.html> (accessed 13 February 2024), (n.d.).
- [143] A. Alexiou, K. Mathioudakis, Development of Gas Turbine Performance Models Using a Generic Simulation Tool, in: Volume 1: Turbo Expo 2005, ASMEDC, 2005: pp. 185–194. <https://doi.org/10.1115/GT2005-68678>.
- [144] A. Bala, V. Sethi, E.L.E. Lo Gatto, V. Pachidis, P. Pilidis, PROOSIS—A Collaborative Venture for Gas Turbine Performance Simulation using an Object Oriented Programming Schema, (2007).
- [145] <https://www.ecosimpro.com/products/proosis/> (accessed 13 February 2024), (n.d.).
- [146] <https://www.plm.automation.siemens.com/global/en/products/simulation-test/propulsion-system-simulation.html> (access on 15-11-2022), (n.d.).
- [147] I. Roumeliotis, T. Nikolaidis, V. Pachidis, O. Broca, D. Unlu, DYNAMIC SIMULATION OF A ROTORCRAFT HYBRID ENGINE IN SIMCENTER AMESIM, in: 44th European Rotorcraft Forum, 2018: pp. 37–39.
- [148] Y. Liu, X. Sun, V. Sethi, D. Nalianda, Y.G. Li, L. Wang, Review of modern low emissions combustion technologies for aero gas turbine engines, Progress in Aerospace Sciences 94 (2017) 12–45. <https://doi.org/10.1016/j.paerosci.2017.08.001>.
- [149] A.H. Lefebvre, D.R. Ballal, Gas Turbine Combustion, CRC Press, 2010. <https://doi.org/10.1201/9781420086058>.
- [150] Z. Li, Aircraft engine transient performance modelling with heat soakage effects. Ph.D thesis, Cranfield University, 2019.
- [151] A. Kiss, Two Investigations of Compressor Stability : Spike Stall Inception and Transient Heat Transfer Effects, (2015).
- [152] https://en.wikipedia.org/wiki/Components_of_jet_engines (accessed 17 February 2024), (n.d.).

- [153] M. Vieweg, F. Wolters, R.-G. Becker, Comparison of a Heat Soakage Model With Turbofan Transient Engine Data, in: Volume 1: Aircraft Engine; Fans and Blowers; Marine; Honors and Awards, American Society of Mechanical Engineers, 2017: pp. 1–12. <https://doi.org/10.1115/GT2017-63461>.
- [154] V. Pachidis, Gas turbine advanced performance simulation, PhD Thesis (2006) 1–384.
- [155] W. van Antwerpen, Multi-Quadrant Performance Simulation for Subsonic Axial Flow Compressors, Master Thesis (2007) 1–176.
- [156] S. Lieblein, Loss and Stall Analysis of Compressor Cascades, Journal of Basic Engineering 81 (1959) 387–397. <https://doi.org/10.1115/1.4008481>.
- [157] C.C. Koch, L.H. Smith, Loss Sources and Magnitudes in Axial-Flow Compressors, Journal of Engineering for Power 98 (1976) 411–424. <https://doi.org/10.1115/1.3446202>.
- [158] Wright P I, Miller D C, An Improved Compressor Performance Prediction Model, (1992).
- [159] N. Falck, Axial Flow Compressor Mean Line Design, Lund University (2008) 119.
- [160] H.F. Creveling, R.H. Carmody, Axial Flow Compressor Computer Program for Calculating Off-Design Performance (Program IV), NASA Report CR-72427 (1968).
- [161] X. Zhang, Y. Ju, C. Zhang, Accuracy and sensitivity analysis of aerodynamic performance prediction models for transonic axial-flow compressors, Proceedings of the ASME Turbo Expo 2A-2020 (2020) 1–13. <https://doi.org/10.1115/GT2020-15172>.
- [162] Y. Liu, X. Sun, V. Sethi, Y.G. Li, D. Nalianda, D. Abbott, P. Gauthier, B. Xiao, L. Wang, Development and application of a preliminary design methodology for modern low emissions aero combustors, Proceedings of

- the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 235 (2021) 783–806. <https://doi.org/10.1177/0957650920919549>.
- [163] R. Kumar, V.S. Kumar, M.M. Butt, N.A. Sheikh, S.A. Khan, A. Afzal, Thermo-mechanical analysis and estimation of turbine blade tip clearance of a small gas turbine engine under transient operating conditions, Appl Therm Eng 179 (2020) 115700. <https://doi.org/10.1016/j.applthermaleng.2020.115700>.
- [164] L. Reid, R.D. Moore, Performance of Single stage Axial Transonic Compressor with rotor and stator aspect ratios of 1.19 and 1.26 Respectively and with design pressure ratio of 1.82, Nasa Technical Paper (1982).
- [165] P.N. Shah, Novel Turbomachinery Concepts for Highly Integrated Airframe / Propulsion Systems, Massachusetts Institute of Technology, PhD thesis, 2007.
- [166] <https://www.youtube.com/watch?v=6UZLIVk6vcQ> (accessed 26 February 2024), (n.d.).
- [167] Y. Liu, J. Zhang, D. Ju, Z. Huang, D. Han, Analysis of exergy losses in laminar premixed flames of methane/hydrogen blends, Int J Hydrogen Energy 44 (2019) 24043–24053. <https://doi.org/10.1016/j.ijhydene.2019.07.123>.

APPENDICES

Appendix A Heat Transfer Coefficient Estimation

The calculation of heat transfer coefficient is based on the Nusselt number. When calculating the heat transfer coefficient for compressor blades, casing and disc, it is assumed that the flow starts with laminar flow and then transfer into turbulence flow. The weighted average method is applied during this process. For laminar flow, the heat transfer coefficient is estimated by Eqs. (A-1), where N_u is the Nusselt number, h is the heat transfer coefficient, λ is the thermal conductivity of gas or air, L is the representative dimension of the component, Re is the Reynold number, Pr is the Prandtl number. For turbulence flow, the heat transfer coefficient is estimated by Eqs. (A-2). The average heat transfer coefficient is estimated by Eqs. (A-3), where f is the average coefficient. For turbine and combustor, it is assumed the flow is turbulence flow and Eqs. (A-4) is used for heat transfer coefficient estimation. The Reynolds number is estimated by using Eqs. (B-5), where V is the local velocity, ν is the kinematic viscosity. The Prandtl number is estimated by using Eqs. (B-6), where C_p is specific heat, and ρ is the static density of air or gas. The kinematic viscosity is estimated based on dynamic viscosity by using Eqs. (B-7). And the dynamic viscosity is estimated by using Sutherland's Law as shown in Eqs. (B-8), where μ_0 is reference viscosity, T_0 is reference temperature, and S_μ is Sutherland constant. The value of reference viscosity, reference temperature, and Sutherland constant is based on air property and the values are 1.716×10^{-5} (N*s/m²), 273 (K), and 111 (K).

$$N_u = \frac{h_L \cdot \lambda}{L} = 0.664(Re)^{0.5}(Pr)^{0.333} \quad (A-1)$$

$$N_u = \frac{h_T \cdot \lambda}{L} = 0.037(Re)^{0.8}(Pr)^{0.333} \quad (A-2)$$

$$h_{ave} = f \cdot h_L + (1 - f) \cdot h_T \quad (A-3)$$

$$N_u = \frac{h \cdot \lambda}{L} = 0.235(Re)^{0.64} \quad (A-4)$$

$$Re = \frac{V \cdot L}{\nu} \quad (A-5)$$

$$Pr = \frac{Cp \cdot v \cdot \rho}{\lambda} \quad (\text{A-6})$$

$$v = \frac{\mu}{\rho} \quad (\text{A-7})$$

$$\frac{\mu}{\mu_0} = \left(\frac{T}{T_0} \right)^{1.5} \cdot \frac{T_0 + S_\mu}{T + S_\mu} \quad (\text{A-8})$$

Appendix B Gas Property Calculation

The calculation of dry air Cp is based on the model provided in Walsh and Fletcher [65] according to which:

$$Cp = A_0 + A_1TZ + A_2TZ^2 + A_3TZ^3 + A_4TZ^4 + A_5TZ^5 + A_6TZ^6 + A_7TZ^7 + A_8TZ^8 \quad 1$$

where $TZ=T/1000$ and A are constants for dry air as shown below:

A_0	0.992313
A_1	0.236688
A_2	-1.852148
A_3	6.083152
A_4	-8.893933
A_5	7.097112
A_6	-3.234725
A_7	0.794571
A_8	-0.081873

The calculation of ratio of specific heats γ is calculated by using equation list below, where R is the gas constant.

$$\gamma = \frac{C_p}{C_p - R}$$

Appendix C Heat Transfer Model Implementation

The primary focus of this chapter is to describe the individual model algorithms developed for this investigation. In addition to detailing the design, structure, and execution of each algorithm, this chapter also discusses the general programming features and principles adopted in this work.

C.1 Creation of Input File

In this section, the required input file for heat transfer calculation is discussed. Two parts are included: the creation of .hhs input file, and the modification in .ttr file.

C.1.1 Creation of .hhs Input File

When Turbomatch is run in heat transfer modification mode, one file for heat transfer simulation settings needs to be created. This file has to have the same name as the engine input file and transient setting file, differing only by extension. For example, if the Turbomatch input file is named 'Example.dat', the transient input file will be named 'Example.ttr', and the heat transfer file will be named 'Example.hhs'.

In the .hhs file, the title section will be determined first. Any text typed at the beginning of the file is considered a comment for the user and will not be processed. An example of the title section is shown in.

```
Heat Transfer Effect Calculation Input File
Purpose: To test Turbomatch Heat Transfer Effect Routines
Author: Yimin Yang
```

Figure C-1 Example of .hhs title section

After the title section, the component heat transfer model data section is created. The CODEIN – CODEND input sequence concept is adopted during this data section. Three components will be considered in the data section: compressor, combustor, and turbine.

For each compressor, the data section starts with the CODEIN keyword 'COMPREXX', where XX represents the number of the compressor. For example, if the compressor number in the Turbomatch .dat input file is 1, then the keyword is 'COMPRE01'. The compressor number must match that in the .dat file. Following the CODEIN keyword, the heat transfer analysis data will be processed. For compressor, there are 21 heat transfer analysis data, as shown in Table C-1.

Table C-1 Compressor heat transfer analysis data

Data Number	Meaning
1	Compressor Shaft Radius (m) (Equals to the inner radius of disc)
2	Compressor Number of Stage
3	Compressor Inlet Hub/Tip Ratio
4	Design Point Compressor Inlet Mach Number
5	Annulus Configuration (1=Constant Hub Diameter; 2=Constant Tip Diameter; 3=Constant Mean Diameter)
6	Compressor Blade Aspect Ratio
7	Compressor Blade Pitch/Chord Ratio
8	Mean Blade Thickness (% of Chord)
9	Casing Thickness (m)
10	Blade Material Density (kg/m ³)

11	Casing Material Density (kg/m ³) (if =-1, the blade parameter is invoked)
12	Disc Material Density (kg/m ³) (if =-1, the blade parameter is invoked)
13	Blade Material Thermal Conductivity (W/(m*K))
14	Casing Material Thermal Conductivity (W/(m*K)) (if =-1, the blade parameter is invoked)
15	Disc Material Thermal Conductivity (W/(m*K)) (if =-1, the blade parameter is invoked)
16	Blade Material Specific Heat (J/Kg)
17	Casing Material Specific Heat (J/Kg) (if =-1, the blade parameter is invoked)
18	Disc Material Specific Heat (J/Kg) (if =-1, the blade parameter is invoked)
19	Blade Surface Heat Transfer Coefficient (W/(m ² *K)) (if =-1, means the heat transfer coefficient will be calculated automatically)
20	Casing Surface Heat Transfer Coefficient (W/(m ² *K)) (if =-1, means the heat transfer coefficient will be calculated automatically)
21	Disc Surface Heat Transfer Coefficient (W/(m ² *K)) (if =-1, means the heat transfer coefficient will be calculated automatically)

After the heat transfer analysis data, the CODEND keyword 'COMEND' should follow. The CODEND keyword represents the end of one compressor's heat transfer analysis data. If there are multiple compressors, a new data list can be created starting with a new CODEIN keyword based on the compressor number.

The same CODEIN - CODEND input sequence is set up for the combustor. In our case, the CODEIN keyword for the combustor is named 'BURNERXX' to maintain consistent character length. The heat transfer analysis data for the combustor is listed in Table C-2.

Table C-2 Combustor heat transfer analysis data

Data Number	Meaning
1	Combustor Type (1=Tubular; 2=Tuboannular; 3=Annular)
2	Combustor Length (m)
3	Number of Flame Tube (Only used for Tubular and Tuboannular. For annular combustor, -1 is used)
4	Flame Tube Wall Thickness (m)
5	Combustor Casing Inner Diameter (m)
6	Flame Tube Material Density (kg/m ³)
7	Flame Tube Material Thermal Conductivity (W/(m*K))
8	Flame Tube Material Specific Heat (J/Kg)
9	Flame Tube Surface Heat Transfer Coefficient (W/(m ² *K))

The CODEND keyword "BUREND" represents the end of combustor heat transfer analysis data.

For Turbine, the CODEIN keyword for the turbine is named after "TURBINXX". The heat transfer analysis data for the combustor is listed in Table C-3.

Table C-3 Turbine heat transfer analysis data

Data Number	Meaning
1	Turbine Shaft Radius (m) (Equals to the inner radius of disc)
2	Turbine Number of Stage

3	Turbine Inlet Hub/Tip Ratio
4	Design Point Turbine Inlet Mach Number
5	Annulus Configuration (1=Constant Hub Diameter; 2=Constant Tip Diameter; 3=Constant Mean Diameter)
6	Turbine Blade Aspect Ratio
7	Turbine Blade Pitch/Chord Ratio
8	Mean Blade Thickness (% of Chord)
9	Casing Thickness (m)
10	Thermal Barrier Coating Thickness (m) (if =0, means no thermal barrier coating)
11	Blade Material Density (kg/m ³)
12	Casing Material Density (kg/m ³) (if =-1, the blade parameter is invoked)
13	Disc Material Density (kg/m ³) (if =-1, the blade parameter is invoked)
14	Thermal Barrier Coating Material Density (kg/m ³)
15	Blade Material Thermal Conductivity (W/(m*K))
16	Casing Material Thermal Conductivity (W/(m*K)) (if =-1, the blade parameter is invoked)
17	Disc Material Thermal Conductivity (W/(m*K)) (if =-1, the blade parameter is invoked)
18	Thermal Barrier Coating Material Thermal Conductivity (W/(m*K))
19	Blade Material Specific Heat (J/Kg)

20	Casing Material Specific Heat (J/Kg) (if =-1, the blade parameter is invoked)
21	Disc Material Specific Heat (J/Kg) (if =-1, the blade parameter is invoked)
22	Thermal Barrier Coating Material Specific Heat (J/Kg)
23	Blade Surface Heat Transfer Coefficient (W/(m ² *K))
24	Casing Surface Heat Transfer Coefficient (W/(m ² *K))
25	Disc Surface Heat Transfer Coefficient (W/(m ² *K))
26	Thermal Barrier Coating Surface Heat Transfer Coefficient (W/(m ² *K))

The CODEND keyword “TUREND” represents the end of turbine heat transfer analysis data.

C.1.2 Modification of .ttr Input File

In the original .ttr file, there are 9 input setting data for the transient simulation. The Input sequence is shown in Figure C-2.

```

CODEIN
PRECED      2.      ! No. of preceding DP and OD simulation (initial DP calc.including)
INITIM      0.      ! Initial time [sec]
TRANGE     40.      ! Transient performance simulation duration [sec]
STEPLN     0.001   ! Length of one transient step [sec]
FSCHED      2       ! Type of the fuel
FSTBLE     14       ! No. of records in fuel schedule table
PRINTS     2,50,20 ! Results plotting preferences
BDTRAN     D1,3     ! Brick data which are changed during TR simulation (e.q. D2,27)
SVTRAN      ! Station vectors which are changed during TR simulation (e.q. S5V6)
CODEND

```

Figure C-2 Turbomatch .ttr file original input sequence

Because the heat transfer effect only affects the gas turbine transient performance, one additional input setting data is added to the original .ttr input sequence. The information of the added setting data is shown in Table C-4.

Table C-4 Added transient .ttr file input sequence

Name of setting data	Meaning
HSANLY	Heat Transfer Analysis: 0 = Heat Transfer is not Included; 1 = Heat Transfer is Included.

C.2 Heat Transfer Effect Algorithm

After creating the .hhs input file, Turbomatch can then initiate the heat transfer effect analysis during the gas turbine transient simulation. This section will discuss the algorithm for the heat transfer effect analysis.

The reading process of the .hhs input data begins with the input file check. Turbomatch will first check all the input files. If a .hhs file with the same name as the .dat file is found, the heat transfer effect file find indicator will be set to 1, indicating that the .hhs file is found. Then, the .hhs file needs to be read to store the user-defined parameters. When the heat soakage file is read, a module named 'M_ClassHeatSoakageIO.f90' will be called. The purpose of this module is to read the .hhs input file line by line and store the data for each component. The heat transfer effect data for each component will be stored in their respective data arrays. Information about these arrays is listed in files named 'M_ClassCompreStructure.f90', 'M_ClassBurnerStructure.f90', and 'M_ClassTurbinStructure.f90'. The flow chart for the file reading and storing process is shown in Figure C-3.

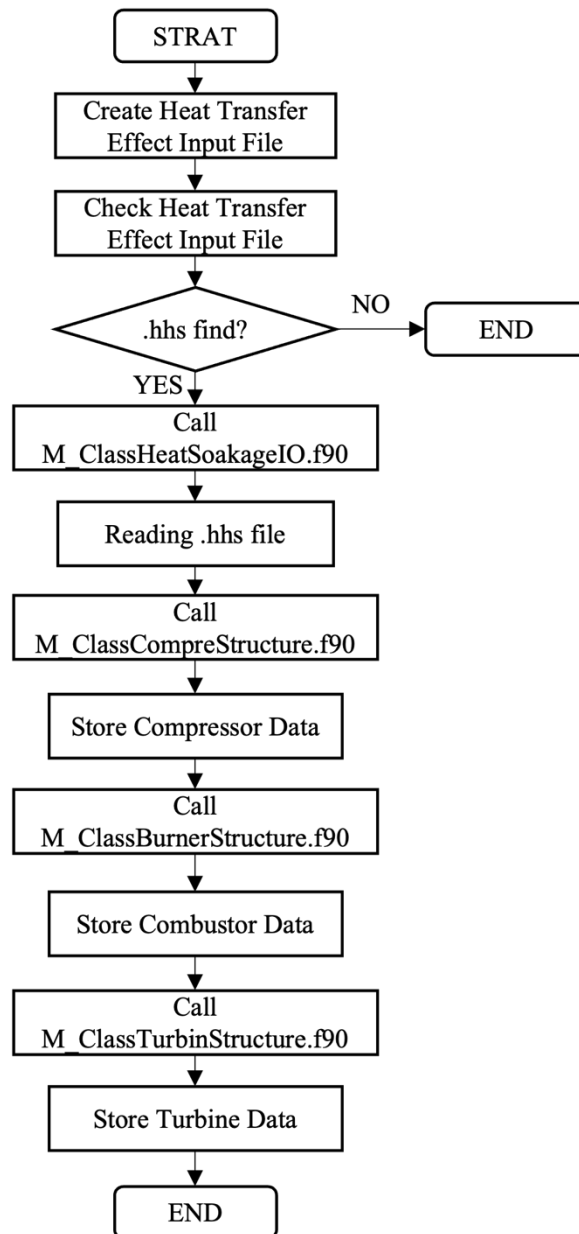


Figure C-3 Flow chart for .hhs file reading and storing process

After reading and storing the .hhs file information, Turbomatch is ready for heat transfer effect calculation. If the HSANLY parameter in the .ttr file is set to 1, the heat transfer effect analysis handler will also be set to 1, indicating that the heat transfer effect will be calculated. Otherwise, the heat transfer effect will not be calculated during the entire simulation. As shown in Figure 3-1, design point, off-design point and transient simulation will include different heat transfer effect simulation steps.

In any condition, Turbomatch will simulate the gas turbine design point performance first. During the design point, a global handle will indicate the design point condition. For each component, geometry calculations will be performed during the design point simulation. The module named 'M_ClassHSCompStruCalc.f90' includes the geometry calculation subroutines for the compressor, combustor, and turbine. The subroutine 'CompressorStaging' is called by the module 'BricksA_33_COMORE.f90'. The subroutine 'BurnerStructure' is called by the module 'BricksA_34_BURNER.f90'. The subroutine 'TurbineStaging' is called by the module 'BricksA_35_TURBIN.f90'. The methodologies discussed in Section 3.2 are integrated to estimate the geometry of each component during the design point calculation. Apart from the geometry calculation, the datum condition of the tip clearance calculation is also initiated during the design point calculation. Based on the design point temperature, pressure, and rotational speed, the datum condition will be calculated using the subroutines named 'TipClearanceInitialConditionCompre' and 'TipClearanceInitialConditionTurbin'. These two subroutines are called by 'BricksA_33_COMORE.f90' and 'BricksA_35_TURBIN.f90'.

After simulating the design point working condition, Turbomatch will enter the off-design condition simulation. Users can set several off-design simulations in the .dat file. However, for the heat transfer effect simulation, only the last off-design condition is important as it represents the initial heat transfer calculation condition. Under this condition, the initial temperature calculation cannot be directly called by 'BricksA_33_COMORE.f90', 'BricksA_34_BURNER.f90', and 'BricksA_35_TURBIN.f90' because calling the initial temperature calculation at each off-design calculation would significantly increase the calculation time. To accurately perform the initial temperature calculation at the last off-design condition, the module named 'SR_OUTSEQ.f90' is selected as the integration file. Within this module, a counter indicates the current off-design condition number. When it reaches the last off-design condition, the 'M_ClassComlInitTemp.f90' module will be called. This module includes the subroutines 'CompressorInitialTemp' and 'TurbineInitialTemp' for the compressor and turbine, respectively. For the combustor, the 'M_ClassBurnerZoneTempCalc.f90' module

is utilized. By using the methodology described in Section 3.3.4, the initial temperature for each component will be estimated. The initial temperature outcome for heat soakage estimation will be stored in component array.

After completing the off-design conditions, Turbomatch enters the transient simulation phase. During transient simulation, 'BricksA_33_COMORE.f90' will call 'M_ClassHSCalculation.f90'. Within 'M_ClassHSCalculation.f90', subroutines named 'CompressorHeatSoakage', 'BurnerHeatSoakage', and 'TurbineHeatSoakage' are included. These subroutines are called by 'BricksA_33_COMORE.f90', 'BricksA_34_BURNER.f90', and 'BricksA_35_TURBIN.f90'. Additionally, the tip clearance will be calculated using subroutines named 'TipClearanceCompre' and 'TipClearanceTurbin'. For the map modification model, it is directly integrated into 'BricksA_33_COMORE.f90'. The component performance modification methods described in Section 3.3.3, Section 3.4.2, and Section 3.5.3 will be used to modify the component performance.

The flow chart for the design point, off-design point and transient calculation logic is shown in Figure C-4.

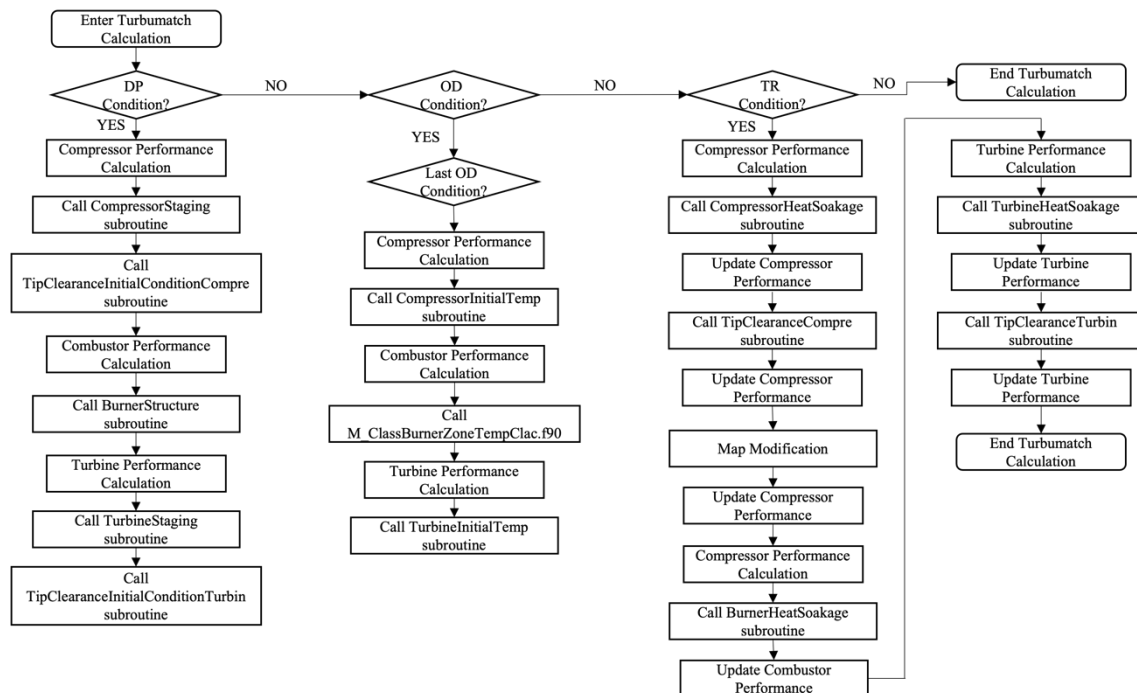


Figure C-4 Flow chart for heat transfer effect calculation process

Appendix D Sensitivity Analysis on Burner Time Constant

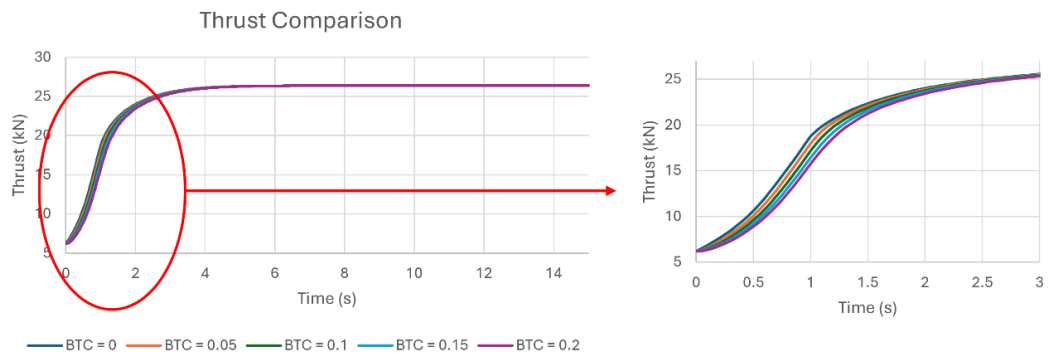
As mentioned in Section 5.1.4.2, Turbomatch does not consider the burner time constant during the transient simulation when GasTurb considered this feature. When validating Turbomatch transient simulation accuracy, the burner time constant was set to 0 in GasTurb to keep the consistency.

Based on the literature review, the burner time constant considers delays in heat release like the time needed for evaporating the fuel. To investigate the effect coming from burner time constant, a sensitivity analysis is launched by using GasTurb.

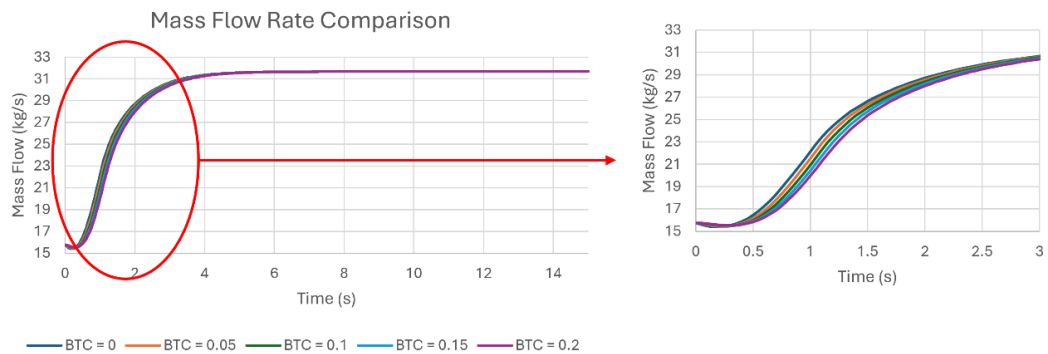
The gas turbine model described in Section 4.2.1 and transient fuel flow setting described in Section 4.2.5.1 were used during the sensitivity analysis. After a literature review for burner time constant, there was no open literature discuss the setting logic of the burner time constant and no detailed information was provided by GasTurb manual. So, the default value of burner time constant equals to 0.1 provided by GasTurb was chosen as the baseline value. Several calculations were performed with modified burner time constant. Specifically, default burner time constant was multiplied by factors of 0, 0.5, 1.5 and 2.0. The thrust, mass flow rate and combustor outlet temperature lines are shown in Figure D-1 (a) to (d).

The burner time constant will directed affect the combustor outlet temperature during the transient manoeuvre, as shown in Figure D-1 (d). The greater the burner time constant, the slower the combustor outlet temperature change rate. And due to the mass flow increases during the transient, the maximum combustor outlet temperature is different. For the default condition (burner time constant equals to 0.1), the maximum combustor outlet temperature is 1680.7 K at 1.05 s simulation time. For the no burner time constant condition, the maximum combustor outlet temperature is 1706 K at 1.0 s simulation time. An investigation on thrust, mass flow rate, and corrected rotational speed increase delay is also analyzed. Based on Figure D-1, a linear relation is found between the delay in

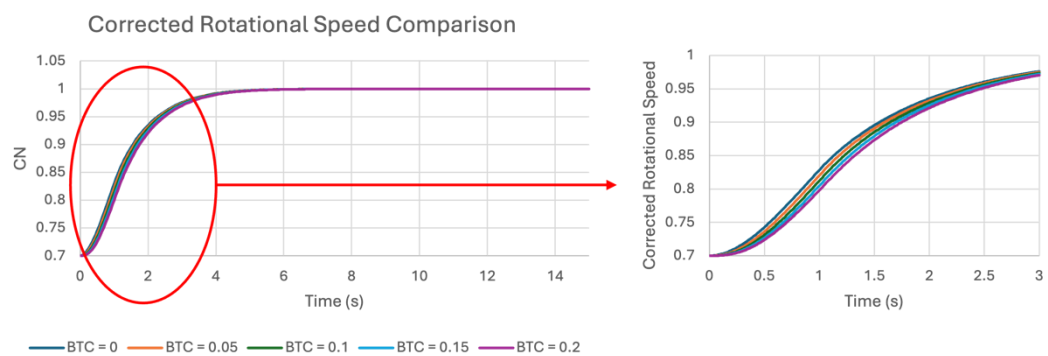
acceleration time and burner time constant value. Every 0.05 increase in burner time constant value will lead to 0.05 delay in reaching 100% of working condition.



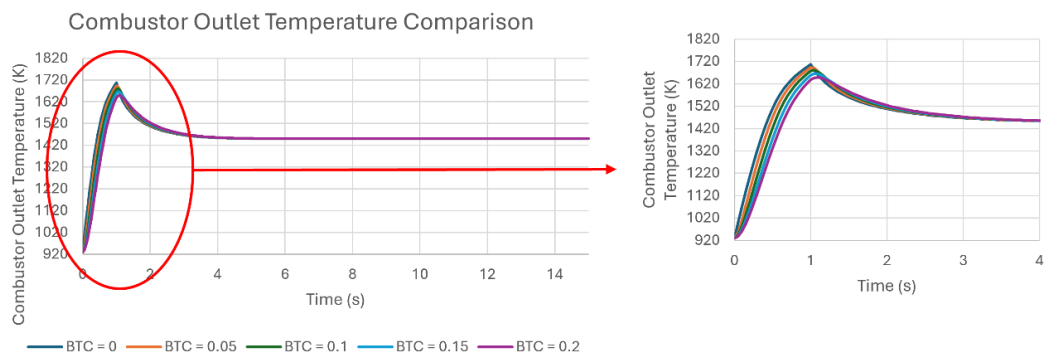
(a)



(b)



(c)



(d)

Figure D-1 Burner time constant effect on gas turbine transient performance

Table D-1 Burner time constant sensitivity analysis outcome

Parameter	BTC=0	BTC=0.05	BTC=0.1	BTC=0.15	BTC=0.2
Time to 50% Thrust Increase (s)	0.68	0.73	0.77	0.81	0.85
Difference (s)	-0.09	-0.04	0	0.04	0.08
Time to 100% Thrust Increase (s)	6.8	6.85	6.9	6.95	7.0
Difference (s)	-0.1	-0.05	0	0.05	0.1
Time to 50% Mass Flow Increase (s)	0.43	0.47	0.5	0.52	0.54
Difference (s)	-0.07	-0.03	0	0.02	0.04
Time to 100% Mass Increase Flow (s)	5.25	5.3	5.35	5.4	5.45
Difference (s)	-0.1	-0.05	0	0.05	0.1
Time to 50% CN Increase (s)	1.13	1.18	1.23	1.28	1.32
Difference (s)	-0.1	-0.05	0	0.05	0.09
Time to 100% CN Increase (s)	6	6.05	6.1	6.15	6.2
Difference (s)	-0.1	-0.05	0	0.05	0.1

Appendix E Component Stage Tip Clearance Change

E.1 Fan Tip Clearance Change

The tip clearance change of fan is shown in Figure E-1.

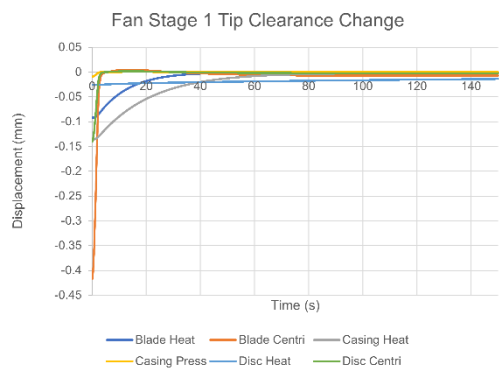
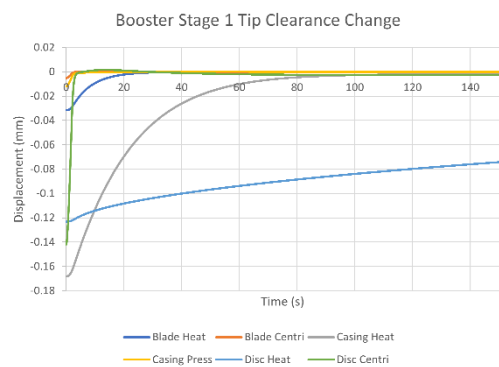


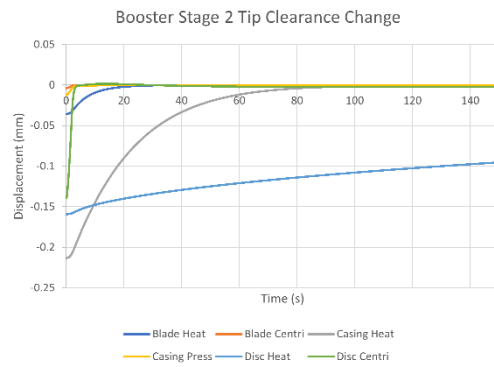
Figure E-1 Fan tip clearance movement

E.2 Booster Tip Clearance Change

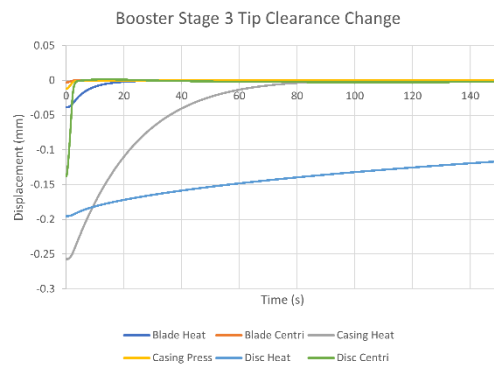
The 1st to 3rd stage's tip clearance change of each stage of booster is shown in Figure E-2 (a) to (c).



(a)



(b)

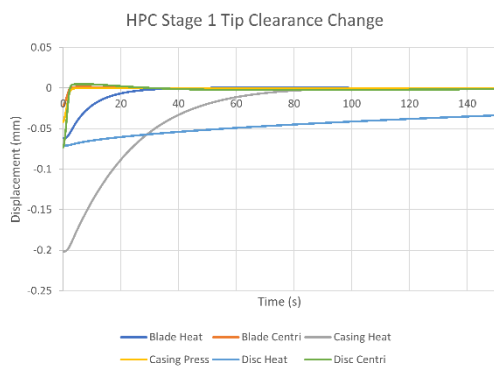


(c)

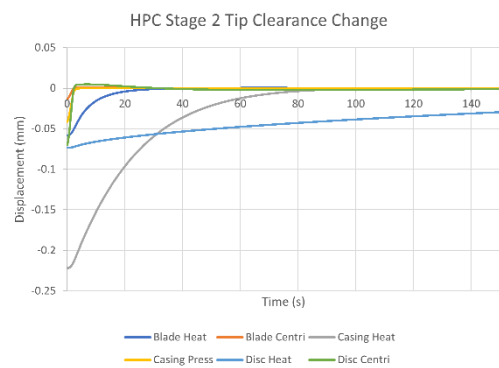
Figure E-2 Booster tip clearance movement

E.3 HPC Tip Clearance Change

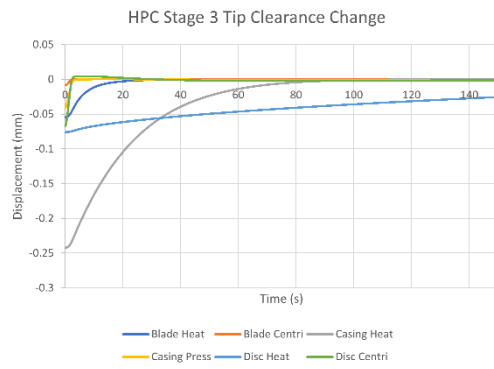
The 1st to 9th stage's tip clearance change of HPC is shown in Figure E-3 (a) to (i).



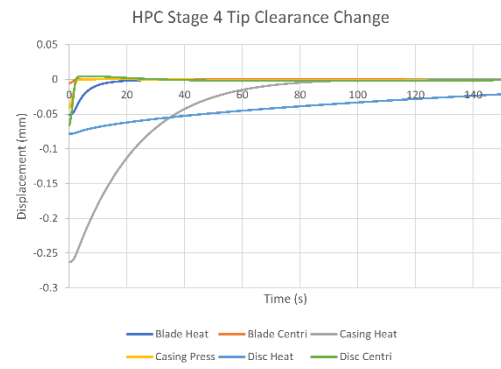
(a)



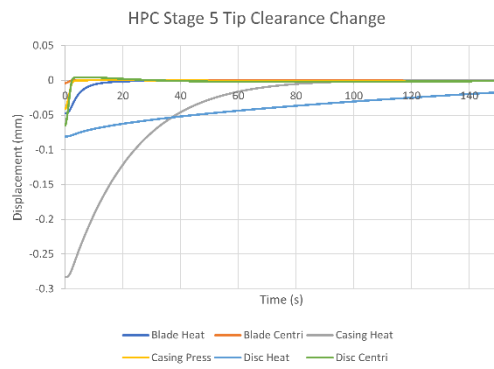
(b)



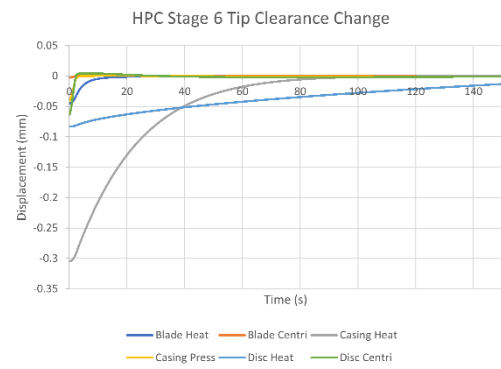
(c)



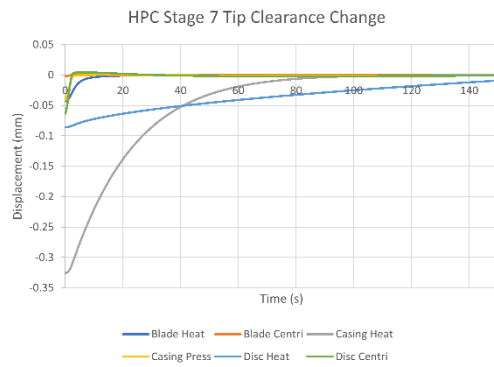
(d)



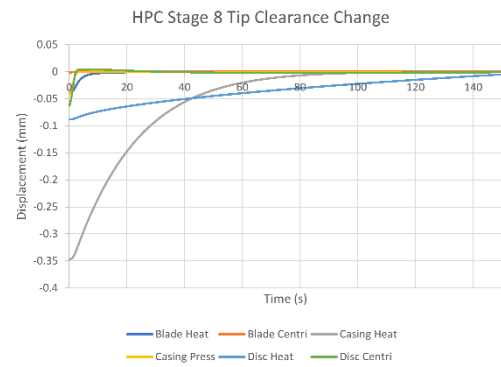
(e)



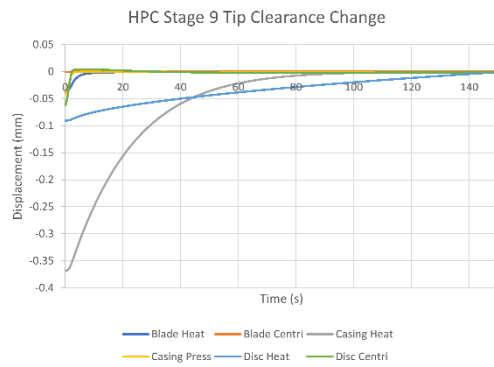
(f)



(g)



(h)



(i)

Figure E-3 HPC tip clearance movement

E.4 HPT Tip Clearance Change

The tip clearance change of HPT is shown in Figure E-4.

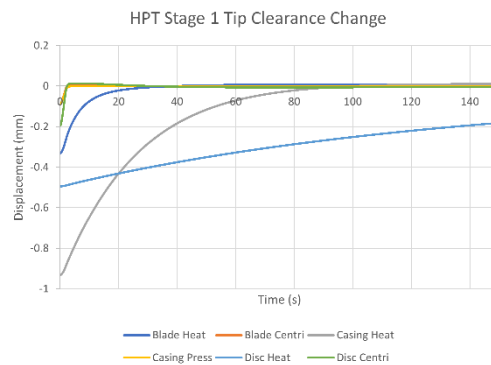
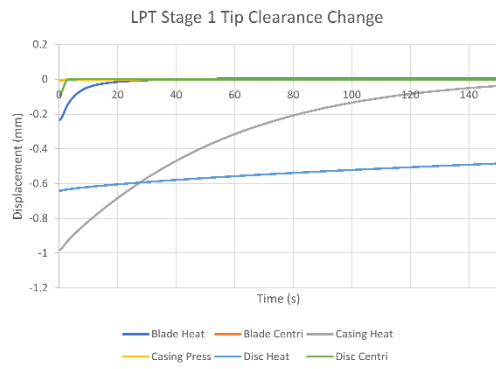


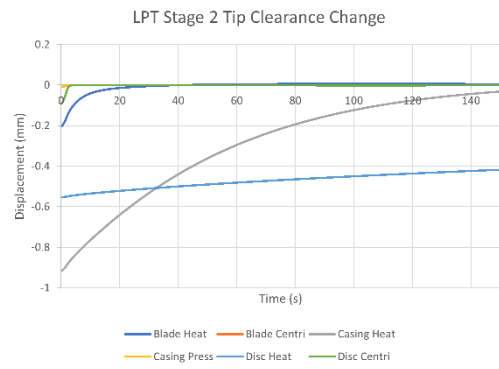
Figure E-4 HPT tip clearance movement

E.5 LPT Tip Clearance Change

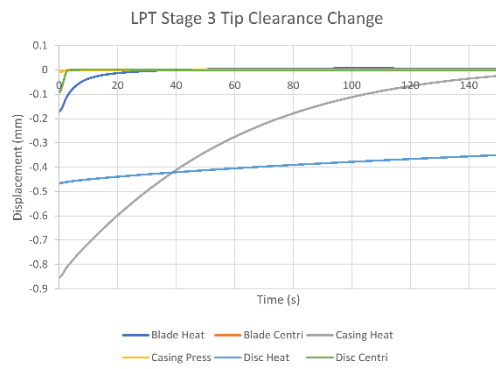
The 1st to 4th stage's tip clearance change of LPT is shown in Figure E-5.



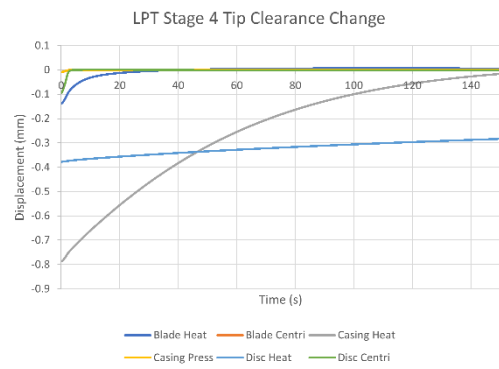
(a)



(b)



(c)



(d)

Figure E-5 LPT tip clearance movement