

ISTANBUL TECHNICAL UNIVERSITY ★ GRADUATE SCHOOL

**CALIBRATION AND OPTIMIZATION OF RANS TURBULENCE MODELS
FOR VARIOUS CASES**



Ph.D. THESIS

Cem TURUTOĞLU

Department of Mechanical Engineering

Mechanical Engineering Programme

FEBRUARY 2026

ISTANBUL TECHNICAL UNIVERSITY ★ GRADUATE SCHOOL

**CALIBRATION AND OPTIMIZATION OF RANS TURBULENCE MODELS
FOR VARIOUS CASES**

Ph.D. THESIS

**Cem TURUTOĞLU
(503182032)**

Department of Mechanical Engineering

Mechanical Engineering Programme

Thesis Advisor : Assoc. Prof. Dr. Sertaç ÇADIRCI

FEBRUARY 2026

İSTANBUL TEKNİK ÜNİVERSİTESİ ★ LİSANSÜSTÜ EĞİTİM ENSTİTÜSÜ

**FARKLI PROBLEMLER İÇİN RANS TÜRBÜLANS MODELLERİNİN
KALİBRASYONU VE OPTİMİZASYONU**

DOKTORA TEZİ

**Cem TURUTOĞLU
(503182032)**

Makina Mühendisliği Anabilim Dalı

Makina Mühendisliği Programı

Tez Danışmanı : Doç. Dr. Sertaç ÇADIRCI

ŞUBAT 2026

Cem TURUTOĞLU, a Ph.D. student of ITU Graduate School student ID 503182032, successfully defended the dissertation entitled “CALIBRATION AND OPTIMIZATION OF RANS TURBULENCE MODELS FOR VARIOUS CASES”, which he/she prepared after fulfilling the requirements specified in the associated legislations, before the jury whose signatures are below.

Thesis Advisor : **Assoc. Prof. Dr. Sertaç ÇADIRCI**
Istanbul Technical University

Jury Members : **Prof. Dr. Bayram ŞAHİN**
Istanbul Technical University

Prof. Dr. Hakan DEMİR
Yildiz Technical University

Assoc. Prof. Dr. Gamze GEDİZ İLİŞ
Gebze Technical University

Assist. Prof. Dr. Duygu ERDEM
Istanbul Technical University

Date of Submission : 04 February 2026

Date of Defense : 17 February 2026





To my family,



FOREWORD

I extend my deepest gratitude to Assoc. Prof. Dr. Sertaç ÇADIRCI, my supervisor for the past ten years, for his unwavering guidance, patience, and high standards, which have shaped both this dissertation and my approach to research. I also thank to thesis committee members, Assoc. Prof. Dr. Gamze GEDİZ İLİŞ and Assist. Prof. Dr. Duygu ERDEM, whose insightful comments, constructive criticisms, and steady direction were instrumental in refining the scope, methods, and conclusions of this work.

I am indebted to Serdar YILMAZ for conducting the jet flow experiments and sharing the measurement data in exemplary order, and to Cansu KUZUGÜDEN for carrying out and providing the experimental measurements used in the curved backward-facing step study. I gratefully acknowledge Prof. Dr. Ahmet ÜÇER for his valuable perspectives on the axial compressor cascade studies and for facilitating access to AGARD reports. I further thank Dr. Ferit BOYSAN for his generous advice and guidance during the turbulence model development process, and for the critical knowledge he shared that significantly strengthened this research. Financial support for the model calibration studies on the curved backward-facing step problems from the Scientific Research Projects Department of Istanbul Technical University (Project Number: MGA-2025-46465) is gratefully acknowledged.

I am sincerely thankful to Research Assistant Alperen YILDIZELİ for his friendship and for the illuminating discussions that consistently steered the turbulence modeling efforts in the right direction. I express heartfelt thanks to my mother Fatma, father Kadir, and my brothers Cenk and Cihan, for their unfailing support, which made this journey possible. And the most importantly, I extend my deepest appreciation to my beloved wife, Diana SATTAROVA TURUTOĞLU, whose strength and steadfast support carried me through difficult times and with whom I have overcome many challenges.

February 2026

Cem TURUTOĞLU
(Mechanical Engineer)



TABLE OF CONTENTS

	<u>Page</u>
FOREWORD.....	ix
TABLE OF CONTENTS.....	xi
ABBREVIATIONS	xiii
SYMBOLS.....	xv
LIST OF TABLES	xix
LIST OF FIGURES	xxi
SUMMARY	xxv
ÖZET	xxix
1. INTRODUCTION.....	1
2. IMPROVEMENT OF STANDARD K-EPSILON TURBULENCE MODEL FOR ROUND FREE JETS BY ADJUSTING CLOSURE COEFFICIENTS 9	
2.1 Introduction	9
2.2 Methodology.....	11
2.2.1 Problem description	11
2.2.2 Computational grid	12
2.2.3 Initial and boundary conditions	12
2.2.4 Governing equations	13
2.2.5 Calibration of closure coefficients	14
2.3 Results and Discussion	15
2.3.1 Calibration of model coefficients.....	15
2.4 Conclusion.....	24
3. CALIBRATION OF THE REYNOLDS STRESS MODEL FOR TURBU- LENT ROUND FREE JETS BASED ON JET HALF-WIDTH.....	25
3.1 Introduction	25
3.2 Methodology.....	34
3.2.1 Experimental setup.....	34
3.2.1.1 Measurement locations	36
3.2.2 Numerical model.....	37
3.2.2.1 Problem description	37
3.2.2.2 Boundary conditions.....	38
3.2.2.3 Governing equations	39
3.2.2.4 Grid study	41
3.2.2.5 Optimization process	43
3.3 Results	44
3.3.1 Sensitivity analysis.....	44
3.3.2 Calibration of closure coefficients	47
3.3.2.1 Results of calibration study	47
3.3.3 Budget of turbulence kinetic energy	54

3.3.4 Normalized dissipation rate and Kolmogorov length scale along jet centerline.....	56
3.3.5 Results for $Re=20000$	58
3.4 Discussion.....	62
3.5 Conclusion.....	67
4. A NEW VARIANT OF RENORMALIZATION GROUP TURBULENCE MODEL FOR FLOWS IN AXIAL COMPRESSOR CASCADE.....	69
4.1 Introduction	69
4.2 Methodology.....	76
4.2.1 V2 compressor cascade.....	76
4.2.2 Numerical method.....	77
4.2.3 Initial and boundary conditions	81
4.2.4 Grid structure	81
4.2.5 Definition of RNG $k - \varepsilon$ modification	83
4.3 Results and Discussion	84
4.3.1 V2 cascade off-design conditions	84
4.3.2 V2 cascade design condition.....	97
4.3.3 V103 cascade	99
4.4 Conclusion.....	104
5. ENHANCING THE K-ω SHEAR STRESS TRANSPORT TURBULENCE MODEL THROUGH RENORMALIZATION GROUP THEORY FOR FLOWS SEPARATED FROM A CONTINUOUS SURFACE	107
5.1 Introduction	107
5.2 Formulation of RNG-SST Model.....	112
5.3 Methodology.....	115
5.3.1 Numerical model.....	115
5.3.2 Description of test cases.....	117
5.3.3 Experimental campaign	120
5.4 Results and Discussion	121
5.4.1 Flow over curved backward-facing step	122
5.4.1.1 Assessment of inflow boundary conditions	123
5.4.1.2 Model results	131
5.4.2 Other test cases	137
5.4.2.1 Flow over backward-facing step.....	137
5.4.2.2 Flow over periodic hills	143
5.4.3 Experimental investigation on backward-facing ramp.....	146
5.5 Conclusion.....	154
6. CONCLUSIONS AND RECOMMENDATIONS	157
REFERENCES.....	161
CURRICULUM VITAE.....	179

ABBREVIATIONS

AC	: Application Challenge
ADE	: Ansys DesignXplorer
AGARD	: The Advisory Group for Aerospace Research and Development
AMO	: Adaptive Multi-Objective
APG	: Adverse Pressure Gradient
Approx	: Approximate
ATAAC	: Advanced Turbulence Simulation for Aerodynamic Application Challenges
AVDR	: Axial Velocity Density Ratio
BFS	: Backward-Facing Step
CAD	: Computer Aided Design
CBFS	: Curved Backward-Facing Step
CFD	: Computational Fluid Dynamics
DCA	: Double Circular Arc
DNS	: Direct Numerical Simulation
EARSM	: Explicit Algebraic Reynolds Stress Model
EKF	: Ensemble Kalman Filter
ERCOTAC	: European Research Community on Flow, Turbulence and Combustion
Err	: Error
EVM	: Eddy Viscosity Model
EWI	: Enhanced Wall Treatment
Exp	: Experimental
FIML	: Field Inversion and Machine Learning
HPC	: High Performance Computing
IP	: Isotropization of Production
KBwiki	: Knowledge Base Wiki
LDA	: Laser Doppler Anemometry
LES	: Large-Eddy Simulation
LRR	: Launder, Reece and Rodi
Ma	: Mach Number
MOGA	: Multi-Objective Genetic Algorithm
NASA	: National Aeronautics and Space Administration
N-S	: Navier-Stokes
PD	: Pressure Diffusion
PDE	: Partial Differential Equation
PINN	: Physics-Informed Neural Network
PIV	: Particle Image Velocimetry
RANS	: Reynolds-Averaged Navier-Stokes
Re	: Reynolds Number
Reatt	: Reattachment
Rel	: Relative
RMS	: Root-Mean-Square

RMSE	: Root-Mean-Square Error
RNG	: Renormalization Group
RSM	: Reynolds Stress Model
SA	: Spalart-Allmaras
Sc	: Scalable
Sep	: Separation
SIG	: Special Interest Group
SIMPLE	: Semi-Implicit Method for Pressure Linked Equations
SSG	: Speziale, Sarkar and Gatski
SST	: Shear Stress Transport
STT	: Stream Tube Thickness
TD	: Turbulent Diffusion
TDR	: Turbulence Dissipation Rate
TI	: Turbulence Intensity
TKE	: Turbulence Kinetic Energy
UFR	: Underlying Flow Regime
WF	: Wall Function
WG	: Working Group

SYMBOLS

a_1	: Shear Stress Transport Model Constant
arg_1	: Argument 1
arg_2	: Argument 2
B	: Jet Velocity Decay
C_1	: RSM Constant
C_2	: RSM Constant
C_D	: Drag Coefficient
C_L	: Lift Coefficient
C_P	: Pressure Coefficient
C_S	: RSM Constant
C_f	: Skin Friction Coefficient
C_ε	: RSM Constant
$C_{\varepsilon 1}$	: Turbulence Model Constant
$C_{\varepsilon 1 RNG}$	: RNG $k - \varepsilon$ Model Constant
$C_{\varepsilon 2}$	: Turbulence Model Constant
C_μ	: Turbulence Model Constant
C_v	: Effective Viscosity Equation Constant
CD_{kw}	: Shear Stress Transport Model Argument
D	: Diameter
d	: Profile Thickness
e	: Outlet Plane Axial Distance
F_1	: First Blending Function of Shear Stress Transport Model
F_2	: Second Blending Function of Shear Stress Transport Model
$F_{Reattach}$	: Blending Function of RNG-SST Model
H	: Shape Factor
H_R	: Step Height
h	: Enthalpy
$h_{Tot.}$	: Total Enthalpy
k	: Turbulence Kinetic Energy
l	: Chord Length or Turbulent Length Scale
l_{ax}	: Axial Chord Length
L	: Characteristic Length
L_e	: Height of the Inflow Boundary
Ma_{is}	: Isentropic Mach Number
P	: Static Pressure
P'	: Fluctuating Pressure
P''	: Fluctuating Pressure
P_0	: Total Pressure
P_{01}	: Inlet Plane Total Pressure
P_{02}	: Outlet Plane Total Pressure
P_1	: Inlet Plane Static Pressure

P_2	: Outlet Plane Static Pressure
P_{ij}	: Exact Production Term
P_k	: Production Term
$\bar{p}$	: Static Pressure
R	: RNG $k - \varepsilon$ Model Source Term
$r_{1/2}$	: Jet Half-Width
Re_D	: Reynolds Number Based on Jet Diameter
Re_{cr}	: Critical Reynolds Number
Re_θ	: Reynolds Number Based on Momentum Thickness
S	: Shear Strain Rate or Jet Spreading Rate
T	: Static Temperature
t/l	: Pitch-to-Chord Ratio
U	: Mean Velocity
U_0	: Mean Velocity at Nozzle
U_b	: Bulk Velocity
U_c	: Centerline Mean Velocity
$U_{Ref.}$	: Reference Velocity
U_∞	: Freestream Velocity
u^*	: Velocity Scale
u_τ	: Friction Velocity
u'_{RMS}	: Axial Component of Root-Mean-Squared Fluctuating Velocity
v'	: Fluctuating Velocity
$\bar{v}'$	: Averaged Fluctuating Velocity
$\bar{v}$	: Averaged Velocity
$v_{i,j,k}$	: Velocity Components
v'_{RMS}	: Longitudinal Component of Root-Mean-Squared Fluctuating Velocity
w'_{RMS}	: Lateral Component of Root-Mean-Squared Fluctuating Velocity
x	: Axial Direction
x_0	: Virtual Jet Origin
$x_{i,j,k}$	: Coordinate Component
y	: Longitudinal Direction
y^*	: Dimensionless Wall Distance
y^+	: Dimensionless Wall Distance
z	: Lateral Direction
α	: Flow Angle
α_1	: Inflow Angle or Shear Stress Transport Model Constant
α_2	: Outflow Angle or Shear Stress Transport Model Constant
α_3	: Shear Stress Transport Model Constant
β^*	: Shear Stress Transport Model Constant
β_1	: Profile Inflow Angle or Shear Stress Transport Model Constant
β_2	: Profile Outflow Angle or Shear Stress Transport Model Constant
β_3	: Shear Stress Transport Model Constant
β_{RNG}	: RNG $k - \varepsilon$ Model Constant
β_s	: Profile Stagger Angle
γ	: Intermittency
δ_{ij}, δ_{kl}	: Kronecker Delta
δ_0	: Boundary Layer Thickness
δ^*	: Displacement Thickness

ε	: Turbulence Dissipation Rate
ε^*	: Normalized Dissipation Rate
η	: RNG $k - \varepsilon$ Model Constant or Kolmogorov Length Scale
η_0	: RNG $k - \varepsilon$ Model Constant
θ	: Momentum Thickness
κ	: von Kármán Constant
λ	: Thermal Conductivity
μ	: Dynamic Viscosity
μ_t	: Turbulent Dynamic Viscosity
$\mu_{eff.}$	: Effective Viscosity
ν	: Kinematic Viscosity
ν_t	: Turbulent Kinematic Viscosity
$\hat{\nu}$	: Viscosity Ratio
ρ	: Density
σ_e	: Turbulence Model Constant
σ_k	: Turbulence Model Constant
σ_{k1}	: Shear Stress Transport Model Constant
σ_{k2}	: Shear Stress Transport Model Constant
σ_{k3}	: Shear Stress Transport Model Constant
$\sigma_{\omega 1}$	: Shear Stress Transport Model Constant
$\sigma_{\omega 2}$	: Shear Stress Transport Model Constant
$\sigma_{\omega 3}$	: Shear Stress Transport Model Constant
τ_{ij}	: Wall Shear Stress
ϕ	: Profile Camber
Φ_{ij}	: Pressure-Strain Correlation
$\Phi_{1,2,3}$	: Shear Stress Transport Model Coefficient Function
ω	: Turbulence Dissipation Frequency
ω_o	: Total Pressure Loss Coefficient at Outlet Plane
$\bar{\omega}$	: Total Pressure Loss Coefficient



LIST OF TABLES

	<u>Page</u>
Table 2.1 : Cases with modified coefficients.	16
Table 3.1 : Measurement locations for $Re=10000$	36
Table 3.2 : Relative errors of the decay constant (B) and spreading rate (S) for different grids.	42
Table 3.3 : Sensitivities of the closure coefficients.	46
Table 3.4 : Decay constant and spreading rate results.	54
Table 4.1 : Geometric parameters of V2 cascade.	76
Table 4.2 : Operating conditions of test cases.	77
Table 4.3 : Model coefficients.	79
Table 4.4 : Results of the mesh independence study using SST turbulence model.	82
Table 4.5 : RNG $k - \varepsilon$ and standard $k - \varepsilon$ with RNG source term (RNG - $sk\varepsilon$ RNG) results comparison.	85
Table 4.6 : Results comparison of low y^+ grid and errors with respect to experimental measurements.	86
Table 4.7 : Results comparison of high y^+ grid and errors with respect to experimental measurements.	91
Table 4.8 : Results comparison of Exp. 4 and errors with respect to experimental measurements.	98
Table 4.9 : Design parameters of V103 axial compressor cascade.	101
Table 5.1 : Details of the benchmark cases.	117
Table 5.2 : Inflow boundary conditions for turbulence eddy frequency equation of SST model.	124
Table 5.3 : Comparison of estimated separation and reattachment points for different inlet boundary conditions in the CBFS case and relative errors with benchmark LES.	127
Table 5.4 : Predicted separation and reattachment points in the CBFS case and relative errors with benchmark LES.	132
Table 5.5 : Reattachment positions predicted in the present study and their relative errors with the experimental measurements for the BFS cases.	139
Table 5.6 : Comparison of the estimated reattachment points in the present study with the benchmark studies of periodic hills case.	145
Table 5.7 : Boundary layer parameters calculated with measured velocity along the wall-normal direction at each station.	147
Table 5.8 : Separation and reattachment locations of experimental campaign on curved backward-facing step.	152



LIST OF FIGURES

	<u>Page</u>
Figure 2.1 : Computational domain.....	11
Figure 2.2 : Computational grid.	12
Figure 2.3 : Points selected for optimization.....	14
Figure 2.4 : Velocity profiles for Re=10000.....	17
Figure 2.5 : TI distributions for Re=10000.	18
Figure 2.6 : Jet centerline velocity distribution for Re=10000.....	19
Figure 2.7 : Jet half-width distribution for Re=10000.....	19
Figure 2.8 : Velocities with calibrated coefficients for Re=10000.	21
Figure 2.9 : Velocities at Re=20000 with adjusted coefficients.	22
Figure 2.10 : Jet centerline velocity distributions for Re=10000 with adjusted coefficients.	23
Figure 2.11 : Jet centerline velocity distribution for Re=5000 with adjusted coefficients.	23
Figure 2.12 : Jet centerline velocity distribution for Re=20000 with adjusted coefficients.	24
Figure 3.1 : Experimental setup.....	36
Figure 3.2 : Computational domain of round jet.	37
Figure 3.3 : Jet exit profiles for Re=10000 (a) velocity profiles (b) TI profiles.	39
Figure 3.4 : Mesh independence results (a) for jet half-widths (b) for jet centerline velocity distributions.	42
Figure 3.5 : Computational grid structure.	42
Figure 3.6 : Optimization diagram.	45
Figure 3.7 : Variation of $C_{\epsilon 1}$ constant at 1050 different examinations.....	47
Figure 3.8 : Convergence stability percentage of optimization.	48
Figure 3.9 : Jet half-width results along the jet centerline for Re=10000.	49
Figure 3.10 : Velocity profiles along jet axis for Re=10000.	51
Figure 3.11 : TI along the jet centerline for Re=10000.....	52
Figure 3.12 : TI profiles along jet axis for Re=10000.	53
Figure 3.13 : Jet centerline velocity distribution for Re=10000.....	54
Figure 3.14 : TKE budget for Re=11000. Symbols: $\circ$, Panchapakesan and Lumley [97]; $\times$, Balajee and Panchapakesan [100], - - - (dashed lines), current numerical study with default closure coefficients; — (solid lines), current numerical study with calibrated closure coefficients. Colors: orange, advection; blue, production; green, diffusion; purple, dissipation; gray, pressure diffusion from Balajee and Panchapakesan [100]; pink, arithmetic average of pressure diffusion and turbulent diffusion calculated from Balajee and Panchapakesan [100]......	56
Figure 3.15 : Normalized dissipation rate along the jet centerline.....	57
Figure 3.16 : Kolmogorov length scale results along the jet centerline.	58
Figure 3.17 : Jet half-width results along the jet centerline for Re=20000.	59

Figure 3.18	: Centerline velocity results along the jet centerline for $Re=20000$	60
Figure 3.19	: Velocity profiles along jet axis for $Re=20000$	61
Figure 3.20	: Sensitivities of closure coefficients and TKE contour of default and calibrated coefficient results.....	63
Figure 3.21	: Contours of default and calibrated coefficient results (a) TKE and (b) TI.	66
Figure 3.22	: Mean velocity contours of default and calibrated coefficients.....	67
Figure 4.1	: (a) V2 cascade geometry and locations of measurement planes (b) computational domain with boundary conditions.	77
Figure 4.2	: (a) Grid structure of V2 cascade defined as medium (b) low y^+ medium grid trailing edge detail (c) high y^+ medium grid trailing edge detail.	82
Figure 4.3	: C_p distribution on V2 cascade surfaces for low y^+ grid (a) Exp. 1 results (b) Exp. 2 results (c) Exp. 3 results.....	87
Figure 4.4	: Wake profiles for low y^+ grid (a) Exp. 1 results (b) Exp. 2 results (c) Exp. 3 results.	89
Figure 4.5	: C_p distribution on V2 cascade surfaces for high y^+ grid (a) Exp. 1 results (b) Exp. 2 results (c) Exp. 3 results.....	92
Figure 4.6	: Wake profiles for high y^+ grid (a) Exp. 1 results (b) Exp. 2 results (c) Exp. 3 results.	93
Figure 4.7	: RMS of governing equations for Exp. 3 with RNG F_c model (a) low y^+ grid results (b) high y^+ grid results.....	95
Figure 4.8	: f_η contours of Exp. 3 test case (a) RNG with high y^+ grid (b) RNG F_c with high y^+ grid (c) RNG with low y^+ grid (d) RNG F_c with low y^+ grid.....	96
Figure 4.9	: f_η distribution on cascade surfaces for Exp. 3 test condition.....	97
Figure 4.10	: Design condition (Exp. 4) results of (a) low y^+ grid C_p distribution (b) high y^+ grid C_p distributions.	99
Figure 4.11	: Wake profiles of Exp. 4 (a) low y^+ grid results (b) high y^+ grid results.	100
Figure 4.12	: Isentropic Mach number distributions on V103 cascade (a) inlet $Re=450000$ result (b) inlet $Re=800000$ result.....	102
Figure 4.13	: Total pressure loss coefficient results of V103 cascade for different operating conditions.	103
Figure 4.14	: Mach number distribution on midplane for V103 cascade at design condition (a) SST result (b) RNG result (c) RNG F_c result (d) LES result.	104
Figure 5.1	: Computational grids used in this study for (a) CBFS and (b) periodic hills.....	119
Figure 5.2	: Computational grids used for (a) 90° step angle BFS and for (b) 20° step angle BFS.....	120
Figure 5.3	: Comparison of calculated (a) C_p and (b) C_f distributions along the wall with LES.....	126
Figure 5.4	: TKE budget terms at the inflow boundary for boundary condition assessment cases (a) convection term (b) production term (c) destruction term (d) total diffusion term results comparison.	128

Figure 5.5	: TKE budget terms at the onset of the ramp for boundary condition assessment cases (a) convection term (b) production term (c) destruction term (d) total diffusion term results comparison.	130
Figure 5.6	: RNG-SST model results comparison with SST and LES for (a) C_p and (b) C_f distributions along the wall.	133
Figure 5.7	: RNG-SST model results comparison with SST and LES for (a) dimensionless mean velocity profiles and (b) dimensionless k profiles calculated in the flow field in CBFS case.	134
Figure 5.8	: TKE budget comparisons at (a) $x/H_R = 1.5$, (b) $x/H_R = 2$, (c) $x/H_R = 3$, (d) $x/H_R = 4$. Symbols: $\bullet$, LES results; — (solid lines), present numerical study with SST model; - - - (dashed lines), present numerical study with RNG-SST model. Colors: orange, convection; blue, production; green, total diffusion; purple, dissipation.	135
Figure 5.9	: $F_{Reattach}$ distribution computed in CBFS case.	136
Figure 5.10	: $-f_\eta$ distribution computed in CBFS case.	137
Figure 5.11	: Mean (top) and fluctuating (bottom) velocity profile comparisons of BFS-a case.	139
Figure 5.12	: Mean (top) and fluctuating (bottom) velocity profile comparisons of BFS-b case.	140
Figure 5.13	: Mean (top) and fluctuating (bottom) velocity profile comparisons of BFS-c case.	141
Figure 5.14	: Mean (top) and fluctuating (bottom) velocity profile comparisons of BFS-d case.	142
Figure 5.15	: Streamwise velocity profile comparison of Periodic Hills-a case (top) and Periodic Hills-b case (bottom).	144
Figure 5.16	: C_f distribution on the hills' bottom wall at $Re=10595$	146
Figure 5.17	: Relative error of predicted separation position for each computational grid.	147
Figure 5.18	: Computational grid used in the numerical investigations of CBFS experiment.	148
Figure 5.19	: C_p distribution along the wall.	149
Figure 5.20	: Mean velocity distribution in the flow field.	149
Figure 5.21	: Dimensionless momentum thickness (top) and displacement thickness (bottom) results.	151
Figure 5.22	: Shape factor results calculated at measurement locations.	152



CALIBRATION AND OPTIMIZATION OF RANS TURBULENCE MODELS FOR VARIOUS CASES

SUMMARY

Reynolds-Averaged Navier-Stokes turbulence models dominate both engineering applications and academic research due to their low computational cost and ability to deliver accurate predictions across a wide range of turbulent flows. Although high-fidelity approaches such as large-eddy simulations and direct numerical simulations can resolve a wide range of turbulent scales, their computational cost makes them impractical for most real-world problem investigations. Even though they offer intriguing alternatives, hybrid approaches like wall-modeled large-eddy simulations and Reynolds-Averaged Navier-Stokes/Large Eddy Simulation coupling still depend on turbulence modeling. Improving engineering design procedures' prediction performance in turbulent flows remains pivotal aspect today because they naturally advance from low-fidelity to high-fidelity methodologies within allowable computational resources.

From this perspective, the modeling capabilities of widely-used Reynolds-Averaged Navier-Stokes turbulence models were enhanced for canonical problems in engineering applications and academic research. These improvements were achieved through optimization of their closure coefficients, replacement with defined functions, or coupling different turbulence models. The studies conducted within the scope of thesis have been disseminated to the international literature through one conference paper and three articles.

In the first part of the study, round free jet was modeled with standard $k - \varepsilon$ turbulence model and the results were compared with experimental measurements. The aim of the first study was to improve the prediction capability of standard $k - \varepsilon$ turbulence model, which was known as unsuccessful in modeling flows involving anisotropic Reynolds stresses such as turbulent jets. It was aimed to investigate the effect of model constants such as $C_{\varepsilon 1}$, the coefficient of turbulent production; $C_{\varepsilon 2}$, the coefficient of turbulent kinetic energy dissipation and C_{μ} , used to calculate turbulence viscosity. Steady-state and three-dimensional computational fluid dynamics analyses were performed by systematically changing these constants and the results were evaluated based on dimensionless velocity and turbulence profiles, jet half-width, centerline velocity and turbulence intensity along the jet centerline. It was observed that the adjusted new constants achieve more successful results than the default coefficients. By calibrating the model coefficients that alters from the default model coefficients by 16% for $C_{\varepsilon 1}$, 17% for $C_{\varepsilon 2}$, and 1.2% for C_{μ} , it is revealed that jet centerline velocity distribution and jet half-width distribution along the jet center axis can be estimated better. The new closure coefficients of the standard $k - \varepsilon$ were also validated for different Reynolds numbers. The numerical simulations revealed that the velocity profile change with

the calibrated model coefficients and the numerical results aligned well with the experimental observations. By this way, the computation errors due to uncertainty of turbulence modeling encountered in numerical investigations were reduced.

In the second part, the investigation of turbulent jet flows was extended. Since Reynolds Stress Models directly solve the Reynolds stress equations rather than relying on modeling approximations, they were anticipated to provide more accurate numerical results for flows characterized by strong anisotropy. Nevertheless, it was observed that the calculated jet half-width, velocity decay, and spreading rate differed from experimental results due to uncertainties inherent in the turbulence model. Consequently, closure coefficients of the Reynolds Stress Model were calibrated using a variant of the Multi-Objective Genetic Algorithm based on jet half-width data obtained experimentally in the near-field region of the jet. With the use of appropriate discretization scheme and computational grid, adjusted coefficient combination for the turbulence model showed improved accuracy in predicting jet half-width at Reynolds numbers of 10000 and 20000, reducing the errors of calculated decay constant and spreading rate approximately from 2% to 1% and from 16% to 5%, respectively. A detailed examination of the turbulence budget along the longitudinal axis in the self-similar region revealed that the new model coefficients enhanced the modeling of diffusion term but compromised the advection term. As a result of the altered advection term, increased error margins were observed in turbulence intensity and velocity distribution along the jet centerline, although dissipation along the axis was improved. Hence, the modeling error in jet half-width calculations using the numerical method was decreased, enhancing performance of the Reynolds Stress Model compared to default coefficients.

For the third part of the study, a modified Renormalization Group $k - \varepsilon$ turbulence model for axial compressor cascade flows was introduced, developed to improve the prediction of total pressure loss coefficient and wake profiles at various operating conditions. Built on the theoretical foundation of Renormalization Group theory of turbulence, the model incorporates a locally adaptive η_0 coefficient, enabling it to better capture flow physics across varying near-wall resolutions and wall treatment approaches. The model was named as RNG F_c and validated using experimental data of The Advisory Group for Aerospace Research and Development Working Group 18, focusing on the V2 and V103 compressor cascades under on design and off-design conditions. Comparative analyses versus original renormalization group $k - \varepsilon$ and Shear Stress Transport $k - \omega$ models demonstrate that the presented model achieves superior accuracy in total pressure loss calculations and wake region predictions at various inlet Mach numbers. The model's performance remains consistent across different computational grids, including low y^+ and high y^+ structures, and is compatible with both scalable wall functions and enhanced wall treatments. Notably, the modified model exhibits computational cost efficiency and numerical stability, making it a promising tool for engineering applications. This work highlighted the potential of the RNG F_c model as a robust and cost-effective alternative for turbulence modeling in axial compressor cascade investigations, offering significant improvements over traditional Reynolds Averaged Navier-Stokes models.

To overcome another recognized limitation of the Reynolds-Averaged Navier-Stokes framework, particularly in prediction of reattachment mechanisms for flows separating from continuous surfaces, the final part of this study aimed to overcome this deficiency

with the widely-used Shear Stress Transport $k - \omega$ turbulence model. Although the Shear Stress Transport model performs well for a wide range of flows, its predictive capability deteriorates in wake regions dominated by complex turbulent structures. To enhance its accuracy, a modified model incorporating the Renormalization Group theory of turbulence is proposed. The model correction is introduced as an additional source term into the dissipation equation and limited to the reattachment region through a blending function to ensure computational efficiency and numerical stability. The model is validated using several canonical problems, including the curved backward-facing step, vertical and inclined backward-facing steps, and periodic hills case at various Reynolds numbers. Effects of the inflow boundary condition are carefully examined to isolate the intrinsic performance of the modified model. Comparisons with benchmark Large Eddy Simulation studies and experimental data showed that the RNG-SST model substantially improves the prediction of mean velocity, turbulence kinetic energy, and wall quantities such as pressure coefficient, wall shear stress, and boundary layer parameters. In particular, prediction of the reattachment position is improved, yielding reduced relative errors by nearly half of the default Shear Stress Transport model. These findings demonstrate that integrating Renormalization Group theory provides a computationally cost-effective, numerically robust, and physically consistent improvement to the default Shear Stress Transport turbulence model for separated flows from a continuous surface.

Collectively, the thesis demonstrates that targeted calibration, physics-informed coefficient adjustment, and selective coupling can meaningfully reduce turbulence model induced uncertainty in Reynolds Averaged Navier-Stokes closures across jets, compressor cascades, and separated flows. The proposed models retain the low computational cost and robustness required in engineering while delivering accuracy gains typically associated with more complex approaches, thereby advancing the reliability of turbulence modeling within practical numerical investigations.



FARKLI PROBLEMLER İÇİN RANS TÜRBÜLANS MODELLERİNİN KALİBRASYONU VE OPTİMİZASYONU

ÖZET

Reynolds Ortalamalı Navier-Stokes türbülans modelleri, düşük hesaplama maliyetleri ve geniş bir yelpazedeki türbülanslı akış problemlerinde tatmin edici tahmin doğruluğu sunmaları nedeniyle hem mühendislik uygulamalarında hem de akademik araştırmalarda en sık tercih edilen modelleme yöntemi konumundadır. Büyük Girdap Benzeşimi ve Doğrudan Sayısal Benzeşim yöntemi gibi yüksek doğruluklu yaklaşımlar türbülansın geniş bir ölçek aralığını çözümler; ancak hesaplama maliyetleri, gerçek dünya problemlerinin büyük çoğunluğunun incelenmesinde pek pratik değildir. Buna karşılık, duvar modeli kullanılan Büyük Girdap Benzeşimi ve Reynolds Ortalamalı Navier-Stokes/Büyük Girdap Benzeşimi melezlemeleri gibi hibrit yöntemler cazip alternatifler sunsa da özünde hâlâ türbülans modellemesine bağımlıdır. Bu çerçevede, düşük güvenilirlikten yüksek güvenilirliğe ilerleyen mühendislik tasarım süreçlerinin türbülanslı akışlardaki hesaplama performansını, hesaplama kaynakları elverdiğince arttırmak güncelliğini koruyan önemli bir gereksinimdir.

Bu tezde, yaygın kullanılan Reynolds Ortalamalı Navier-Stokes türbülans modellerinin modelleme kabiliyetleri, mühendislik uygulamaları ve akademik araştırmalarda sık incelenen temel problemler özelinde arttırılmıştır. İyileştirmeler, kapama katsayılarının optimizasyonu, model sabitlerinin fizik temelli tanımlanan fonksiyonlarla ikame edilmesi ve farklı türbülans modellerinin kontrollü biçimde birleştirilmesi yoluyla gerçekleştirilmiştir. Tez kapsamındaki çalışmalar bir konferans bildirisi ve üç makale ile uluslararası literatüre sunulmuştur.

Tezin ilk bölümünde, dairesel serbest jet akışı standart $k - \varepsilon$ türbülans modeli ile çözümlenmiş ve sonuçlar deneysel ölçümlerle karşılaştırılmıştır. Anizotropik Reynolds gerilmelerinin hâkim olduğu türbülanslı jetler gibi akışlarda başarımının sınırlı olduğu bilinen standart $k - \varepsilon$ modelinin öngörü yeteneğinin artırılması amaçlanmıştır. Bu amaçla, türbülans üretimine etki eden $C_{\varepsilon 1}$, türbülans kinetik enerjisinin yitimine etki eden $C_{\varepsilon 2}$ ve türbülanslı viskozitenin hesaplanmasında kullanılan C_{μ} sabitlerinin etkisi sistematik olarak incelenmiştir. Üç boyutlu, daimi rejimde hesaplamalı akışkanlar dinamiği analizleri koşturulmuş; sabitler adım adım değiştirilerek boyutsuz hız ve türbülans şiddeti profilleri, jet yarı kalınlığı, eksenel merkez hattı hızı ve merkez hattı boyunca türbülans şiddeti dağılımı üzerinden değerlendirme yapılmıştır. Ayarlanmış yeni katsayıların, asıl katsayılarla kıyasla daha başarılı sonuçlar verdiği görülmüştür. $C_{\varepsilon 1}$ için %16, $C_{\varepsilon 2}$ için %17 ve C_{μ} için %1.2 değişim içeren kalibre edilmiş katsayılar ile jet merkez hattı hızının azalma dağılımı ve merkez hattı boyunca jet yarı kalınlığının dağılımı daha iyi tahmin edilmiştir. Standart $k - \varepsilon$ modelinin yeni katsayıları farklı Reynolds sayılarında da sınanmış; kalibre katsayılarla elde edilen hız profillerinin deneysel gözlemlerle daha fazla uyum gösterdiği ortaya konmuştur. Böylece, sayısal incelemelerde türbülans modellemesinden kaynaklanan belirsizliklerin tetiklediği hesaplama hataları azaltılmıştır.

İkinci bölümde, türbülanslı jet akışlarına ilişkin incelemeler genişletilerek Reynolds Gerilmesi Modelleri ele alınmıştır. Reynolds Gerilmesi Modelleri, yaklaşık modellere göre Reynolds gerilmeleri için taşıyım denklemlerini doğrudan çözdükleri için, şiddetli anizotropiye sahip akışlarda daha yüksek doğruluk potansiyeli taşımaktadır. Bununla birlikte, türbülans modelinin belirsizliğinden ötürü hesaplanan jet yarı kalınlığı, merkez hattı hızındaki azalma ve jet yayılım oranı, deneysel ölçümlerden kayda değer ölçüde sapmaktadır. Bu nedenle model kapama katsayıları, jet çıkışına yakın bölgedeki deneysel olarak elde edilen yarı kalınlık verileri esas alınarak çok hedefli genetik algoritmanın bir varyantı ile kalibre edilmiştir. Uygun ayırıklaştırma şeması ve çözüm ağı ile birlikte ayarlanmış katsayı kombinasyonu, Reynolds sayıları 10000 ve 20000 için jet yarı kalınlığı hesaplamalarını iyileştirmiş; merkez hızın azalma sabiti ve jet yayılma oranı hatalarını sırasıyla yaklaşık %2'den %1'e ve %16'dan %5'e düşürmüştür. Jetin eşbenzer bölgesinde boyuna eksen boyunca gerçekleştirilen türbülans kinetik enerji bütçesi hesaplamaları, yeni katsayıların difüzyon teriminin modellenmesini güçlendirdiğini; buna karşılık adveksiyon teriminin etkisini zayıflattığını ortaya koymuştur. Bu değişimin sonucu olarak eksen boyunca türbülans yitim davranışı iyileşse de merkez hattı hız dağılımı ve türbülans şiddetinde hata payları artmıştır. Bununla birlikte, sayısal yöntemin jet yarı kalınlığı tahminlerindeki modelleme hatası anlamlı biçimde azalmış ve Reynolds Gerilmesi Modeli'nin asıl katsayılarıyla elde edilen sonuçlara kıyasla genel performans artışı sağlanmıştır.

Üçüncü bölümde, eksenel kompresör kanat dizisi problemleri için Yeniden Normalleştirme Grubu $k - \varepsilon$ türbülans modelinin bir türevi geliştirilmiştir. Amaç, farklı çalışma koşullarında toplam basınç kaybı katsayısı ve ard iz bölgesi profillerinin tahminini iyileştirmektir. Yeniden Normalleştirme Grubu kuramsal temeli üzerine inşa edilen yaklaşım, yerel olarak ayarlanabilir η_0 katsayısı içermektedir; böylece duvara yakın bölgede farklı çözüm ağı yapılarında ve farklı duvar modellerinde akış fiziğinin daha tutarlı yakalanması sağlanmaktadır. Model RNG F_c olarak adlandırılmış ve Advisory Group for Aerospace Research and Development Working Group 18 veri tabanı kullanılarak, V2 ve V103 kanat dizileri üzerinde hem tasarım koşullarında hem tasarım dışı koşullarda doğrulanmıştır. Karşılaştırmalar, özgün Yeniden Normalleştirme Grubu $k - \varepsilon$ ve Shear Stress Transport $k - \omega$ modellerine kıyasla, önerilen yaklaşımın çeşitli giriş Mach sayılarında toplam basınç kaybı ve ard iz bölgesi öngörülerinde daha yüksek doğruluk sunduğunu göstermektedir. Model, düşük y^+ ve yüksek y^+ değerlerine sahip farklı çözüm ağları dâhil olmak üzere farklı sonlu hacim çözücülerinde tutarlı performans sergilemekte; ölçeklenebilir duvar fonksiyonları ve geliştirilmiş duvar modellemeleri ile uyumlu çalışmaktadır. Sayısal kararlılık ve hesaplama maliyeti verimliliği korunmuş; bu nedenle yöntem mühendislik uygulamaları için pratik bir araç olarak öne çıkmıştır. Böylelikle RNG F_c yaklaşımı, eksenel kompresör kanat dizisi incelemelerinde geleneksel Reynolds Ortalamalı Navier-Stokes modellerine nazaran sağlam ve maliyet etkin bir seçenek olarak belirgin kazanımlar sunmaktadır.

Son bölümde, özellikle kesintisiz yüzeylerden ayrılıp yeniden birleşen akışlarda yeniden birleşme mekanizmalarının öngörülmesindeki bilinen bir sınırlılığın aşılması amaçlanmıştır. Bu amaçla, yaygın kullanılan Shear Stress Transport $k - \omega$ modeli, Yeniden Normalleştirme Grubu türbülans modelinden türetilen bir düzeltme ile geliştirilmiştir. Düzeltme, türbülans kinetik enerji yitim denklemi içine ek bir kaynak terimi olarak uygulanmış ve hesaplama verimliliği ile sayısal kararlılığın

korunması için bir belirleyici fonksiyon aracılığıyla yeniden birleşme bölgesi ile sınırlandırılmıştır. Model; ters basamak rampası, düşey ve eğik ters basamaklar ve farklı Reynolds sayılarında sıralı tepelikler gibi temel akış ayrılması problemleri üzerinde doğrulanmıştır. Giriş yüzeyindeki sınır şartının etkileri, modelin öz davranışını görebilmek amacıyla dikkatle sınanmıştır. Büyük Girdap Benzeşimi kıstas çalışmaları ve deneysel verilerle yapılan karşılaştırmalar, RNG-SST yaklaşımının ortalama hız, türbülans kinetik enerjisi ve basınç katsayısı, kayma gerilmesi ve sınır tabaka parametreleri gibi duvar büyüklüklerinin hesaplamalarını anlamlı derecede iyileştirdiğini göstermiştir. Özellikle yeniden birleşme konumu tahmini, asıl Shear Stress Transport modeline kıyasla yaklaşık yarı yarıya daha düşük bağıl hata ile elde edilmiştir. Bu bulgular, yeniden normalleştirme grubu kuramının bütünlenmesinin, kesintisiz bir yüzeyden ayrılıp yeniden birleşen akışların incelenmesinde, asıl Shear Stress Transport türbülans modeline kıyasla hesaplama maliyeti bakımından elverişli, sayısal kararlı ve fizik ile tutarlı bir gelişim sağladığını göstermektedir.

Genel olarak, bu tezde sunulan sonuçlar; hedefli kalibrasyon, fizik temelli katsayı ikameleri ve özel model geliştirmelerinin, jetler, kompresör kanat dizileri ve ayrılmalı akışlarda Reynolds Ortalamalı Navier-Stokes türbülans modellerinden kaynaklı belirsizliği anlamlı bir düzeyde azaltabildiğini ortaya koymaktadır. Önerilen yaklaşımlar, mühendislik uygulamalarının gerektirdiği düşük hesaplama maliyeti ve tutarlılığı korurken, genellikle daha karmaşık yöntemlerle ilişkilendirilen doğruluk kazanımlarını erişilebilir kılmakta; böylece pratik sayısal incelemelerde türbülans modellemesinin güvenilirliğini ileri taşımaktadır.



1. INTRODUCTION

For almost all living beings, fluids are indispensable substances. Their ubiquitous presence in every aspect of life has rendered their properties, behavior, and effects a perpetual source of mystery and fascination for humanity throughout history. This curiosity led to the emergence of the field of fluid mechanics. It would not be an exaggeration to state that nearly all actions performed by living beings, from microscopic organisms to giants such as the blue whale or the giant sequoia to sustain their existence, involve fluid mechanics, which is divided into the sub-fields of fluid dynamics, when forces act on a fluid giving it velocity, and fluid statics, when a fluid at rest assumes the shape of its container. Breathing, the movement of blood through vessels, the transport of water and minerals from roots to leaves in plants, the transport of sound waves, the flight of an aircraft, or a rocket's journey into space, all constitute fluid mechanics problems.

Throughout history, this significant topic has attracted the interest of and garnered contributions from invaluable figures such as the Greek mathematician Archimedes, who discovered the buoyancy of water, and the artist, scientist, and engineer Leonardo da Vinci, who produced one of the earliest and most masterfully depicted paintings of wave motion. Today, we are able to understand fluid mechanics problems more deeply and calculate fluid motion under various assumptions, thanks to the foundational work of Sir Isaac Newton [1].

The investigation of physical phenomena consists of two principal steps. In the first step, all variables influencing the phenomenon are identified, reasonable assumptions are made, relevant physical laws and principles are applied, and the problem is formulated mathematically. In the second step, the problem is solved using an appropriate approach, and the results are interpreted. In the mathematical modeling of fluid behavior, building on the pioneering studies of Newton, Daniel Bernoulli [2] and Leonhard Euler [3] established the mathematical foundations of fluid mechanics for inviscid flow, leading to the formulation of the equation bearing Bernoulli's name. Based on these foundations, C. Louis Navier [4] incorporated fluid friction to develop

the momentum transfer equations, with their modern form being finalized by G. Gabriel Stokes [5] through the definition of molecular viscosity. Subsequently, with the inclusion of the conservation of mass, the momentum transfer equation, known as the Navier-Stokes (N-S) equations, was established. The N-S equations are widely regarded as the most important equations in fluid mechanics [6, 7]. This achievement enabled the physical principles of fluid mechanics to be expressed mathematically.

Many processes in nature, which may appear random and devoid of order, are in fact governed by visible or hidden physical laws. Whether we are aware of them or not, these laws exist and manage seemingly ordinary events in a consistent and predictable manner [7]. Most of these laws are well-defined and thoroughly understood by scientists. However, due to its chaotic nature, fluid mechanics involves a physical phenomenon whose laws are still not fully defined today; it exhibits time-dependent irregular variations, always possesses three-dimensional effects, and is diffusive and dissipative: turbulence. Unfortunately, even today, we cannot fully explain the mechanics of turbulence. In 1877, Joseph V. Boussinesq [8] proposed a highly significant solution concept by suggesting that in open channel flows, turbulence has a diffusive effect on the mean flow, which could be represented by eddy viscosity. This remarkable engineering approach laid the groundwork for the development of models, referred to as the Eddy Viscosity Models (EVM), that enable the calculation of the Reynolds stress terms contained within the Reynolds-Averaged Navier-Stokes (RANS) equations and the Reynolds decomposition, which expresses variables such as velocity and pressure in the N-S equations as time-dependent and mean components, introduced by Osborne Reynolds in 1895 [9].

All these studies, along with contributions from scientists such as Leibniz, d'Alembert, Lagrange, Laplace, Gauss, Cauchy, Fourier, St. Venant, and many other known and unknown mathematical and physical geniuses, made it possible to mathematically model the physical laws of fluid mechanics. One of the most important theories introduced in fluid mechanics is the boundary-layer theory, presented by Ludwig Prandtl in 1904 [10]. Prandtl showed that fluid friction has significant effects near a wall while being negligible in the mean flow, and the boundary-layer allows for the distinction between these two regions. Other notable scientists, such as Theodore von Kármán, Paul Blasius, and Johann Nikuradse, later advanced this theory, developing solution approaches with various assumptions. Prandtl, who made

significant contributions to fluid mechanics, is also the creator of the mixing-length theory, recognized as the first known turbulence model [11, 12].

Although the RANS equations presented by Reynolds [9] provided an extraordinary contribution to fluid mechanics, paving the way for the refinement of the mathematical model, the inclusion of Reynolds stress terms made solving the equations impossible for many everyday problems. The inability to solve the non-linear RANS equations directly led to the birth of a numerical solution approach, represented by the pioneering work of Richardson [13] in 1922, known today as Computational Fluid Dynamics (CFD). Furthermore, his statement that "we realize that: big whirls have little whirls that feed on their velocity, and little whirls have lesser whirls and so on to viscosity—in the molecular sense," laid the foundation for the energy cascade theorem presented by Kolmogorov [14] in 1941. According to Kolmogorov's hypothesis, random fluctuations in turbulence produce a broad spectrum of energy scales through dissipation processes driven by turbulent eddies [14]. While no universally exact definition of a turbulent eddy exists, Kolmogorov asserts that large-scale turbulent eddies extract energy from the mean flow, transferring it progressively to smaller structures through the cascade process until it is dissipated by viscosity at the smallest scales [15]. This cascade process defines the core mechanism of turbulence.

CFD enables the prediction of a fluid mechanics problem's behavior in a computer environment or the mathematical examination of various aspects of a phenomenon without costly and time spending experiments. By employing a suitable and realistic mathematical model, highly accurate results can be obtained with relatively little effort. In fact, creating very accurate but complex models is not very difficult. However, if such models entail high computational cost and are time consuming to solve, they lose their usability in practice. The preparation of these models requires sufficient knowledge about the relevant natural phenomena and their governing laws, as well as sound reasoning. An unrealistic model will inevitably yield incorrect and therefore unacceptable results. It should not be forgotten that the results obtained from an analysis are, at best, only as accurate as the assumptions made while simplifying the problem [7]. The renowned statistician George Box succinctly summarized this situation by stating, "all models are wrong, but some are useful" [16].

According to Wilcox [17], CFD has three fundamental elements. The first is grid

generation, which involves converting the relevant problem into a form usable by the computer. Thus, the grid represents the real-world problem as a series of discrete points, allowing for the computation of the parameter of interest at each node. The second is the numerical algorithm used to transform the equations into a form suitable for iterative solution on the computational grid and to solve them. The third, and the central theme of this thesis, is turbulence modeling.

With the development of solution methods and the increase in computational power due to High-Performance Computing (HPC), the first computer-aided, time-dependent N-S computations were achieved by Harlow and his team at Los Alamos, although their work became part of the open literature in the 1960s. In 1963, Smagorinsky introduced a new and significant solution approach for flow field prediction using RANS, known today as Large-Eddy Simulation (LES), which directly computes large- and intermediate-scales of turbulent eddies and models the smaller (or dissipative) scales [18]. In 1970, Orszag [19] presented the theory of Direct Numerical Simulation (DNS), which resolves all scales of eddies directly without the need for turbulence modeling. However, the computational cost of methods like DNS was prohibitively high for many everyday problems, considering the computational power of the era. Even the closure model for the Reynolds stresses, first introduced by Chou [20] in 1945 and developed by Rotta [21] in 1951, required solving six different equations and remained computationally expensive for the time. Consequently, a need arose to model the Reynolds stresses in the RANS equations for turbulent flows using closure equations that could deliver consistent results through reasonable assumptions, thereby reducing computational cost, rather than solving them directly. Although various modeling approaches were developed during that period, alongside Prandtl's mixing-length theory, Harlow and Nakayama [22, 23] laid the foundations for the turbulence models widely used today by introducing the turbulence transport equations and the transport of turbulence energy decay equations, based on the acceptance of the existence of turbulent kinetic energy.

In 1972, Patankar and Spalding [24] introduced the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) algorithm, used today for solving many engineering problems, which established a relation between pressure and velocity and enabled an iterative solution for the RANS equations through the prediction-correction method. Following this significant development, Jones and Launder [25] presented in 1972

the standard $k - \varepsilon$ turbulence model, the first example of what are now known as two-equation turbulence models. Over the past fifty years, enormous effort has been dedicated to improving the numerical accuracy of CFD simulations for predicting complex turbulent flows. Almost all of these models contain coefficients determined through experimental, analytical, or numerical methods. These coefficients, also known as closure coefficients, belong to the additional closure equations that must be solved to calculate the Reynolds stress term in the RANS equations, and different coefficients are used in different models. As emphasized by many researchers, these coefficients introduce uncertainty into turbulence models. Numerous studies have demonstrated that calibrating model coefficients for specific problems can reduce computational errors. While high-fidelity approaches such as LES and DNS can resolve a wide range of turbulent scales, their prohibitive computational cost renders them impractical for most industrial and complex applications. Hybrid methods such as RANS-LES coupling and wall-modeled LES provide promising alternatives but still rely heavily on turbulence modeling. Since engineering design processes naturally progress from low- to high-fidelity methods as computational resources allow, Reynolds-Averaged Navier-Stokes turbulence models continue to dominate both industry and academics due to their low computational cost and ability to deliver consistent results across a wide range of turbulent flow problems. Thus, reducing uncertainties and error margins in turbulent RANS predictions remains as important today as it has ever been.

This thesis aims to enhance confidence in CFD for generic engineering problems by reducing turbulence model uncertainty, increasing computational success through the case-specific calibration of RANS turbulence model coefficients. To achieve this, the objective is to improve these coefficients using reliable experimental data or high-fidelity numerical results, such as LES or DNS for the corresponding case, through a suitable optimization method, to minimize computational errors. Although computational errors can be significantly reduced by optimizing coefficients for a specific problem, using the same coefficients under different conditions (e.g., different Reynolds numbers, different boundary conditions) on the same problem does not always yield better results. This situation has led some researchers to replace the optimized constant coefficients with a function or a variable, thereby eliminating the need for repeated optimization. Due to the complex nature of turbulence, many

researchers hesitate to work on this subject. The results and suggestions presented in this thesis are straightforward and intended to encourage further research in this area. As evidenced by the literature surveys presented in each related chapter, the calibration of turbulence closure coefficients is a current and active topic in this research field. All existing model coefficient calibration studies are international. The works carried out within the scope of this thesis have not been previously undertaken in the national literature.

From an economic perspective, using the CFD analysis method is a key factor in the product development process, facilitated by numerical simulations that avoid the cost of prototype production. Therefore, this study aims to improve turbulence models, a pivotal factor of CFD, and was conducted with the goal of increasing the reliability. During this process, one conference paper was presented at an international congress, and three peer-reviewed international journal papers have been published.

The challenges encountered during the thesis work transformed the study from a model calibration effort into a model improvement endeavor after a certain point. In Chapter 2 and Chapter 3, which investigate round free jet problems, the coefficients of two different RANS turbulence models were calibrated using a genetic algorithm to ensure results consistent with those obtained experimentally. Turbulent round jet problems, exhibiting self-similarity at Reynolds numbers above 10000, allowed the coefficients obtained through optimization to consistently reduce the error rate compared to the default coefficients, even with varying Reynolds numbers. However, in the axial compressor cascade flow problems examined in Chapter 4, the emergence of different optimal coefficient combinations with changes in Reynolds number, Mach number, and cascade profile revealed the necessity of adding a function to the coefficients of the studied RANS turbulence model, leading to the development of a new variant of the Renormalization Group (RNG) theory of turbulence, presented as the RNG F_c model. Subsequently, optimization studies conducted on problems such as the curved backward-facing ramp, backward-facing step, and periodic hills in Chapter 5 failed to satisfactorily reduce the error rate of the studied RANS turbulence model, regardless of the number of optimization runs, which prompted the coupling of two different turbulence models, resulting in the emergence of the RNG-SST turbulence model.

The calibrated turbulence model coefficients and the performance of the turbulence

models under various boundary conditions and flow regimes presented in this work have been examined comprehensively. According to Wilcox's statement on an ideal turbulence model, it should introduce the minimum amount of complexity while capturing the essence of the relevant physics [17]. Guided by this principle, model improvements have been realized that are successful, computationally cost-effective, numerically stable, and provide robust and reliable calculation capability even under varying conditions for engineering applications. Within the scope of this thesis, the source codes enabling the integration of the RNG F_c model, presented as a turbulence model variant to the international literature, into various commercial software have been shared as open source on GitHub [26], encouraging researchers to evaluate the model variant in different cases and various flow regimes. The newly developed RNG-SST model has been shared with a healthy amount of mathematics, facilitating and encouraging researcher contributions to the model development process.

To summarize, all model improvements and developed models presented within the scope of this thesis have been examined in detail under various conditions. Their performance on problems commonly studied in engineering and academic contexts has been comprehensively evaluated, their effects on numerical stability have been assessed, and their advantages and disadvantages have been thoroughly presented. In this context, calibration and improvement studies on RANS turbulence models have been conducted across all investigations, enhancing their computational performance for various cases, in alignment with the thesis title.



2. IMPROVEMENT OF STANDARD K-EPSILON TURBULENCE MODEL FOR ROUND FREE JETS BY ADJUSTING CLOSURE COEFFICIENTS¹

2.1 Introduction

Round free jet flows are common in many engineering and industrial applications, such as gas turbine combustors, chemical reactors, and waste incinerators. In these systems, a fluid is ejected from a nozzle and expands freely into a surrounding medium. The dynamics of these flows are highly complex and strongly influenced by the Reynolds number, nozzle geometry, and flow conditions. Therefore, accurate prediction and understanding of the characteristics of round free jets is essential for the design and optimization of these systems. Over the past few decades, significant progress has been made in understanding the physics of round free jet flows through experimental, numerical, and theoretical studies. Notable examples include the pioneering work of Wygnanski and Fiedler [27] on the structure of turbulent round jets, the experimental and numerical studies conducted by Bradshaw et al. [28, 29] and further studies on turbulence of round jets, and the development of turbulence models for round free jets by various researchers [30, 31].

The $k - \varepsilon$ turbulence model is a widely used RANS model for simulating turbulent flows in various engineering applications. The model is based on the Reynolds-averaged Navier-Stokes equations and developed in the 1970s [32]. The $k - \varepsilon$ model is a two-equation turbulence model that solves the turbulence kinetic energy (k) and its dissipation (ε). These variables are related to the turbulent eddy viscosity, which governs the momentum and heat transfer in the flow. The $k - \varepsilon$ model assumes that the turbulence in a flow field is isotropic and homogeneous, which means that the turbulence statistics are the same in all directions and at all locations within the flow. The model also assumes that turbulence is steady and that the eddy viscosity

¹This chapter is based on the paper: Turutoglu, C. and Cadirci, S. (2023). Improvement of Standard K-Epsilon Turbulence Model for Round Free Jets by Adjusting Closure Coefficients, *In Proceedings of the ASME 2023 International Mechanical Engineering Congress and Exposition. Volume 9: Fluids Engineering*, October 29-November 2, 2023 New Orleans, Louisiana, USA.

is proportional to the turbulent kinetic energy and the length scale of the turbulent eddies. These assumptions make the $k - \varepsilon$ model relatively simple to implement and computationally efficient compared to more complex turbulence models.

Despite its simplicity, the $k - \varepsilon$ model has been shown to provide reasonable accuracy for a wide range of turbulent flows, including wall-bounded and free-shear flows [33–35]. However, the model's performance is dependent on several factors, including the choice of turbulence closure coefficients and the grid resolution of the computational domain. Therefore, it is important to validate the $k - \varepsilon$ model for specific flow conditions and to calibrate the model parameters accordingly. The closure coefficients are empirical constants used to relate the turbulent transport terms to the mean flow variables. These coefficients are usually determined by fitting the model predictions to experimental data or high-fidelity numerical simulations [36–42]. The accuracy of the model predictions depends critically on the values of these closure coefficients. The calibration process involves adjusting the closure coefficients until the model predictions match the experimental data or numerical simulations as close as possible. The accuracy of the model predictions depends critically on the values of these closure coefficients.

In recent years, there has been a growing interest in developing more accurate and reliable methods to calibrate the closure coefficients of the $k - \varepsilon$ model. Several approaches have been proposed for calibration, including Bayesian calibration, machine learning, and optimization techniques [43–46]. These methods aim to reduce the uncertainty in the closure coefficients and improve the prediction capabilities of the turbulence model. However, there is still a lack of consensus on the appropriate values of these coefficients for different types of flows. In particular, the closure coefficients have been found to vary significantly for different turbulence intensities, Reynolds numbers, and flow geometries [47–50].

The objective of the current study is to investigate the impact of the closure coefficients on the predictions of the standard $k - \varepsilon$ model for round free jet flows. In particular, genetic algorithm is used to calibrate the closure coefficients for various Re of round free jet flows by using the mean flow velocity profiles and turbulence intensities. The sensitivity of the turbulence model predictions to the closure coefficients is also examined and compared with experimental data at different Reynolds numbers.

2.2 Methodology

2.2.1 Problem description

Round free jets are commonly encountered in a variety of industrial and environmental applications, and to predict accurately their behavior is of great importance. However, turbulence models such as the $k - \epsilon$, which is widely used in CFD simulations, require calibration to accurately predict the flow behavior.

The jet flow in the experiments was obtained by directing compressed air through a pressure regulating instrument to the resting chamber, and then discharging it to the atmosphere through a nozzle with a 12 mm outlet diameter. The air outlet velocity can be adjusted by regulating the air pressure in the resting chamber, and the outlet velocities were set for the defined Re of 5000, 10000, and 20000 based on jet outlet diameter. A constant temperature hot-wire anemometer was used for velocity and turbulence measurements. A single wire miniature probe was selected as the measurement probe. The positioning of the probe at certain measurement points was provided by a 3-axis and computer-controlled motion mechanism. The accuracy of the motion mechanism that provides probe movement is 0.016 mm. The velocity and turbulence measurement were automatically performed with a computer package program.

In the geometry created for the CFD analyses, the jet outlet diameter D was determined as 10 mm. A cylindrical control volume was created considering the study of Wawrzak and Tyliczszak [51], and the distance from the jet outlet was determined as $50D$, while the control volume diameter was determined as $30D$. Figure 2.1 shows the computational domain for the CFD setup used in this study.

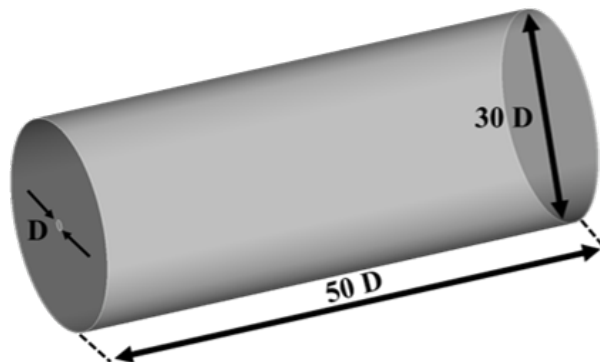


Figure 2.1 : Computational domain.

2.2.2 Computational grid

Computational grid of the created geometry was generated with Ansys®-Workbench Meshing software by using O-grid method. CAD geometry was divided into 9 parts and all parts were connected to each other to generate the grid. 24 different dense grids with number of elements ranging from 5000 to 10.5×10^6 were created to test the mesh independence and the analyses were performed for Re of 10000 using the standard (default) model coefficients recommended for the standard $k - \varepsilon$ turbulence model. The centerline velocities and jet half-widths obtained from the analyses were compared with the experimental results. Grid independence tests revealed that the most appropriate grid for this problem is the one with 800×10^3 elements. The grid structure is shown in Figure 2.2.

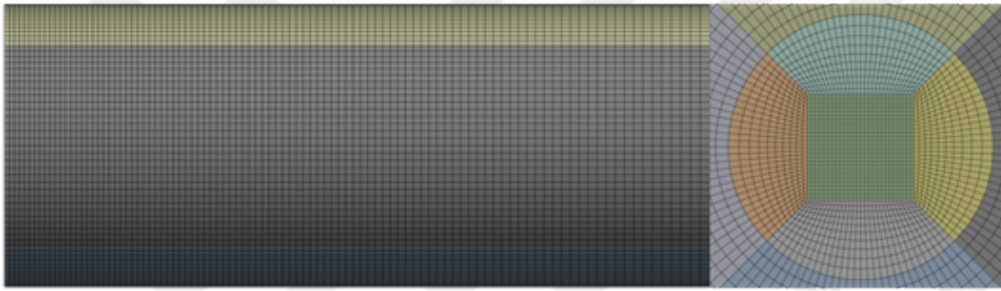


Figure 2.2 : Computational grid.

The quality of the mesh is also of critical importance for CFD analysis. The generated mesh is entirely composed of structural elements, so the quality values are quite good. The maximum skewness value is 0.5, the minimum orthogonal quality is 0.74, and the maximum aspect ratio is calculated as 51.

2.2.3 Initial and boundary conditions

All analyses were run using Workbench parametric calculation methods. Since the problem in this study can be considered relatively simple, the initial condition for velocity was set to 0 m/s and to 1 atm for pressure according to the standard atmospheric condition at $25^\circ C$. The inlet velocities based on Re were applied as velocity-inlet boundary condition as x-velocity component at the inlet cross-sectional area. Cylindrical surfaces parallel to the jet axis and the surfaces orthogonal to the jet axis were defined as 0 Pa atmospheric gauge pressure opening boundary condition.

The convergence criteria for the analyses were defined as the Root-Mean-Square (RMS) values of the conservation equations to be 10^{-6} . In addition, to examine the

accuracy of convergence, the mass balance at the inlet and outlet surfaces of the control volume was monitored, and it was expected that the velocity values at seven points on the jet centerline remained stable.

Since the CFD analyses were run using the Ansys®-CFX software, the COUPLED solution algorithm was used. The high-resolution option was selected for advection scheme and turbulence. The auto timescale option with conservative setting was used for the timescale value required in the COUPLED algorithm, and the timescale was calculated as $4.86 \times 10^{-3} s$. Air at $25^\circ C$ standard atmospheric conditions was defined as the working fluid, and the constant thermophysical properties in the solver database were selected.

2.2.4 Governing equations

The equations solved in this study are continuity and momentum equations along with transport equations for k and ε . Round free jet flow was considered three-dimensional, steady, incompressible and the body forces were neglected. The continuity equation obtained from conservation of mass can be expressed as given in equation 2.1. For steady and incompressible flow, the momentum equations are given in equation 2.2,

$$\frac{\partial (\rho \bar{v}_i)}{\partial x_i} = 0 \quad (2.1)$$

$$\frac{\partial (\rho \bar{v}_i \bar{v}_j)}{\partial x_j} = -\frac{\partial p'}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu_{eff} \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) \right] \quad (2.2)$$

where the modified pressure p' is equal to $p + 2/3 \rho k$. Since the model is based on the eddy viscosity, the effective viscosity μ_{eff} is equal to $\mu + \mu_t$. The $k - \varepsilon$ model has a relation for the turbulence viscosity term which can be calculated with equation 2.3.

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \quad (2.3)$$

The transport equations for k and ε are shown in equation 2.4 and 2.5, respectively, where $v_t = \mu_t / \rho$. For further information readers may refer to Ref. [52].

$$\bar{v}_j \frac{\partial k}{\partial x_j} = v_t \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) \frac{\partial \bar{v}_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[\left(v + \frac{v_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad (2.4)$$

$$\bar{v}_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\varepsilon}{k} \left[C_{\varepsilon 1} v_t \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) \frac{\partial \bar{v}_i}{\partial x_j} - C_{\varepsilon 2} \varepsilon \right] + \frac{\partial}{\partial x_j} \left[\left(v + \frac{v_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \quad (2.5)$$

In equation 2.4 and 2.5; turbulence closure coefficients are $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k and σ_ε and their default values in the solver are 1.44, 1.92, 1.0 and 1.3, respectively. Additionally,

the coefficient C_μ is also considered as the closure coefficient, and its effect has also been examined in this study. Because there is no experimental justification for σ_k , it is not investigated in the calibration process [43]. Ray et al. [47] mentioned that σ_ϵ has a relation with the von Kármán constant $\kappa=0.41$, $C_{\epsilon1}$, $C_{\epsilon2}$ and C_μ .

2.2.5 Calibration of closure coefficients

The calibration of the turbulence model coefficients was carried out based on the velocity and turbulence intensity profiles obtained from the experimental results of Yilmaz et al. [53]. At the first stage of this study, 40 different cases were investigated by changing $C_{\epsilon1}$, $C_{\epsilon2}$ and C_μ values within $\pm 25\%$ margin, including default values of model constants, and analyses were performed for these cases at $Re=10000$. Then, the velocity and turbulence intensity profiles were plotted at 5 stations for different x/D , and the results were investigated visually. In order to expand the investigation, jet half-thickness and jet centerline velocity profiles were also plotted and included in the investigations.

The calibration process was performed using the genetic algorithm with the Design Exploration component in Ansys[®]-Workbench software. For the optimization method used, Multi-Objective Genetic Algorithm (MOGA), design variables and their ranges for optimization need to be determined. Based on the results obtained from 40 different cases, the velocity and turbulence intensity (TI) values were determined for the points farthest from the experimental data as design variables. Accordingly, the velocity and TI values of 6 different points were extracted from the experimental results and determined as the objective function in the optimization. These points are shown in Figure 2.3.

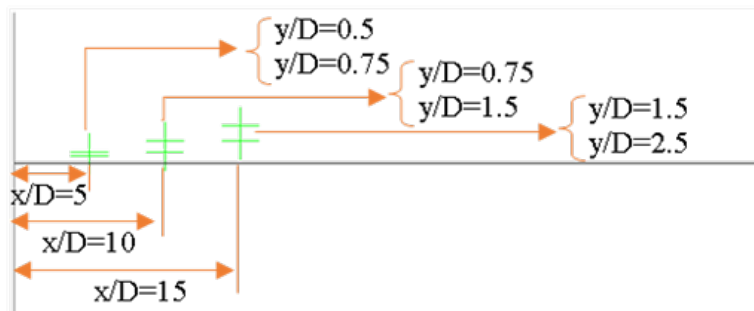


Figure 2.3 : Points selected for optimization.

The MOGA method is an optimization algorithm that aims to solve problems with

multiple, often conflicting objectives. These problems arise frequently in engineering design, where the goal is to optimize a design with respect to several objectives, such as maximizing performance while minimizing cost. The MOGA method uses a genetic algorithm approach to search for the optimal solution in the design space, which is defined by a set of design variables.

The MOGA algorithm operates by maintaining a population of candidate solutions, or individuals, represented by a set of design variables. Each individual is evaluated by a fitness function that reflects its performance with respect to the multiple objectives. The fitness function is a vector function that maps the design variables to the corresponding values of the multiple objectives. The MOGA algorithm then selects the best individuals from the population, based on their fitness values, to create a new generation of individuals through crossover and mutation operators. The new generation is evaluated by the fitness function, and the process continues until a stopping criterion is reached. The advantage of the MOGA method is that it provides a set of optimal solutions representing a trade-off between multiple objectives, instead of a single optimal solution that may not be satisfactory with respect to all objectives. This set of solutions is known as the Pareto optimal set, which represents the optimal solutions that cannot be improved with respect to one objective without worsening the performance with respect to another objective. Adaptive Multi-Objective method with 24 initial samples was chosen for MOGA. 6 number of samples per iteration with 75% maximum allowable pareto percentage ran for maximum 10 numbers of iterations. The optimization converged after 6 successful iterations.

2.3 Results and Discussion

2.3.1 Calibration of model coefficients

All cases were created by changing $C_{\epsilon 1}$, $C_{\epsilon 2}$ and C_{μ} coefficients in the $\pm 25\%$ margin. The Default case has default coefficients provided by the solver. The cases are shown in Table 2.1. The CFD results were obtained for $Re=10000$ and Figure 2.4 shows the comparison of the velocity profiles with experimental results at 5 different stations along the round jet.

At low x/D ratios i.e. 0.5, velocity profiles do not quite match the measurements. The reason is that there are few measurement points for the velocities taken at distances

Table 2.1 : Cases with modified coefficients.

Cases	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	C_{μ}
Default	1.4400	1.9200	0.0900
Case 1	1.5120	2.0160	0.0945
Case 2	1.5840	2.1120	0.0990
Case 3	1.6560	2.2080	0.1035
Case 4	1.7280	2.3040	0.1080
Case 5	1.8000	2.4000	0.1125
Case 6	1.3680	1.8240	0.0855
Case 7	1.2960	1.7280	0.0810
Case 8	1.2240	1.6320	0.0765
Case 9	1.1520	1.5360	0.0720
Case 10	1.0800	1.4400	0.0675
Case 11	1.5120	1.9200	0.0900
Case 12	1.5840	1.9200	0.0900
Case 13	1.6560	1.9200	0.0900
Case 14	1.7280	1.9200	0.0900
Case 15	1.8000	1.9200	0.0900
Case 16	1.3680	1.9200	0.0900
Case 17	1.2960	1.9200	0.0900
Case 18	1.2240	1.9200	0.0900
Case 19	1.1520	1.9200	0.0900
Case 20	1.0800	1.9200	0.0900
Case 21	1.4400	2.0160	0.0900
Case 22	1.4400	2.1120	0.0900
Case 23	1.4400	2.2080	0.0900
Case 24	1.4400	2.3040	0.0900
Case 25	1.4400	2.4000	0.0900
Case 26	1.4400	1.8240	0.0900
Case 27	1.4400	1.7280	0.0900
Case 28	1.4400	1.6320	0.0900
Case 29	1.4400	1.5360	0.0900
Case 30	1.4400	1.4400	0.0900
Case 31	1.4400	1.9200	0.0945
Case 32	1.4400	1.9200	0.0990
Case 33	1.4400	1.9200	0.1035
Case 34	1.4400	1.9200	0.1080
Case 35	1.4400	1.9200	0.1125
Case 36	1.4400	1.9200	0.0855
Case 37	1.4400	1.9200	0.0810
Case 38	1.4400	1.9200	0.0765
Case 39	1.4400	1.9200	0.0720
Case 40	1.4400	1.9200	0.0675

close to the jet exit.

Figure 2.5 shows the comparison of TI profiles with experimental results at 5 different stations along the round jet. It should be underlined that not all 40 cases yielded physically meaningful results according to the measurements, only the successful

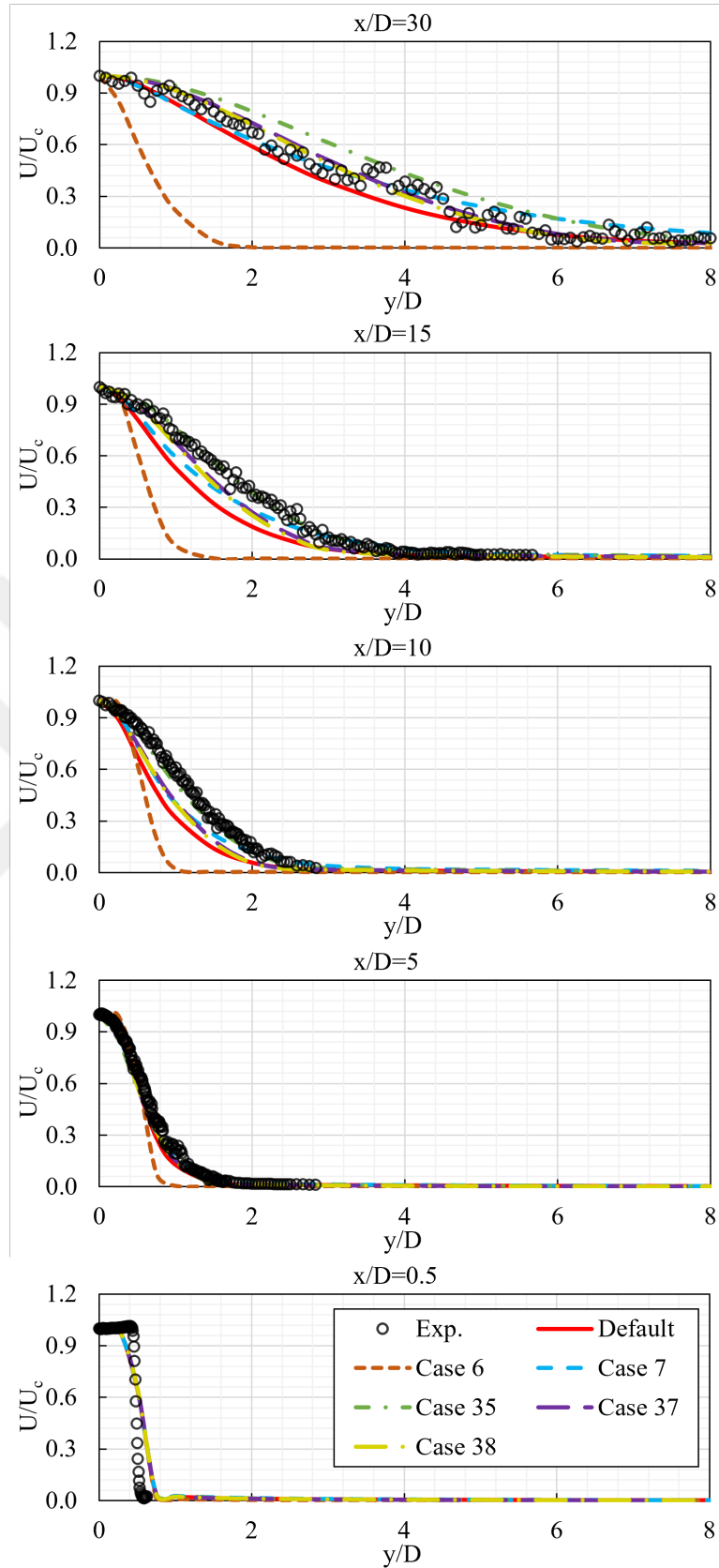


Figure 2.4 : Velocity profiles for $Re=10000$.

predictions are shown in Figure 2.4 and 2.5 .

After evaluating the velocity profiles and TI obtained from the CFD analyses for all

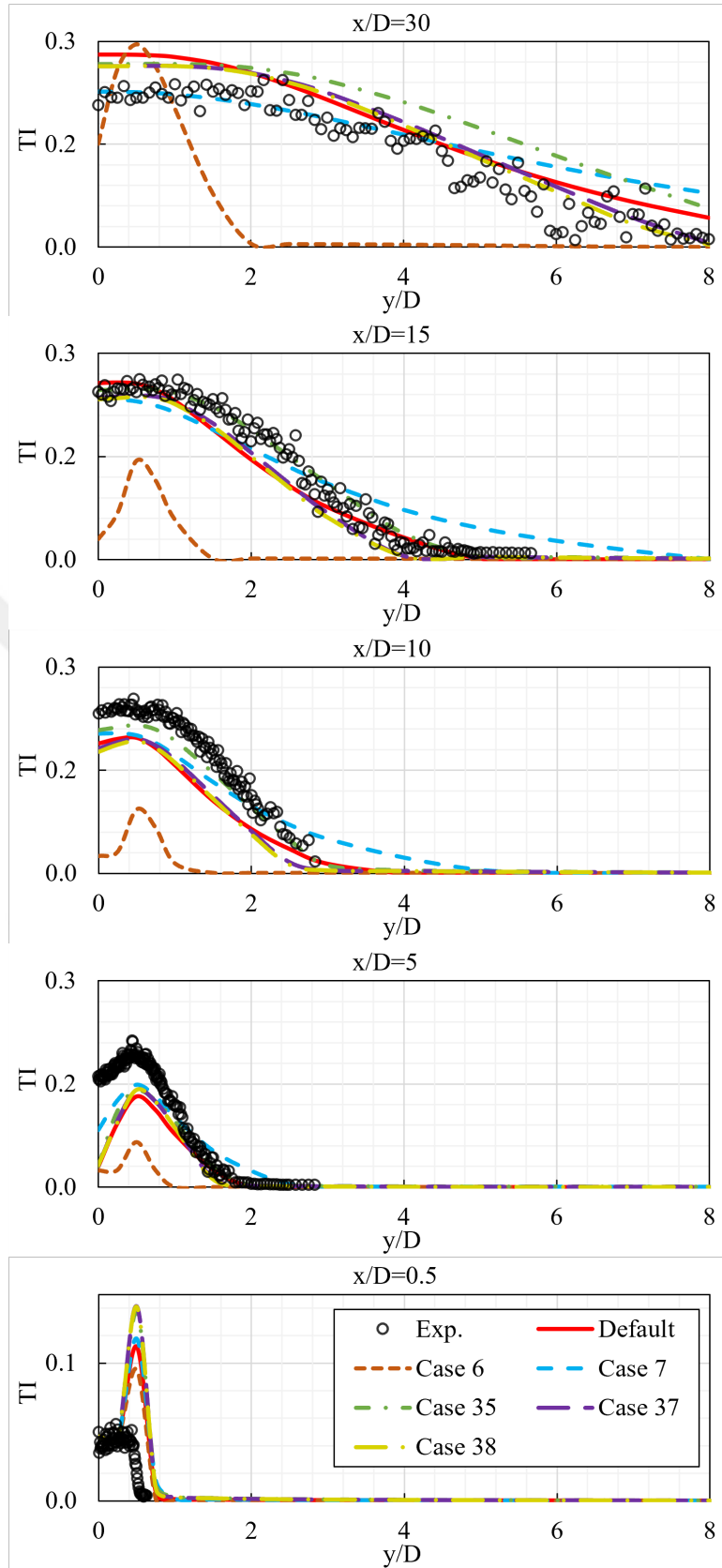


Figure 2.5 : TI distributions for $Re=10000$.

cases, the jet centerline velocity profiles and the jet half-widths for the most successful cases are given in Figure 2.6 and Figure 2.7, respectively.

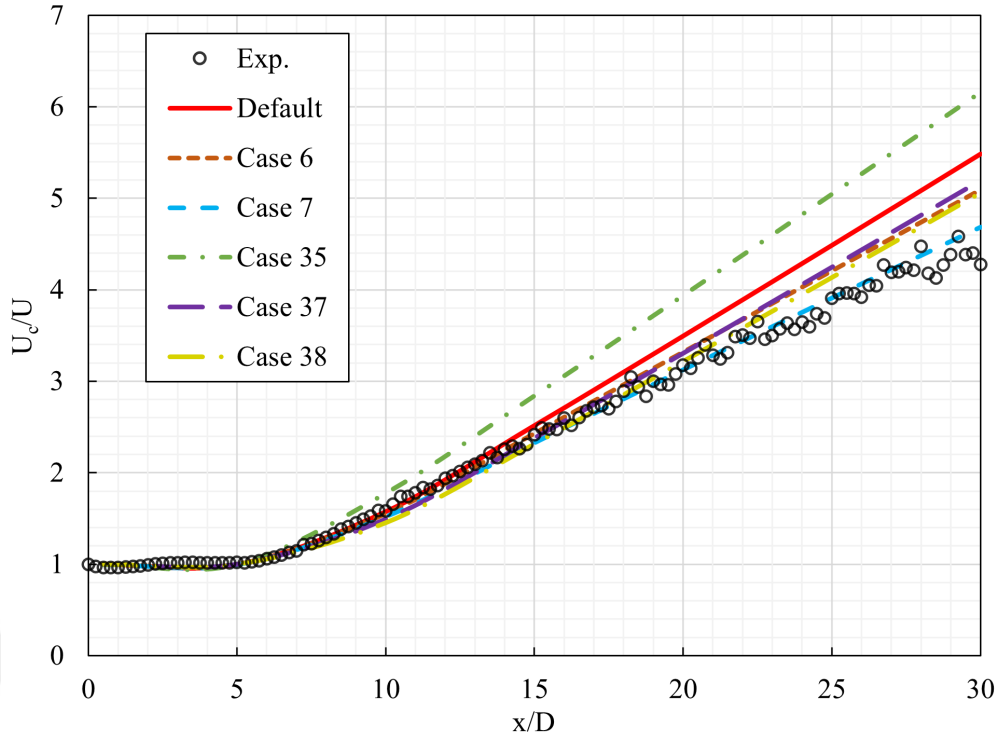


Figure 2.6 : Jet centerline velocity distribution for Re=10000.

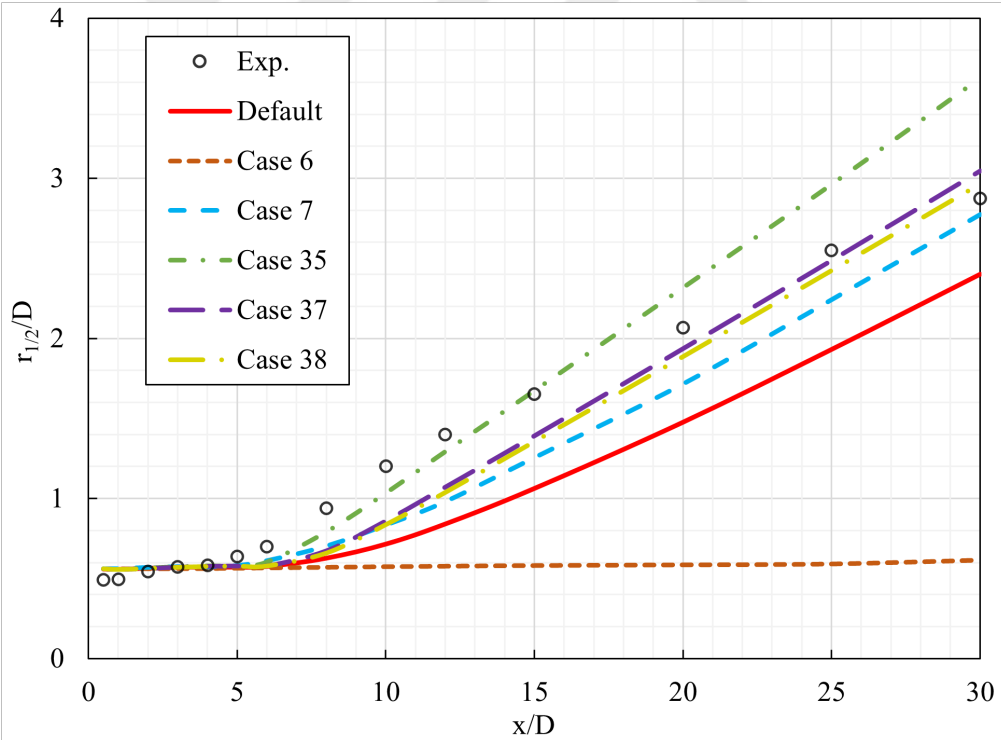


Figure 2.7 : Jet half-width distribution for Re=10000.

When the results obtained from modified the coefficients within $\pm 25\%$ range are evaluated, it can be seen that velocity profiles can be improved by changing the model constants. The variation of the $C_{\epsilon 1}$ coefficient directly affects the production term in the dissipation equation. Increasing the value of $C_{\epsilon 1}$ affects the CFD results to become

closer to the experimental results when $x/D \geq 20$. The variation of the $C_{\varepsilon 2}$ coefficient affects the dissipation term in the dissipation equation. However, changing this value alone within the range of $\pm 25\%$ has a negative impact on the results at all stations. It was observed that the increase in the value of C_{μ} , which is related to the turbulence viscosity, provides closer CFD results to the experimental measurements in the range of $10 \leq x/D \leq 20$.

In the second stage of the study, MOGA was run to reveal the coefficients that will be closer to the velocity and TI values of the measurements. The results were evaluated through the best candidate determined as a result of the optimization. The comparison of the default and calibrated prediction performances is shown in Figure 2.8 for $Re=10000$.

When the velocity profiles obtained from CFD simulations were compared with the experimental measurements, it was observed that only the default and calibrated model coefficients did not well matched at $x/D=10$. However, the results have a good agreement with the experimental measurements for $x/D < 10$ and $x/D > 15$. It can be concluded that the calibrated model coefficients provide a better fit than the standard model coefficients.

It is important to prove the validity of the calibrated coefficients at different Reynolds numbers for the reproduction of the results. Therefore, it is essential to ensure that the closure coefficients are applicable for a wide range of Reynolds numbers. The results obtained for $Re=5000$ and 20000 using the same initial and boundary conditions and computational grid as for $Re=10000$ case indicated that the new calibrated closure coefficients were able to provide satisfactory. The jet velocity profiles obtained using the calibrated coefficients show good agreement with measurements at $Re=20000$ as shown in Figure 2.9, indicating that the new coefficients are robust and reliable.

As Figure 2.10, Figure 2.11, and Figure 2.12 show, the new coefficients provided better predictions for the centerline velocity profiles than the standard model coefficients at all Re and showed a perfect match. These findings suggest that the use of the calibrated closure coefficients can lead to improved predictions in round free jet problems in a wide range of Reynolds numbers.

The new model coefficients obtained from the optimization study for $C_{\varepsilon 1}$, $C_{\varepsilon 2}$ and C_{μ} were found 1.2098, 1.595, and 0.08894, respectively. They differ from the

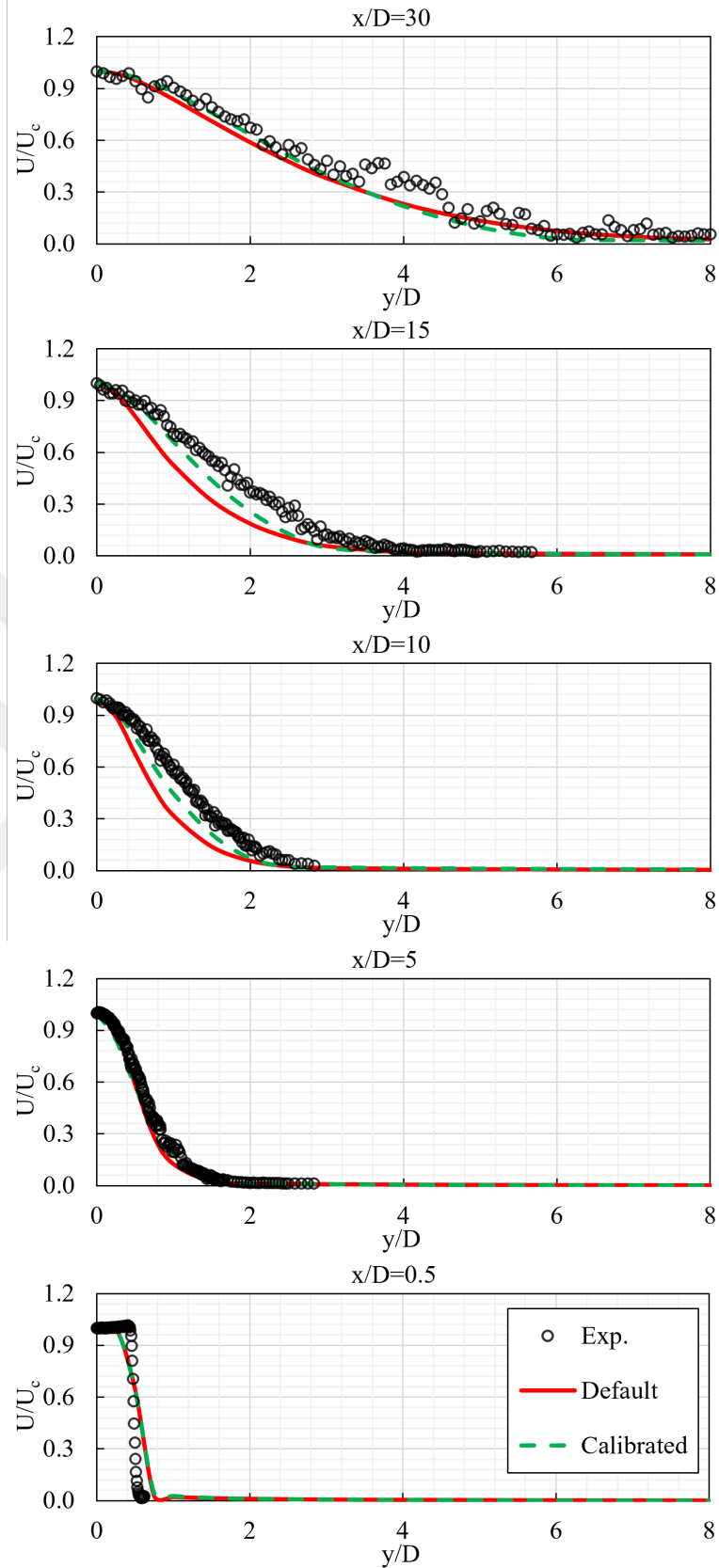


Figure 2.8 : Velocities with calibrated coefficients for $Re=10000$.

standard model coefficients by 16% for $C_{\epsilon 1}$, 17% for $C_{\epsilon 2}$, and 1.2% for C_{μ} . The results obtained with the calibrated and optimized coefficients were closer to the

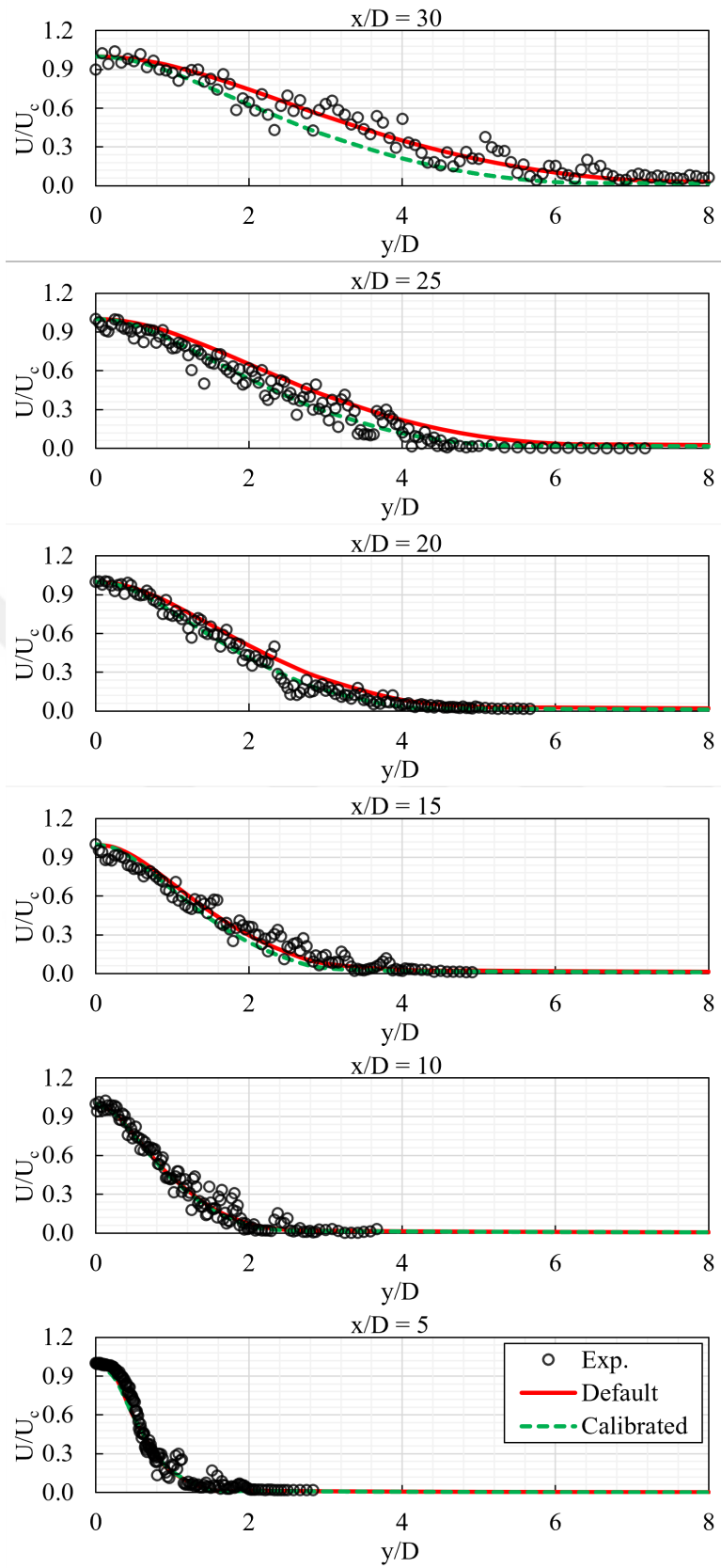


Figure 2.9 : Velocities at $Re=20000$ with adjusted coefficients.

measurements compared to the standard coefficient results.

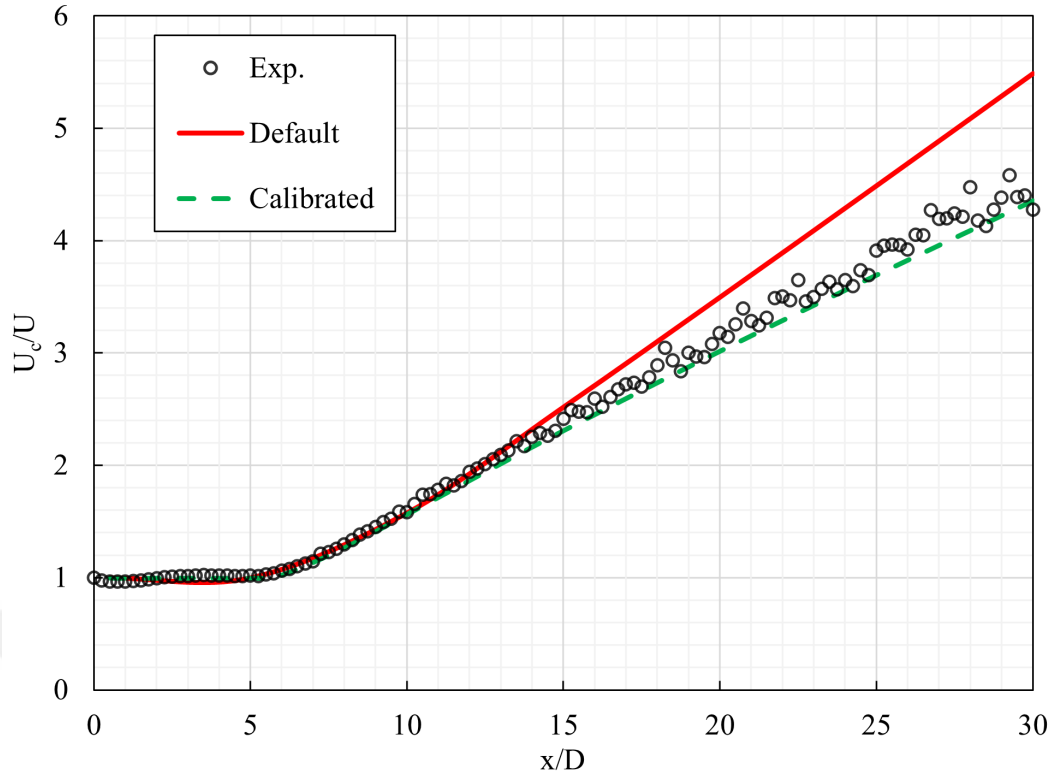


Figure 2.10 : Jet centerline velocity distributions for $Re=10000$ with adjusted coefficients.

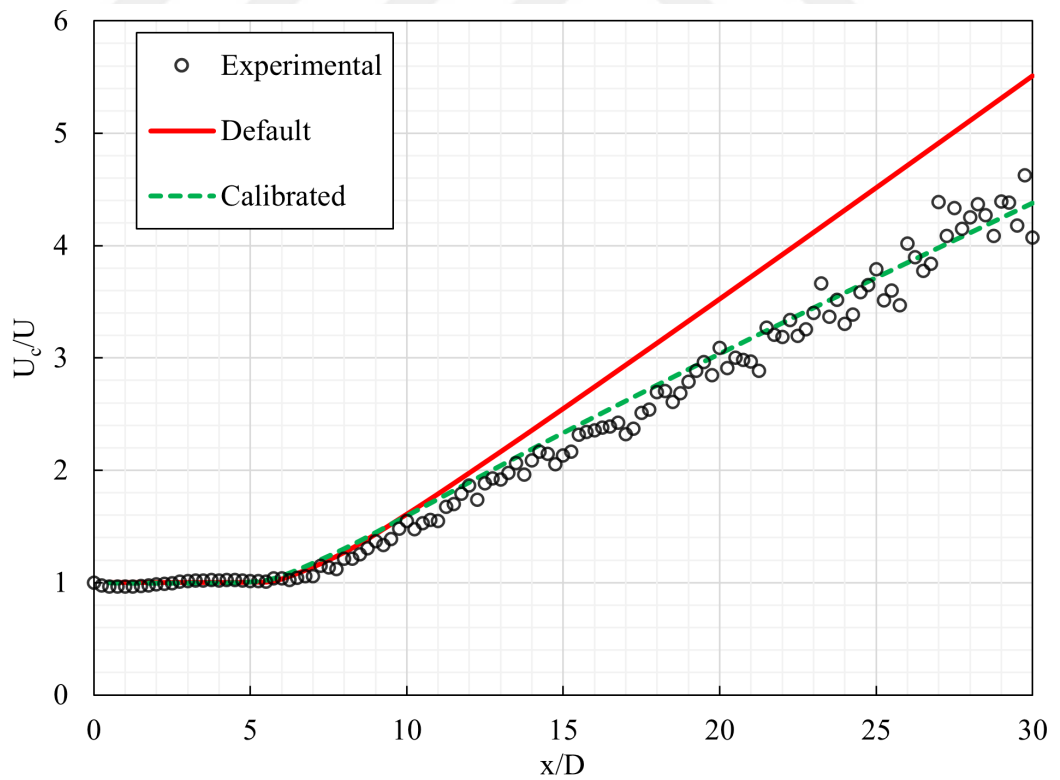


Figure 2.11 : Jet centerline velocity distribution for $Re=5000$ with adjusted coefficients.

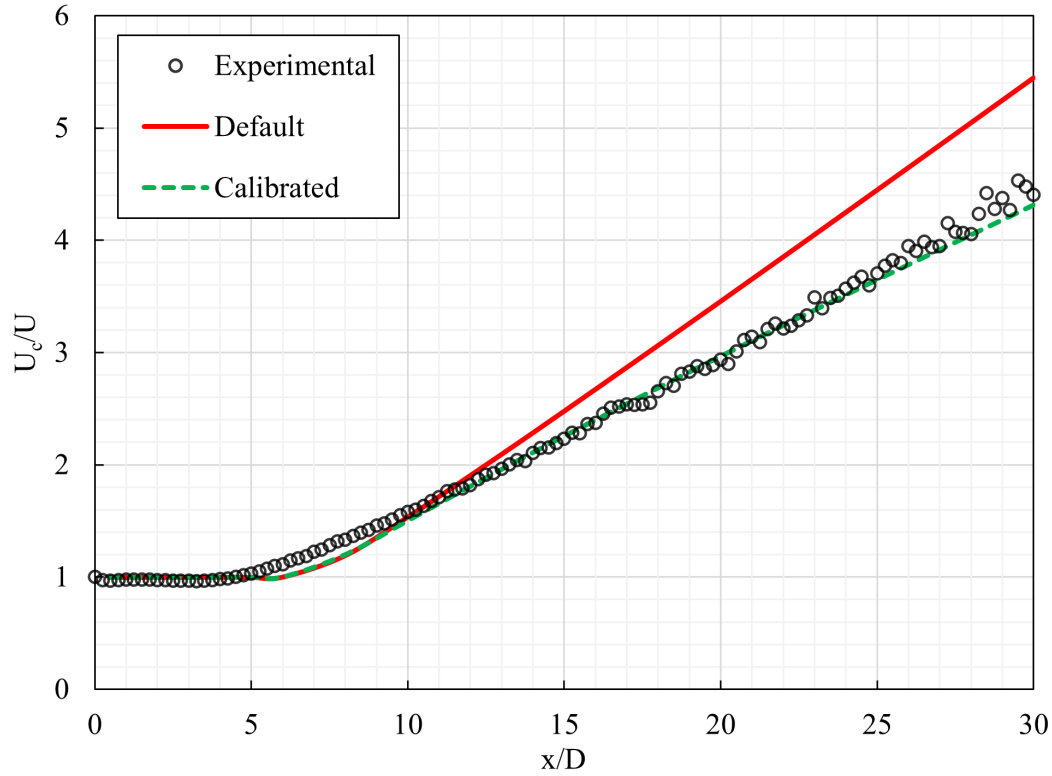


Figure 2.12 : Jet centerline velocity distribution for $Re=20000$ with adjusted coefficients.

2.4 Conclusion

In this study, calibration of the standard $k - \varepsilon$ model coefficients was performed using MOGA method with respect to experimental data for a round free jet at $Re=10000$. The effect of modifying the turbulence model coefficients, $C_{\varepsilon 1}$, $C_{\varepsilon 2}$ and C_{μ} , on the velocity and TI profiles taken from different jet stations were examined. Comparing the jet centerline velocities, it was revealed that the calibrated model coefficients matched the experimental results perfectly. The RANS turbulence closure coefficients are, in fact, adjustable constants, and their optimal values vary depending on the specific flow problem at hand. As such, it is suggested that these coefficients must be calculated specifically for each problem in order to ensure accurate and consistent results. By this way, the calculation errors due to uncertainty of turbulence modeling encountered in CFD calculations can be overcome. The adjusted and optimized coefficients proposed in this study can be safely used in commercial CFD solvers to simulate round free jets under the same boundary conditions.

3. CALIBRATION OF THE REYNOLDS STRESS MODEL FOR TURBULENT ROUND FREE JETS BASED ON JET HALF-WIDTH²

3.1 Introduction

Free jet flows have been extensively examined through both experimental and numerical studies, especially within the realms of aeronautics, as they are generated by the exhaust flow of aircraft engines or used for blade cooling. Additionally, they are prevalent in various industrial and environmental contexts, such as industrial burners, water jets in fountains, laboratory jets, rocket launches, and cooling jets in electronics, among other sources. The dynamics of jet flows are very complicated and significantly affected by factors such as the Reynolds number, nozzle diameter, fluid properties, and flow conditions. Chen and Jaw [41] emphasized that in free shear flows, the pressure gradient and viscous diffusion are negligible and do not significantly influence the flow field; thus, free shear flows are frequently chosen to test the accuracy of turbulence models. The prediction of free-shear flows is highly sensitive to the accurate modeling of Reynolds stress, turbulent kinetic energy, and its dissipation rate. The absence of near-wall turbulence complexity in free jet flow allows precise evaluation of a turbulence model's accuracy in predicting the general flow field without the interference of wall turbulence.

The flow regime of free jet problems predominantly consists of turbulent flows. Turbulent Navier-Stokes equations cannot be solved directly and require High-Performance Computing due to their non-linear nature. With HPC, the direct solution of turbulent flow field using the Direct Numerical Simulation method or partial solution of turbulent structures with Large Eddy Simulation is possible for some relatively simple cases. However, this approach requires highly expensive computer systems. Even with all this infrastructure, HPC still falls short in solving complex and large flow problems found in nature. Therefore, modeling turbulence reduces

²This chapter is based on the paper: Turutoglu, C., Cadirci, S., Yilmaz, S. and Erdem, D. (2024). Calibration of the Reynolds stress model for turbulent round free jets based on jet half-width, *Physics of Fluids*, 36 (11), 115133.

computational costs and allows the simulation of more complex and larger problems even on personal computers. The models used to enable the solution of turbulent flows employ model equations to facilitate this solution rather than directly solving the Reynolds stress term. To clarify the flow physics and determine the optimal design of flow engineering systems, it is necessary to accurately predict the flow fields. Reynolds Averaged Navier-Stokes simulations are the preferred tools for predicting flow fields on an industrial scale due to their computational efficiency. However, current turbulence models are not capable of accurately predicting turbulent mean flows. Compared to experimental data, traditional turbulence models typically struggle to estimate the production and dissipation rates of the flow both near the jet exit and in the far field of the jet.

The early comprehensive experimental studies on jet flows by hot wire anemometer have been carried out by many researchers [27, 54, 55]. It has been observed by Corrsin [54] that velocity and turbulence distributions are radially symmetric in the fully developed flow regime. Hinze and Zijned [55] have examined radial and axial distributions by measuring time-dependent velocity, temperature, and gas concentration. Wygnanski and Fiedler [27] investigated incompressible, and isothermal axisymmetric turbulent jets by using a constant temperature anemometer in 1969. In this study, turbulence intensities were examined in a manner similar to the previous studies. After the flow profiles, the energy balance was calculated through the measurements, and differences from previous studies were interpreted. Bradshaw [56] has performed hot wire anemometer experiments on round jets and revealed turbulence structures in detail. Hussein and George [57] conducted experimental studies using a moving hot wire probe and captured turbulent dissipation components for an axisymmetric jet, and showed that the jet was not regionally equal in all directions, but regionally axisymmetric at small scales. In addition, this study is one of the pioneering studies in which direct measurement of dissipation was realized for this type of flow. Additionally, Hussein and George [58] investigated the effects of the measurement devices by comparing hot wire and Laser Doppler Anemometry (LDA) and revealed that the use of hot wire is not reliable for measuring flows with high turbulence intensities. More experimental studies on round jet flow have been carried out by Lemieux and Oosthuizen [59], who showed that turbulence lateral normal and shear stress level, jet spread rate, and change of centerline velocity were strongly

affected by Reynolds number; on the other hand, longitudinal normal stress level and non-dimensional mean velocity profiles were not perturbed by Reynolds number.

Turbulent jet flow can be categorized under two general effects: Reynolds number (Re) and Mach number. Incompressible jet flows are considered in this study; the Mach number effect was not examined. One of the most recent studies examining the effect of Reynolds number on jet flow is by Fellouah et al. [60], who examined the near region and transition region of the jet flow ($0 \leq x/D \leq 25$) with a constant temperature anemometer at three different Reynolds numbers. They showed how Reynolds number affects the average velocity profile, turbulence intensity, velocity spectra, nozzle exit boundary layer thickness, and how the length of the potential core region changes. Toddle et al. [61] performed the free jet flow experiment at low Reynolds numbers using a hot-wire anemometer measurement and observed that the behavior of jet flows with low Reynolds numbers is different from jet flows with high Reynolds numbers. In addition, it was observed that the flow had more than one regime at the Reynolds numbers examined in this study, and as a result of the experiments, information was gained about the characteristic features of these regimes. Experimental data was acquired by Dimotakis [62] experimentally showed that the critical Reynolds number should be 10^4 for a fully developed turbulent zone, and below this threshold, the flow cannot be considered as fully developed and has a different structure. Another study on the effects of Reynolds number in the jet near region was performed by Suresh et al. [63] in 2008. It was shown that at low Reynolds numbers, the axial distances where the average velocity profiles show self-similarity were shorter than those of the turbulence intensity profiles, and it was also observed that the vortices related to the instability of the shear layer in the near region had different dominant frequencies. In addition, velocity profiles, turbulence intensities, center line velocity deceleration, and turbulence length scale are directly related to Reynolds number and axial distance at low Reynolds numbers. On the other hand, at high Reynolds numbers, turbulence in the far field was observed to be independent of Reynolds number and axial distance.

Jet flow studies on Reynolds number-dependence were also examined systematically by Deo et al. [64]. Experiments were conducted in an open circuit wind tunnel at Reynolds number varying between 4500 and 16500. In their study, researchers examined the effects of Reynolds number on potential core, near field spreading rate, and turbulence intensity. Reynolds dependence of power spectra of axial direction

velocity was pointed out, and large-scale coherent structures in the near field were observed. Despite the increase in Reynolds number, it was observed that the velocity profiles had a similar top hat structure, but the uniform flow expanded sideways, and the outlet boundary layer thickness decreased. It was concluded that this flow feature is associated with the decrease in turbulence intensity in the regions of interest. It was observed that the increment in Reynolds number causes the potential core region to shorten and the near region propagation speed to increase. The study also demonstrates that the peak values in the jet-near-region turbulence intensity distributions decrease with the Reynolds number. The dependency of turbulence properties on the Reynolds number in the transition region of a circular jet was also investigated using excessive hot-wire measurement test by Mi et al. [65]. It was concluded that the mean flow was Re-independent for Re_D higher than 10^4 and was strongly affected when Re_D is lower than 10^4 . Additionally, small scale turbulence properties such as dissipation of turbulence kinetic energy was shown to be different above and below Re_D of 10^4 , assuming this value to be critical.

Although there are many turbulence models applicable to various fluid-flow types, there is still no universal turbulence model that can deliver highly successful results in all problems. Consequently, different turbulence models have their advantages and disadvantages in different flow problems, and the experience and knowledge of the engineer solving the problem play a crucial role in the selection and application of the model to be used. When examining studies that investigate free jet flow problems using the Computational Fluid Dynamics method, one can see that many different turbulence models are used. Perez et al. [66], Kmecova et al.[67], and Weaver and Miskovic [68] investigated the comparison of turbulence models across diverse free jet scenarios. Despite employing various solvers to test all turbulence models, they highlighted their inability to achieve results that closely aligned with experimental data or theoretical calculations.

Kendil et al. [69] mentioned that the $k - \varepsilon$ model describes the average axial velocity well, whereas the turbulence kinetic energy is better represented by the Speziale, Sarkar, and Gatski's [70] (SSG)-Reynolds Stress Model (RSM) model for an axisymmetric jet problem. Meslem et al. [71] revealed that none of the turbulence models can accurately predict all jet characteristics simultaneously in simulating jet flow from a cross-shaped orifice. The Reynolds Stress Model provides

better agreement between numerical results and experimental data for local jet flow expansion, while global flow expansion and ambient air induction are better predicted by the Shear Stress Transport (SST) $k - \omega$ turbulence model. Miltner et al. [72] concluded that the simple and inexpensive standard $k - \varepsilon$ model serves very well to describe the effects in most regions of the straight and slightly swirling turbulent free jets since the model was explicitly developed for free shear flows. Extending the model to the Renormalization Group $k - \varepsilon$ and Realizable $k - \varepsilon$ are disadvantageous in current application. The original development of the standard $k - \omega$ model was based solely on wall-bounded shear flow, making it entirely inapplicable to turbulent free jets. SST $k - \omega$ uses a blended formulation of $k - \omega$ near solid walls and $k - \varepsilon$ in the free shear region, providing higher flexibility for various flow types combined with moderate computational demand. Li et al. [73] used the $k - \varepsilon$ turbulence model for circular jet problems and found that the standard $k - \varepsilon$ turbulence model was not capable of accurately predicting the flow characteristics. Therefore, model calibration was needed, which significantly improved the results. These examples underscore the necessity of calibrating turbulence models to achieve better alignment with experimental data.

Calibration not only refines model accuracy but also enhances the reliability of CFD simulations, making them more applicable to complex and elaborate flow scenarios. Various studies have demonstrated that through proper adjustment, turbulence models can significantly improve their predictive capabilities, thereby providing more accurate and reliable results. As Pope [74] highlighted, modified models can sometimes offer improvements over standard models in different flow conditions. In the literature, there are many studies that have calibrated various turbulence models using different methods and demonstrated that the results obtained through numerical methods using these calibrated turbulence models are more consistent with those obtained through experimental methods or DNS.

Birch et al. [75] showed that better agreement with experiments can be obtained when model constants are assessed for free jet flows using the standard $k - \varepsilon$ turbulence model. Guillas et al. [76] conducted Bayesian calibration for uncertainty analysis in CFD modeling of urban flows. Focusing on turbulent kinetic energy (TKE) in a regular street canyon, they evaluated standard $k - \varepsilon$ turbulence model constants with wind tunnel observations. While tuning these constants did not

achieve perfect alignment with experimental data, their study highlighted avenues for future research. Costa Rocha et al. [77] carried out a computational investigation focusing on the calibration of a turbulence model for a small-scale horizontal axis wind turbine, emphasizing the calibration's independence from turbulence intensity for specific values of β^* , and identified an optimal β^* value of 0.45 for their case. Costa Rocha et al. [78] refined their approach using the root-mean-square error (RMSE) method and determined that β^* value of 0.27 was more suitable for their simulations. Shiradzi et al. [79] improved the case of air flow around buildings in the turbulent atmospheric boundary layer based on stochastic optimization and Monte-Carlo sampling techniques and decreased the error of the estimated reattachment length ratio behind buildings from 170% to 28% using the standard $k - \varepsilon$ turbulence model. Mauro et al. [80] optimized local correlation parameters of the $\gamma - Re_\theta$ transition turbulence model using a micro genetic algorithm for a 2D-NACA-0018 airfoil. Their calibrated parameters, validated with S809 profile data, showed satisfactory performance in simulations, indicating potential applicability to other airfoil configurations. Shiradzi et al. [81] proposed a systematic rectification methodology to enhance urban air flow simulations using CFD simulations based on the RANS turbulence models. In the rectification process, high-fidelity data from different sources were used to define the validation metrics, which are then utilized as the objective function in a stochastic optimization solver to find optimum values for closure coefficients of the RANS turbulence model. Da Ronch et al. [82] focused on calibrating the Spalart-Allmaras turbulence model, identifying key closure coefficients that significantly impact output quantities of interest. Their adjusted model for transonic, wall-bounded flows around the RAE 2822 airfoil outperformed the standard version across various test cases, highlighting its broader applicability and prediction accuracy. Khan et al. [44] utilized genetic algorithms to fine-tune model constants of standard $k - \varepsilon$ model ($C_\mu, C_{\varepsilon 1}, C_{\varepsilon 2}$, and σ_k) for rib-roughened narrow rectangular channels, conducting 2D CFD analyzes. They achieved substantial improvements in performance metrics for different channel heights and Reynolds numbers, emphasizing the effectiveness of calibration in optimizing CFD simulations. Yoder and Orkwis [83] applied formal optimization procedures to recalibrate the Explicit Algebraic Reynolds Stress Model (EARSM) for the self-similar region of the incompressible planar mixing layer. By adjusting turbulence model coefficients

and evaluating optimization fitness based on experimental data, they demonstrated significantly improved agreement with experimental results for this specific flow scenario. Chowdhary et al. [50] conducted the first Bayesian calibration of the SST $k - \omega$ turbulence model for complex hypersonic turbulent flows, where they integrated machine-learning techniques with projection-based model reduction methods to develop data-driven surrogate models for high-fidelity physics models. Their approach was applied to simulate heat flux and pressure on the HIFiRE-1 geometry in Mach 7.16 turbulent flow, demonstrating significant improvements in prediction accuracy. Romanova et al. [84] utilized machine learning techniques to adjust the closure coefficients of the SST $k - \omega$ turbulence model to simulate free surface flows on mountain slopes and achieved improved results. Song et al. [85] enhanced the prediction accuracy of separated flow in a diffuser using a recalibrated RSM turbulence model in which they employed the Ensemble Kalman Filter (EKF) data assimilation method to adjust the closure coefficients of the baseline RSM model, using wall pressure coefficients as key data inputs, and showed upgraded prediction capability. Subbian et al. [86] enhanced the performance of the SST $k - \omega$ turbulence model to simulate flows around aircraft by introducing physical extensions to the ω length-scale equation and employing uncertainty quantification methods for model-alternation. The alternation process initially focused on cases such as flows over backward facing step, axisymmetric transonic bump, and Model53 delta wing at 20° angle of attack. Successive adjustments were conducted for each test case, ensuring reliable updates and ultimately, a comprehensive turbulence model-amendment was presented. The study showed amended surface pressure distributions over the delta wing model compared to the default cases. Yildizeli and Cadirci [87] demonstrated that the standard closure coefficients of the SST $k - \omega$ turbulence model lead to underestimation of diffusion and production terms associated with turbulence variables in the context of the backward-facing step problem. They found that model remediation affects velocity profiles in the log-law region, whereas it has limited effectiveness in the buffer zone. Through the calibration of model closure coefficients, they showed enhancements in velocity profiles, particularly within the log-law region. He et al. [88] adjusted the Spalart-Allmaras turbulence model coefficients with the EKF approach for ARN2 subsonic nozzle geometry using jet exit velocities obtained from Particle Image Velocimetry (PIV) measurements. They

achieved a reduction in average relative error of jet velocity calculations from 13% to 4.6%. Liang et al. [89] corrected the SST $k - \omega$ turbulence model coefficients using experimental wall pressure profiles from a cavity-ramp geometry, employing DNN and genetic algorithm techniques, and successfully reduced the average relative error of wall pressure from 9.4% to 2.7%. Usov et al. [90] conducted a model-adjustment for differential RSM using DNS data of turbulent channel flow at friction Reynolds numbers ranging from 550 to 5200. The study aimed to develop a RANS sublayer model suitable for integration into a hybrid RANS/LES framework. Their adjusted model demonstrated improved mean velocity profiles and peak Reynolds stress values compared to the default model. Younoussi and Ettaouil [91] performed a numerical study to enhance the accuracy of the SST $k - \omega$ turbulence model in wind turbine simulations near stall conditions and adjusted α_1 and β^* coefficients in the turbulence model. They later applied this approach to the NREL S826 NTNU wind rotor, with the aim to modify power and thrust coefficients in accordance with the experimental data. The results showed a significant reduction in relative error from 18% to 4.5% during light stall conditions at tip speed ratios between 3 and 6.

Furthermore, genetic algorithms have emerged as a powerful tool for turbulence model calibration, as demonstrated in several studies, offering a robust method to optimize model parameters and achieve improved predictive capacity. For instance, Mauro et al. [80] also used genetic algorithms to optimize local correlation parameters of the $\gamma - Re_\theta$ transition model for a 2D airfoil. The study showed the potential of genetic algorithms to enhance turbulence model predictions in many different configurations. Similarly, Khan et al. [44] applied genetic algorithms to correct turbulence model constants for rib-roughened narrow rectangular channels and showed substantial improvements in performance metrics. Liang et al. [89] utilized genetic algorithms to modify the SST $k - \omega$ turbulence model coefficients using experimental wall pressure profiles from a cavity-ramp geometry. This approach significantly reduced the average relative error of wall pressure, demonstrating the effectiveness of genetic algorithms in improving model accuracy.

Since the pressure gradient and viscous diffusion are negligible and do not significantly influence the flow field, free shear flows are frequently chosen to test the accuracy of turbulence models, as mentioned by Chen and Jaw [41]. The prediction of free shear flows is highly sensitive to the accurate modeling of Reynolds stress, turbulent

kinetic energy, and its dissipation rate. The absence of near-wall turbulence complexity in free jet flows allows the precise evaluation of a turbulence model's accuracy in predicting the general flow field without the interference of wall turbulence. Davidson [15] pointed out that in the RSM, the production terms do not need to be modeled, simplifying the calculations and potentially reducing computational costs while maintaining high accuracy. This supports the use of RSM in jet flow simulations where detailed turbulence modeling is crucial. D'Auria [92] emphasized that the RSM can show better performance in free shear flows with strong anisotropy, in flows with sudden changes in mean strain rate, and in flows where the strain fields are complex. This makes RSM particularly advantageous for modeling free jet flows, which often exhibit such characteristics. In this context, RSM has been recognized for its superiority to accurately predict flows with complex behaviors such as jet flows.

Various studies have underlined the advantages of the RSM turbulence model for free jet flows, making it a preferred choice for CFD studies. Kendil et al. [69] highlighted that while the standard $k - \varepsilon$ model can describe the average axial velocity well, the RSM better represents the turbulence kinetic energy in axisymmetric jet problems, suggesting that RSM can offer more precise predictions for turbulent jet flows. Meslem et al. [71] found that no RANS-based turbulence model can accurately predict all jet characteristics simultaneously. However, they noted that the RSM provides better agreement between numerical results and experimental data for local jet flow expansion. This highlights the superiority of the RSM model over other models in handling the complexities of jet flow dynamics. Yoder and Orkwis [83] applied formal optimization procedures to revise the coefficients in the EARSM for the self-similar region of the incompressible planar mixing layer. Their study showed that this procedure resulted in better agreement with experimental data, demonstrating the model's potential for high-accuracy predictions in jet flows. Song et al. [85] underscored the efficacy of RSM in improving the accuracy of separated flow predictions. The amendment of the RSM using data assimilation methods led to significantly better predictive performance, indicating the model's robustness and reliability in complex flows. Lovella et al. [93] demonstrated that a variant of the RSM should be considered as the most reliable turbulence model to simulate reacting jet flows. Their study shows that RSM can provide more accurate results compared to other models, making it particularly suitable for complex jet flow scenarios.

In summary, these studies have pointed out the significant advantages of using the RSM for jet flow problems. Its application can lead to more reliable and precise predictions, ultimately enhancing the design and analysis of engineering systems involving jet flows. Considering all these reasons, Launder-Reece-Rodi's [94] (LRR) Isotropization of Production (IP) RSM has been used as the turbulence model in the numerical analysis of the free jet flows examined in the numerical part of this study. However, it needs to be calibrated for specific flow cases. Numerous studies collectively highlight the significant advantages of using and calibrating the RSM for specific jet flow and other fluid dynamics problems [71, 85, 90, 93]. Proper calibration ensures that the RSM can accurately capture the complexities of turbulent flows, making it an invaluable tool to achieve reliable and precise predictions in various engineering applications.

In the experimental part of this study, velocity and turbulence were measured at different stations at the exit section of a circular jet by using a hot wire anemometer at Istanbul Technical University Trisonic Research Center. Velocity and turbulence profiles were measured at the near and transition zone of jet flow. Through the data obtained, the fully developed flow structure, jet propagation velocities, and the change of the center line velocity in the axial direction were examined. The jet half-widths obtained by experimental methods for a jet Reynolds number of 10000 were used in the optimization process of the IP-RSM turbulence model coefficients. This approach aimed to improve CFD calculations of jet half-widths along the jet axis for round free jets. The effects of the amended coefficient combination on the RSM equations were examined, and the strengths and weaknesses of the model for this specific problem were discussed. A sensitivity analysis was conducted for the closure coefficients, and the impact of the coefficients at different stations on the jet half-width calculations was evaluated. To test the independence of the coefficients from jet Reynolds number, CFD analyses were repeated for Re of 20000 using the calibrated coefficient combination.

3.2 Methodology

3.2.1 Experimental setup

Experiments are conducted in the Trisonic Research Laboratory of Istanbul Technical University. Experimental setup consists of a jet flow unit which generates a round free jet discharging from a nozzle, and an air supply system. Compressed air used to

generate a jet flow is stored in a tank. Flow is obtained by passing the pressurized air through a pressure regulator and then directing it into a stagnation chamber. The stagnation chamber is a cylindrical chamber with a volume of $\sim 150 \text{ cm}^3$. After the stagnation chamber flow is discharged into the atmosphere passing through a nozzle. The exit diameter of the nozzle is 12 mm. The contraction ratio of the jet nozzle is 4. Jet velocity is controlled by adjusting the pressure inside the stagnation chamber with a closed-loop algorithm in the jet flow unit system. The experimental setup can be seen in Figure 3.1.

Constant temperature anemometer (DANTEC 90C10) has been used for measurement of velocity and turbulence quantities inside the jet flow. Miniature wire probes of Dantec Dynamics having platinum-plated tungsten wire sensors with 5 μm diameter and 1.25 mm length, are used in experiments. Considering the dimensions of the flow area, a miniature straight hot-wire probe (55P11) with a high spatial resolution was selected for measurements. The hot wire probe is mounted to the probe support so that the prongs are parallel to the jet axis. Probe location is controlled by a 3-axis traverse system with $\pm 0.016 \text{ mm}$ uncertainty. Measurement locations are arranged before the test by defining a measurement grid. When tests initiate, the probe moves to the measurement location and waits 2 seconds before data acquisition to minimize flow disturbance caused by the movement. Data is acquired at a specified frequency and time interval, and the probe moves to the next location. The hot-wire signal was digitized using a 16-bit A/D Board with 16 analog input channels. Sampling rate was set at 8 kHz. The whole system is well grounded to avoid isolation problems. Throughout the experiments, the temperature ratio of the hot wire probe was kept constant at around 0.8 and therefore the corresponding wire temperature at 230 °C.

Calibration velocity set point was divided into 2 parts: 0-0.5 m/s; 0.5 m/s to 60 m/s. For calibration, 50 different velocity points are set. Calibration errors were not allowed to exceed the 1% band. After each successive set of experiments, the calibration was checked and repeated to ensure that it was not changed during the measurements. Repeatability of experiments is tested with repetitive experiments at different times. Uncertainty of mean velocity measurements based on the time series errors is $\pm 0.5\%$ with a 99% confidence level.

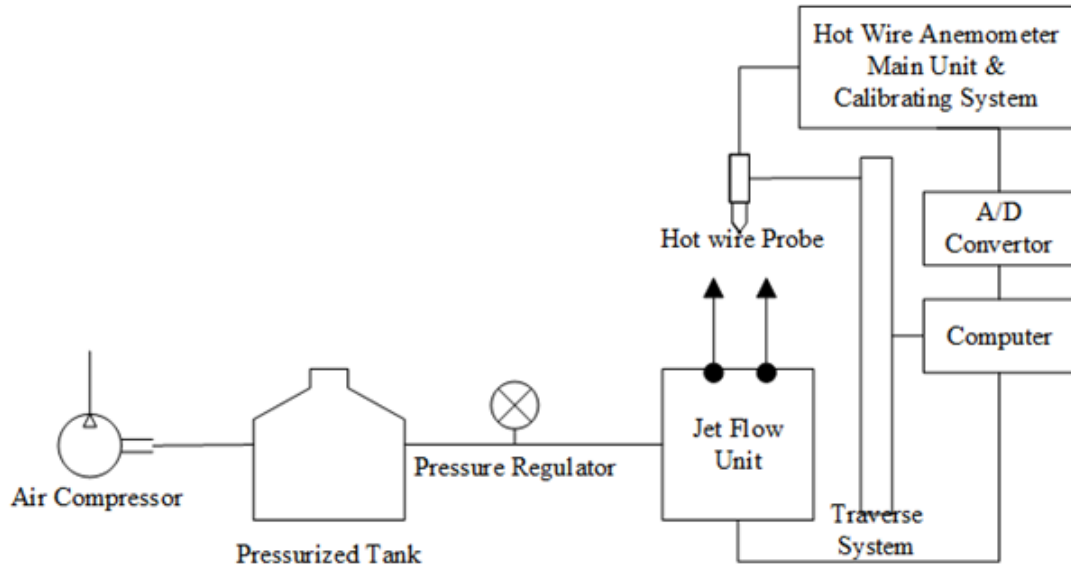


Figure 3.1 : Experimental setup.

3.2.1.1 Measurement locations

Velocity and turbulence profiles are measured by moving the probe in the radial direction at the selected 13 stations, where D is the jet diameter, and x is the distance from the jet exit, as given in Table 3.1. In addition, jet centerline velocities and turbulence intensities are measured along the jet centerline to examine the change of the speed and turbulence intensity on the jet axis with increasing distance from the jet exit plane.

Table 3.1 : Measurement locations for $Re=10000$.

Distance from jet exit [mm] (x- direction)	Measurement Station (x/D)	Axis [mm]	
		Range	Setup
6	0.5	$-8 < y < 8$	0.1
12	1	$-15 < y < 15$	0.1
24	2	$-20 < y < 20$	0.1
36	3	$-25 < y < 25$	0.1
48	4	$-30 < x < 30$	0.1
60	5	$-35 < y < 35$	0.1
72	6	$-40 < y < 40$	0.2
96	8	$-40 < y < 40$	0.2
120	10	$-45 < y < 45$	0.2
144	12	$-50 < y < 50$	0.25
180	15	$-60 < y < 60$	0.5
240	20	$-75 < y < 75$	0.5
300	25	$-90 < y < 90$	0.5

Through the data obtained, the centerline velocity decay and turbulence characteristics in the axial direction, jet half-width, and therefore, jet spreading rates were examined.

3.2.2 Numerical model

A numerical model was developed by aligning the boundary conditions, model properties, convergence criteria, and discretization scheme with the experimental results. On the computational domain, a grid was generated with a sufficient number of elements to obtain a precise CFD model without any extra numerical errors. The CFD model with the IP RSM turbulence model is briefly presented in this section.

3.2.2.1 Problem description

In the geometry created for the CFD analyses, the jet outlet diameter D was determined as 10 mm. Following the approach of Wawrzak and Tyliszczak [51], a cylindrical control volume was created with a distance of $50D$ from the jet outlet and a diameter of $30D$. This size ensures it can adequately capture the flow field without being influenced by boundary conditions, as indicated in their research. Figure 3.2 shows the computational domain for the CFD setup used in this study.

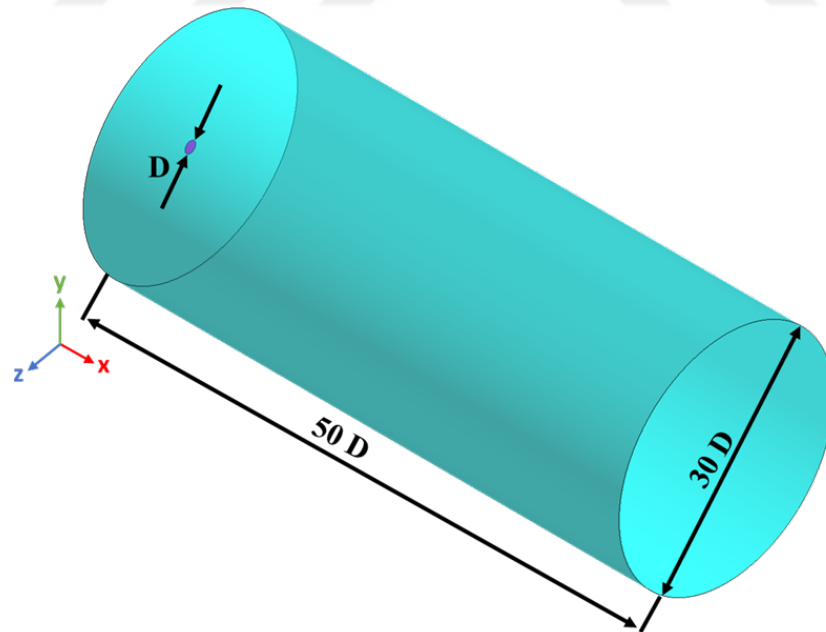


Figure 3.2 : Computational domain of round jet.

3.2.2.2 Boundary conditions

All analyses were conducted using Ansys® Workbench's parametric calculation methods. Given the relative simplicity of the problem, the initial conditions were set to 0 *m/s* for velocity and 1 *atm* for pressure, based on standard atmospheric conditions at 25°C. Air at 25°C was defined as the working fluid, with constant thermophysical properties selected from the solver database. Implementing proper inlet boundary conditions for free jet problems is crucial. The flow field properties, such as jet centerline velocity, potential core, and jet half-widths, are directly affected by the inlet boundary condition. Two types of boundary conditions are commonly used in round free jet problems. One is a velocity profile defined by a function where the velocity is zero at the walls and highest at the center, and the other is a uniform velocity profile. Faghani et al. [95] mentioned that a uniform inlet velocity profile shows better agreement with experimental data from similar studies on jet flow dynamics. They also demonstrated that using a uniform velocity profile with around 6% turbulence intensity produces more accurate results for contoured nozzles. They suggested that this approach could be useful for simulating various nozzle geometries (round, planar, rectangular, triangular, contoured) and other turbulence models. Therefore, the inlet velocities corresponding to different Reynolds numbers were applied as velocity-inlet boundary conditions at the inlet cross-sectional area (jet exit), specified as the uniform x-velocity component. The turbulence intensity at the inlet surface was set to 5% to align with the experimental data. All other surfaces were defined with a zero atmospheric gauge pressure opening-type boundary condition.

The velocity and turbulence intensity profiles obtained at the jet exit for $Re=10000$ using experimental methods are compared with CFD results in Figure 3.3. It can be seen that the boundary conditions applied in the CFD closely align with those from the experiment.

The convergence criteria for the analyses were set with Root-Mean-Square values of all 11 conservation equations at 10^{-6} . Additionally, the accuracy of convergence was verified by monitoring the mass balance at the inlet and outlet surfaces of the control volume and ensuring the stability of velocity values at seven points along the jet centerline. For the CFD analyses, the Ansys®-CFX software was used, employing the COUPLED solution algorithm. The high-resolution option was selected

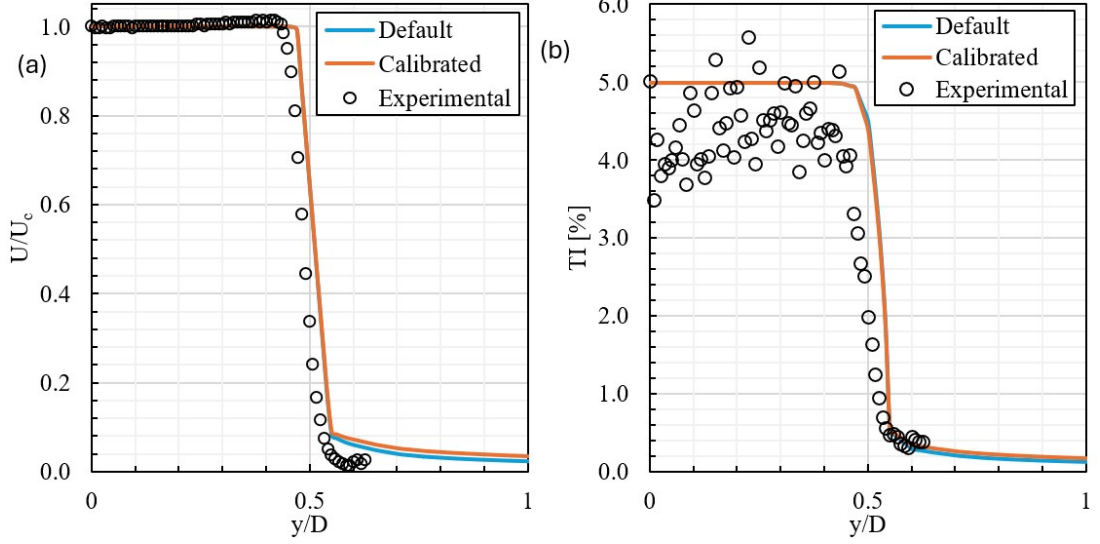


Figure 3.3 : Jet exit profiles for $Re=10000$ (a) velocity profiles (b) TI profiles.

for both the advection scheme and turbulence. The auto timescale option with a conservative setting was used to determine the timescale value required for the COUPLED algorithm, calculated as $4.86174 \times 10^{-3} s$.

3.2.2.3 Governing equations

The equations solved in this study are the continuity and momentum equations, along with the transport equations of Reynolds stresses. Considering the problem as three-dimensional, steady, incompressible, and neglecting body forces, the RANS equations can be written as follows

$$\frac{\partial \bar{v}_i}{\partial x_i} = 0 \quad (3.1)$$

$$\frac{\partial}{\partial x_j} (\rho \bar{v}_i \bar{v}_j) = -\frac{\partial p''}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) \right] - \frac{\partial}{\partial x_j} (\rho \bar{v}'_i \bar{v}'_j) \quad (3.2)$$

Here the modified pressure p'' , is equal to $\bar{p} + \left(\frac{2}{3}\right) \mu \left(\frac{\partial \bar{v}_k}{\partial x_k}\right)$. As seen in the momentum equation, the Reynolds stress term $\rho \bar{v}'_i \bar{v}'_j$ requires solving a transport equation for each of the six Reynolds stress components. The differential equation of Reynolds stress transport is given by

$$\bar{v}_k \frac{\partial}{\partial x_k} (\rho \bar{v}'_i \bar{v}'_j) = \frac{\partial}{\partial x_k} \left[\left(\mu + \frac{2}{3} C_s \frac{\rho k^2}{\varepsilon} \right) \frac{\partial \bar{v}'_i \bar{v}'_j}{\partial x_k} \right] + P_{ij} - \frac{2}{3} \delta_{ij} \rho \varepsilon + \Phi_{ij} \quad (3.3)$$

where Φ_{ij} is the pressure-strain correlation, and P_{ij} , the exact production term, is given by

$$P_{ij} = -\rho \bar{v}'_i \bar{v}'_k \frac{\partial \bar{v}_j}{\partial x_k} - \rho \bar{v}'_j \bar{v}'_k \frac{\partial \bar{v}_i}{\partial x_k} \quad (3.4)$$

In these equations, the anisotropic diffusion coefficients of the original models have been replaced by an isotropic formulation, which increases the robustness of the RSM. An equation for ε is still required due to its appearance in the individual stress equations. The anisotropic form of the ε equation used in this study is [52]

$$\frac{\partial}{\partial x_k} (\rho \overline{v_k} \varepsilon) = \frac{\varepsilon}{k} (C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon) + \frac{\partial}{\partial x_k} \left[\left(\mu \delta_{kl} + C_{\varepsilon} \frac{\rho k}{\varepsilon} \overline{v'_k v'_l} \right) \frac{\partial \varepsilon}{\partial x_l} \right] \quad (3.5)$$

where P_k is the turbulence production due to viscous forces, modeled for incompressible flows as

$$P_k = \mu_t \left(\frac{\partial \overline{v_i}}{\partial x_j} + \frac{\partial \overline{v_j}}{\partial x_i} \right) \frac{\partial \overline{v_i}}{\partial x_j} \quad (3.6)$$

where the turbulent viscosity μ_t is defined as

$$\mu_t = C_{\mu} \frac{\rho k^2}{\varepsilon} \quad (3.7)$$

Even though the RSM model does not use the eddy viscosity hypothesis, it still solves the turbulence production term in the dissipation equation due to its use of the dissipation equation, causing the RSM model to include the eddy viscosity hypothesis at some point.

The pressure-strain correlation, Φ_{ij} , is an important term in Reynolds stress models, driving turbulence towards isotropy by redistributing the Reynolds stresses. This term significantly contributes to the Reynolds stress equation and acts as a sink term for the shear stress equation. It is often referred to as the "Robin Hood" term because it "takes from the rich and gives to the poor," making the turbulence more isotropic by decreasing large normal stresses and the magnitude of shear stress while increasing small normal stresses [15].

$$\Phi_{ij} = \Phi_{ij,1} + \Phi_{ij,2} \quad (3.8)$$

It can be split into two parts: the "slow" part, $\Phi_{ij,1}$, also known as the return-to-isotropy term, and the "rapid" part, $\Phi_{ij,2}$. The slow and rapid parts are mathematically defined as [94]

$$\Phi_{ij,1} = -C_1 \frac{\rho \varepsilon}{k} \left(\overline{v'_i v'_j} - \frac{2}{3} \delta_{ij} k \right) \quad (3.9)$$

$$\Phi_{ij,2} = -C_2 \left(P_{ij} - \frac{2}{3} P \delta_{ij} \right) \quad (3.10)$$

where P is given by $0.5P_{ii}$ [52]. Thus, considering the experimental method and boundary conditions, the reduced conservation equations and IP RSM turbulence model equations are defined, introducing the closure coefficients C_S , $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, C_{ε} ,

C_μ , C_1 , and C_2 . The default values of these coefficients vary according to different researchers and CFD software, though the differences are not significant. In this study, Ansys®-CFX software is used as the CFD solver, and the default values of these coefficients in the solver are defined as 0.22, 1.45, 1.9, 0.18, 0.1152, 1.8, and 0.6 for C_S , $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, C_ε , C_μ , C_1 , and C_2 , respectively [52]. The calibration study has been conducted based on these coefficients.

3.2.2.4 Grid study

The computational grid for the designed geometry was constructed using the O-grid method in Ansys® Workbench Meshing. The Computer-Aided Design (CAD) geometry was segmented into nine sections, with each part interconnected to form the grid. To ensure mesh independence, 24 different grids with varying element counts from 5250 to 10.5×10^6 were generated. The analyses were conducted for a Reynolds number of 10000, utilizing the default model coefficients recommended for the IP RSM turbulence model.

In the mesh independence study, analyses were performed with the advection scheme set entirely to second-order upwind (Specified Blend Factor = 1.0 in CFX) to minimize the effects of numerical diffusion. When comparing the results, the changes in centerline velocities and jet half-widths, as shown in Figure 3.4, were considered. Additionally, decay constants and jet spreading rates were calculated, and relative errors were calculated by comparing the results to those obtained through experimental methods, as shown in Table 3.2. Finally, taking computational cost into account, the most suitable grid for use in the calibration process was selected. Figure 3.4 and Table 3.2 present only the six results closest to the experimental data, while the remaining 18 mesh results are not included due to high errors in some cases and convergence issues in others. Based on all these findings, the mesh independence study revealed that the most appropriate grid for this problem is the one with 810000 elements, named Mesh B.

The grid quality is crucial for accurate CFD analysis. The created mesh consists solely of structural elements, ensuring high-quality values. The maximum skewness value reaches 0.5, the minimum orthogonal quality is 0.74, and the maximum aspect ratio is 51. The mesh configuration is illustrated in Figure 3.5.

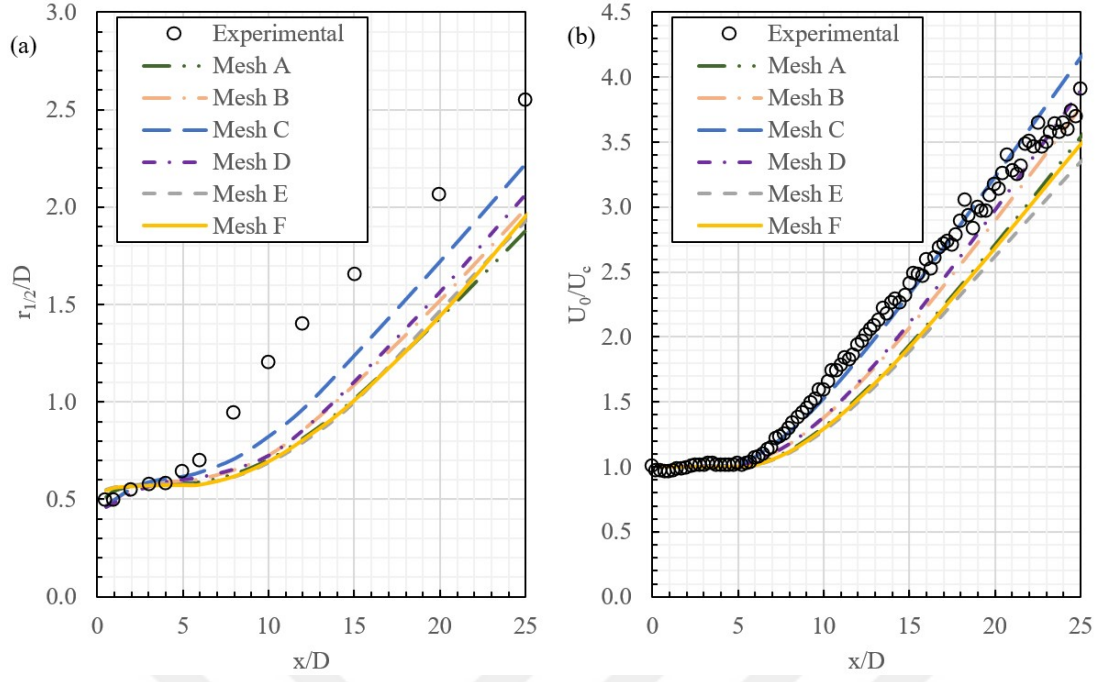


Figure 3.4 : Mesh independence results (a) for jet half-widths (b) for jet centerline velocity distributions.

Table 3.2 : Relative errors of the decay constant (B) and spreading rate (S) for different grids.

Name	B	S	Error B [%]	Error S [%]	Cell Count ($\times 10^3$)
Experiment	6.46	0.091	0	0	0
Mesh A	7.08	0.071	9.6	-21.6	500
Mesh B	6.61	0.076	2.3	-16.1	810
Mesh C	5.97	0.086	-7.5	-5.9	1040
Mesh D	6.34	0.080	-1.9	-12.0	1250
Mesh E	7.79	0.074	20.6	-19.1	1365
Mesh F	7.31	0.077	13.2	-15.0	5460

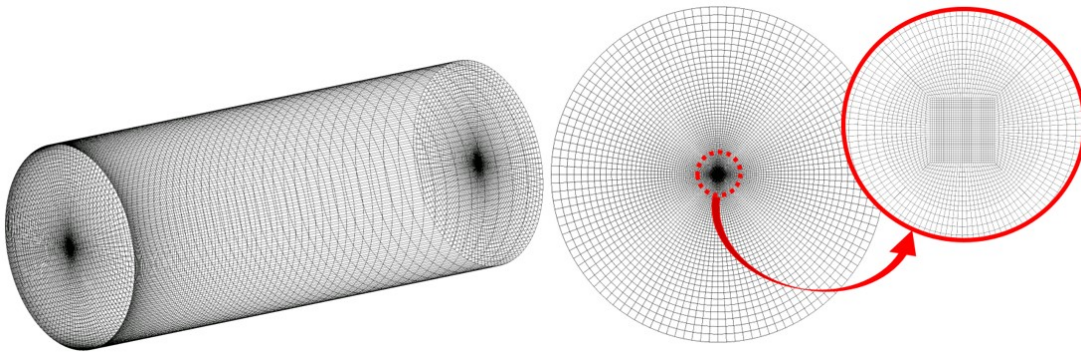


Figure 3.5 : Computational grid structure.

3.2.2.5 Optimization process

The aim of the optimization process was to fine-tune the closure coefficients of the IP RSM turbulence model to accurately predict the jet half-width values obtained from experimental methods of this study with a Reynolds number of 10,000. The coefficients to be optimized were C_S , $C_{\epsilon 1}$, $C_{\epsilon 2}$, C_{ϵ} , C_{μ} , C_1 , and C_2 . These coefficients were adjusted within a range of $\pm 25\%$ from their default values to ensure comprehensive coverage of potential variations. Using the default values of the IP RSM turbulence model's coefficients at $Re = 10000$, the CFD analysis did not produce jet half-width values that aligned with the experimental results. By finding the optimal combination of these coefficients, the CFD analysis results for jet half-width would match the experimental values. Since the default model coefficients did not provide accurate jet half-width predictions, it is necessary to calibrate closure coefficients to get an optimum coefficient combination for the RSM turbulence model for the round free jet problem through optimization.

The experimental jet half-width values were derived from experiments at 13 different stations along the jet axis and were used as target goal seeks for the optimization, with a tolerance of $\pm 10\%$ on account of CFD errors and experimental uncertainties. The stations were located at $x/D = 0.5, 1, 2, 3, 4, 5, 6, 8, 10, 12, 15, 20$, and 25 . Accurate prediction of jet half-width is crucial and ensures that other critical outputs, such as the velocity and turbulent intensity profiles along the jet axis, are also accurately predicted. The jet half-width is defined as the distance on the y-axis from the centerline at any point x along the jet axis where the velocity is equal to half of the centerline velocity at that point, and it is calculated as shown in equation 3.11.

$$r_{\frac{1}{2}}(x) = y|_{U(y)=\frac{U_c}{2}} \quad (3.11)$$

The optimization utilized the Adaptive Multi-objective Optimization (AMO) method in Ansys® DesignXplorer (ADE). AMO combines Kriging and Multi-objective Genetic Algorithm to explore the design space efficiently. Kriging is a statistical modeling technique that creates a surrogate model of the objective functions based on a limited number of sample points. This surrogate model is used to predict the objective function values for new design points, reducing the need for exhaustive CFD simulations. Kriging essentially provides a way to interpolate the results of a small number of simulations to predict the outcomes of many other possible configurations.

Since the problem is three-dimensional and has a large enough number of finite volume cells, this is crucial for saving computational resources and time [96]. MOGA, on the other hand, is an evolutionary algorithm that optimizes multiple conflicting objectives simultaneously. Genetic algorithms mimic the process of natural selection, where the fittest individuals are selected for reproduction in order to produce offspring of the next generation. MOGA works with a population of potential solutions, evaluating each one based on how well it meets the multiple objectives. The algorithm uses selection, crossover, and mutation to evolve the population over successive generations. Selection chooses the best solutions to pass their genes to the next generation. Crossover mixes the genes of parent solutions to create offspring, while mutation introduces random changes to individual genes to maintain genetic diversity. The goal is to evolve the population towards the Pareto front, a set of optimal solutions where no single solution can be considered better than another without degrading at least one objective. The Pareto front represents the best trade-offs between conflicting objectives [96]. The optimization workflow diagram is illustrated in Figure 3.6. Note that the AMO method may not be the most cutting-edge technique or fastest, but it is selected to show the viability of the approximation when conducted to the turbulence model closure coefficients calibration.

The optimization process started with the creation of an initial sample set using the Optimal Space-filling method, which ensures that the initial sample points are well-distributed across the design space, preventing clustering and ensuring a good coverage of possible solutions [96]. Specifically, 100 initial samples were generated to provide a broad overview of the design space. The number of samples per iteration, set at 50, balanced the need for new information with computational efficiency, with a maximum number of 50 iterations.

3.3 Results

3.3.1 Sensitivity analysis

In the sensitivity analysis studies, the sensitivity indices of the IP RSM turbulence model coefficients were calculated using ADE. The software performs sensitivity analysis during the optimization process by using response surfaces created during the optimization to construct an approximate simulation model. These surfaces provide a simplified representation of the complex relationships between inputs and outputs.

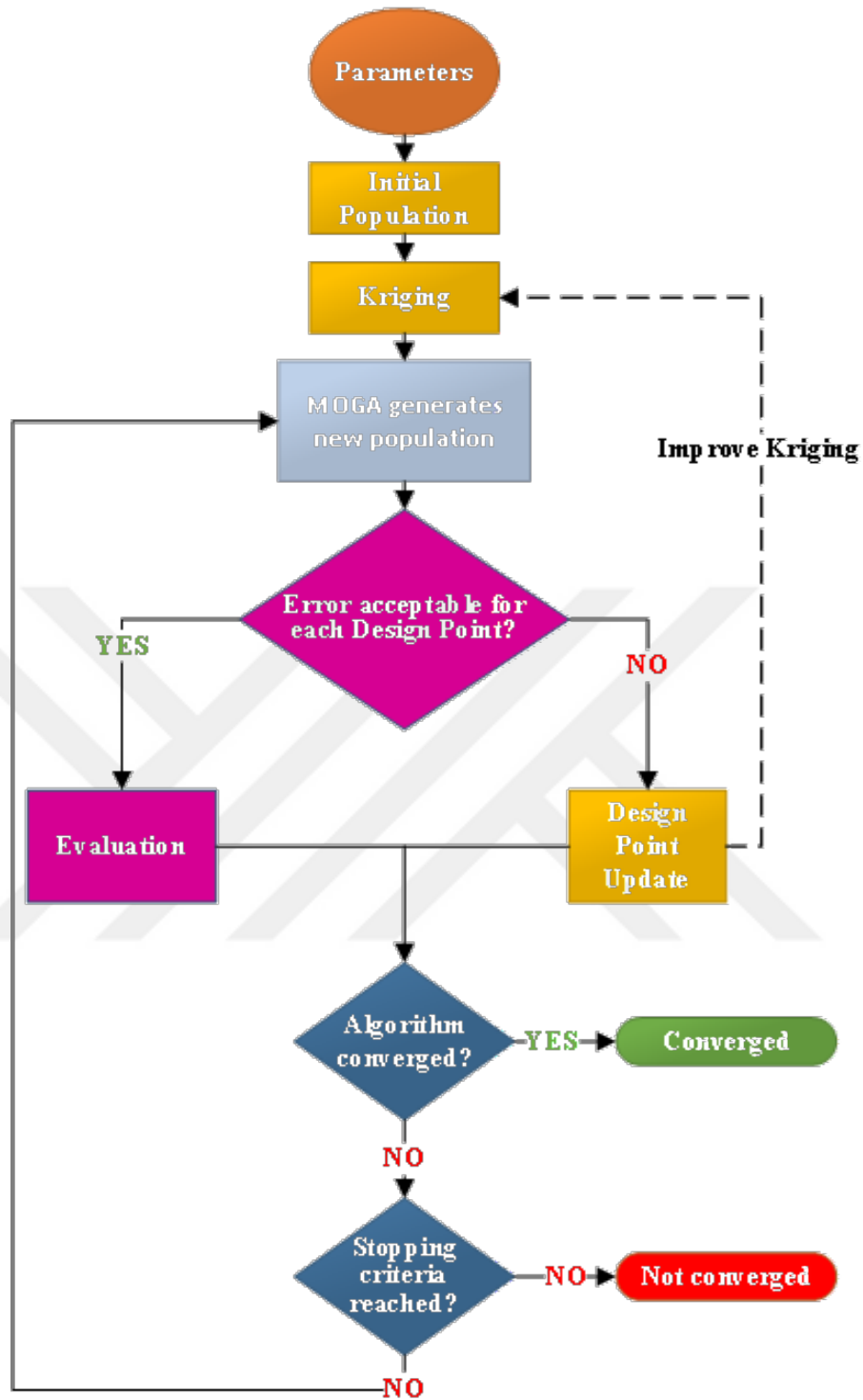


Figure 3.6 : Optimization diagram.

It calculates the relative importance and effect of each independent variable on the dependent variable using variance-based methods, including both main and interaction effects [96]. Additionally, it allows regression analysis to examine the dependency of a dependent variable on an independent variable. In this study, sensitivity analysis calculations were performed using the 795 CFD analysis runs, and the results were

compared. Seven different IP RSM turbulence model coefficients (C_S , $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, C_μ , C_1 , and C_2) were used as input parameters, and jet half-width values obtained from 13 different stations (r_1 , r_2 , r_3 , r_4 , r_5 , r_6 , r_7 , r_8 , r_9 , r_{10} , r_{11} , r_{12} , and r_{13}) were evaluated as model outputs. Table 3.3 presents the sensitivity percentages obtained from ADE by examining the results of 795 CFD analyses.

Table 3.3 : Sensitivities of the closure coefficients.

No	$C_{\varepsilon 1}$ [%]	$C_{\varepsilon 2}$ [%]	C_μ [%]	C_S [%]	C_1 [%]	C_2 [%]	C_ε [%]
1	65.5	27.3	24.7	31.7	6.8	26.2	26.4
2	63.5	27.1	18.0	11.0	10.4	22.1	1.1
3	11.7	53.0	4.2	40.8	51.0	6.5	37.5
4	44.5	35.6	27.0	27.8	23.0	19.2	67.8
5	73.1	45.0	26.4	44.7	28.3	28.0	10.5
6	51.7	24.0	35.2	42.4	6.9	19.1	40.9
7	40.9	14.2	39.9	44.3	0.3	15.8	51.2
8	39.6	14.7	39.6	43.9	0.1	15.8	51.1
9	41.3	15.9	38.9	43.9	1.3	15.9	49.8
10	50.3	24.2	35.6	42.2	7.2	18.1	41.2
11	53.5	26.6	35.0	43.3	7.6	19.8	36.9
12	66.0	40.3	27.2	46.2	22.3	25.6	16.4
13	69.8	43.0	27.4	47.7	25.6	26.5	13.8
Total	671.26	390.82	379.20	509.76	190.68	258.38	444.53

To enable a comparison, the absolute values of the local sensitivity percentages of each input for each output were summed. The reason for taking the absolute values is that ADE provides sensitivity indices as positive or negative. A positive sensitivity index indicates that increasing the input value increases the output value, while a negative sensitivity index indicates that increasing the input value decreases the output value. Therefore, the absolute values of the sensitivity indices were considered in the summation. For example, the local sensitivity values of the $C_{\varepsilon 1}$ coefficient, calculated separately for 13 different stations, were summed by taking their absolute values, obtaining the total effect of this coefficient across all stations.

Additionally, it should be noted that after the first 100 analyses, Kriging and Genetic Algorithm are applied to determine 50 new coefficient combinations in each optimization iteration. This compresses the coefficient combinations created for new analyses into appropriate ranges to bring the results closer to the values defined as the objective function. Figure 3.7 shows the variation graph of the $C_{\varepsilon 1}$ coefficient over 1050 different investigations. The reason for having 1050 investigations but only 795

CFD analysis results is due to factors such as convergence issues, infeasible points, the use of surrogate models, and the filtering of redundant points.

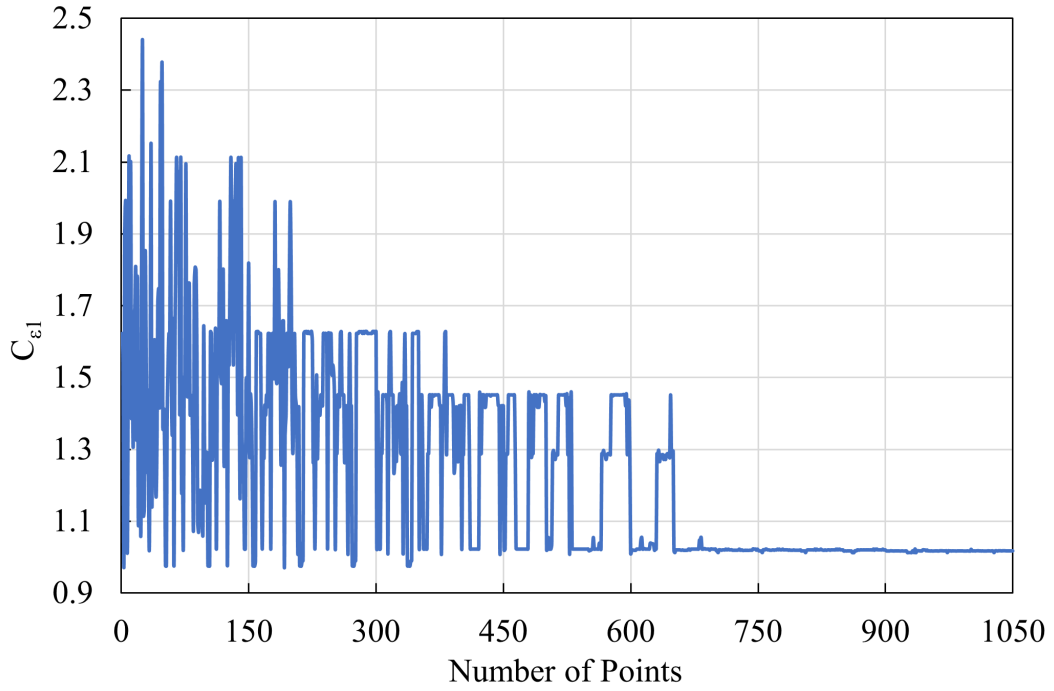


Figure 3.7 : Variation of $C_{\epsilon 1}$ constant at 1050 different examinations.

According to the ranking based on the summation of the absolute values of the sensitivity analysis results calculated for each station for each model coefficient, the model coefficient with the highest total effect is $C_{\epsilon 1}$. This indicates that, considering all stations (all output values), the most influential input (turbulence model coefficient) is the $C_{\epsilon 1}$ coefficient. Similarly, the other model coefficients affecting all stations are ranked as C_S , C_ϵ , $C_{\epsilon 2}$, C_μ , C_2 , and C_1 , respectively.

3.3.2 Calibration of closure coefficients

3.3.2.1 Results of calibration study

The coefficient calibration process was carried out to obtain the optimal coefficient combination that aligns the jet half-widths calculated using the IP RSM turbulence model with those obtained through experimental methods. The optimization process ran for a total of 21 iterations, resulting in 795 CFD analyses. The iterative process involved generating new sample points, running CFD simulations, and updating the surrogate model. The optimization converged when the Pareto front showed stability at 2%, and the convergence stability percentage was 0.26453%. The change in the convergence stability percentage calculated at each iteration during the optimization is

shown in Figure 3.8.

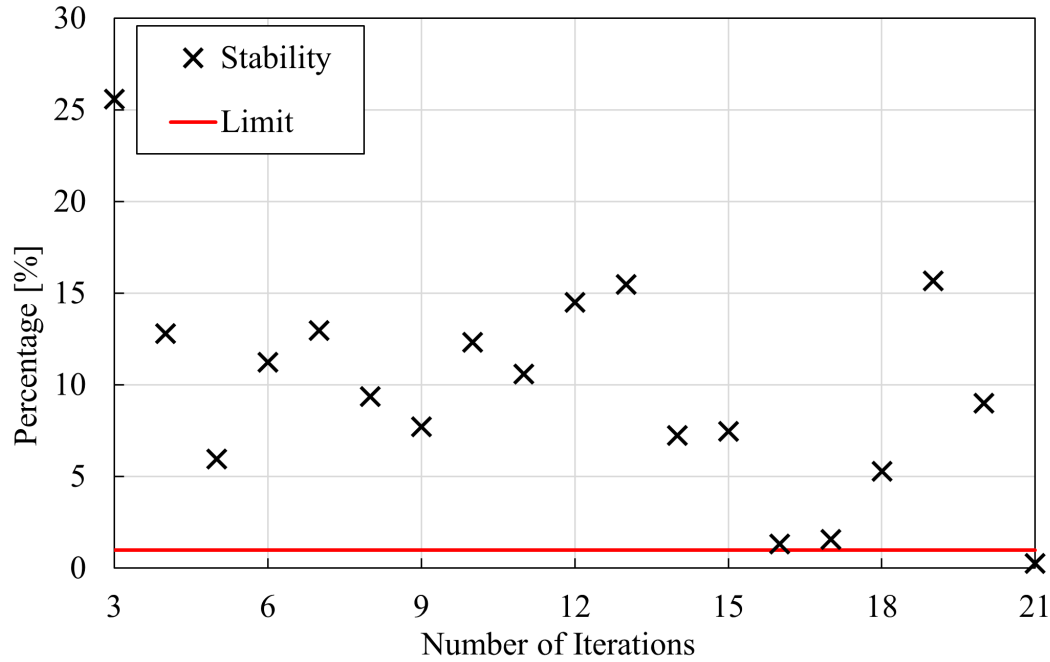


Figure 3.8 : Convergence stability percentage of optimization.

The values obtained for the coefficients C_S , $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, C_ε , C_μ , C_1 , and C_2 as a result of the optimization were 0.17032, 1.02142, 1.38099, 0.14123, 0.14997, 2.14352, and 0.46696, respectively. Using these values in the CFD analysis, the jet half-widths obtained along the jet center axis were compared with the jet half-width results obtained through experimental methods.

The turbulence model coefficients influence the turbulence model equations, and the turbulence model, in turn, affects the momentum equation. As a result, changes in the model coefficients alter the flow properties results obtained for the entire flow field. Therefore, it is insufficient to examine only the changes in jet half-widths. Parameters important for round free jets include the change in centerline velocity along the jet axis, changes in velocity profiles, and how turbulence intensity and turbulence intensity profiles change. Velocity profiles changing along the jet axis directly affect the calculations of jet half-widths.

In Figure 3.9, the jet half-widths obtained for a Reynolds number of 10000 using the default and fine-tuned coefficient combinations in CFD analyses are compared with the results obtained through experimental methods and benchmark studies conducted by Panchapakesan and Lumley [97], and Mi et al. [65]. Note that benchmark studies Reynolds number were different than 10000 and they are shared in Table 3.4. It

has been observed that after calibration, the jet half-width calculations using the IP RSM turbulence model match the experimental data much better compared to the calculations using the default coefficients. This indicates that a successful optimization has been carried out.

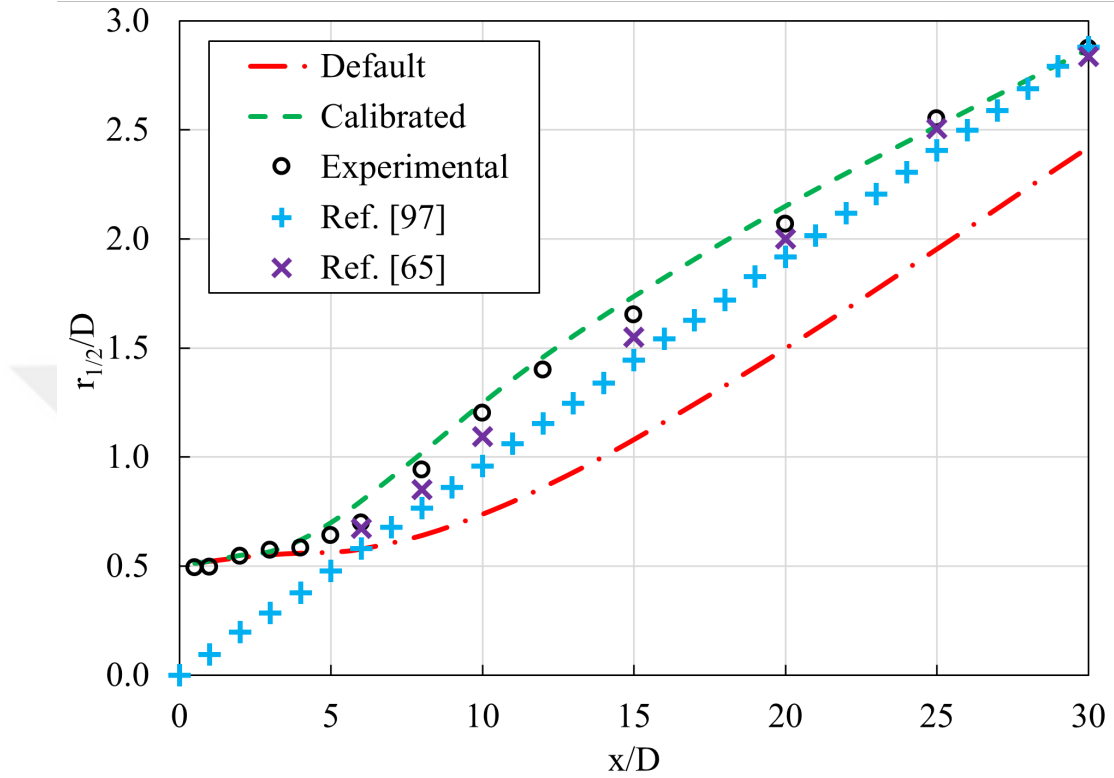


Figure 3.9 : Jet half-width results along the jet centerline for $Re=10000$.

Velocity profiles calculated with both the fine-tuned coefficients and the default coefficients have been compared with the velocity profiles obtained through experimental methods. To examine the flow field calculated with the coefficients obtained after the optimization in more detail, the velocity profiles calculated at selected stations along the jet axis are provided in Figure 3.10. In the initial region (potential core region, $x/D < 5$), the results obtained with the default coefficients are as satisfactory as those with the calibrated coefficients. However, in the developing (transition) region ($x/D > 5$), the results obtained with the calibrated coefficient combination performed much better than the default coefficients, calculating velocity profiles very close to those obtained through experimental methods. Looking at the jet half-widths graph in Figure 3.9, it can be said that the jet initial region development is almost complete, and the potential core is closed at $x/D \approx 5$. The developing region starts downstream of this position. Additionally, this also shows that the

boundary condition applied for the jet exit in the CFD analyses is consistent with the experimental method. If a fully developed flow profile had been applied at the jet exit, the jet half-widths would have started to increase from the jet exit, and this would have disrupted the consistency between the CFD analyses and experimental results. The agreement of the velocity profiles obtained from the fully developed region of the jet flow with the experimental methods once again demonstrates the success of the optimization process.

The fact that the velocity profiles from computational analysis align well with experimental data is not a surprising result. The optimization process was conducted based on jet half-width calculations as mentioned before. As can be understood from equation 3.11, the jet half-width is directly related to the velocity distribution along the y-axis. Therefore, where the jet half-widths are consistent, it is also expected that the velocity profiles along the y-axis will be consistent.

In experimental measurements, both mean velocities and fluctuating velocity values can be measured, allowing for the calculation of turbulence intensity ($TI = (u'_{RMS}/U_c)$). However, in RANS turbulence models, fluctuating velocities are not directly calculated. If we assume that the flow is isotropic with respect to Reynolds stresses ($\overline{v'_1 v'_1} = \overline{v'_2 v'_2} = \overline{v'_3 v'_3}$) and the shear stress components are equal to zero ($\overline{v'_1 v'_2} = \overline{v'_1 v'_3} = \overline{v'_2 v'_3} = 0$), we can obtain the turbulence kinetic energy as shown in equation 3.12.

$$k = \frac{1}{2} (\overline{v'_1 v'_1} + \overline{v'_2 v'_2} + \overline{v'_3 v'_3}) = \frac{3}{2} (\overline{v'_1 v'_1}) \quad (3.12)$$

Thus, using equation 3.12, the RMS of the fluctuating velocity can be calculated. The turbulence intensities calculated using this method and the TI values calculated by taking the RMS of the experimentally measured fluctuating velocity values are plotted along the y-axis at selected stations and compared in Figure 3.12. Comparing the turbulence intensity profiles obtained in the region $x/D < 15$, the results obtained with the adjusted coefficient combination are clearly better than those obtained with the default coefficients. In the region $x/D > 15$, the turbulence kinetic energy calculated at the jet center using the adjusted coefficients is lower than the turbulence kinetic energy calculated using the default coefficients. Consequently, the TI values are also obtained to be lower. It can be seen that when examining the TI distributions along the jet axis in Figure 3.11, it is observed that the TI values obtained with the adjusted

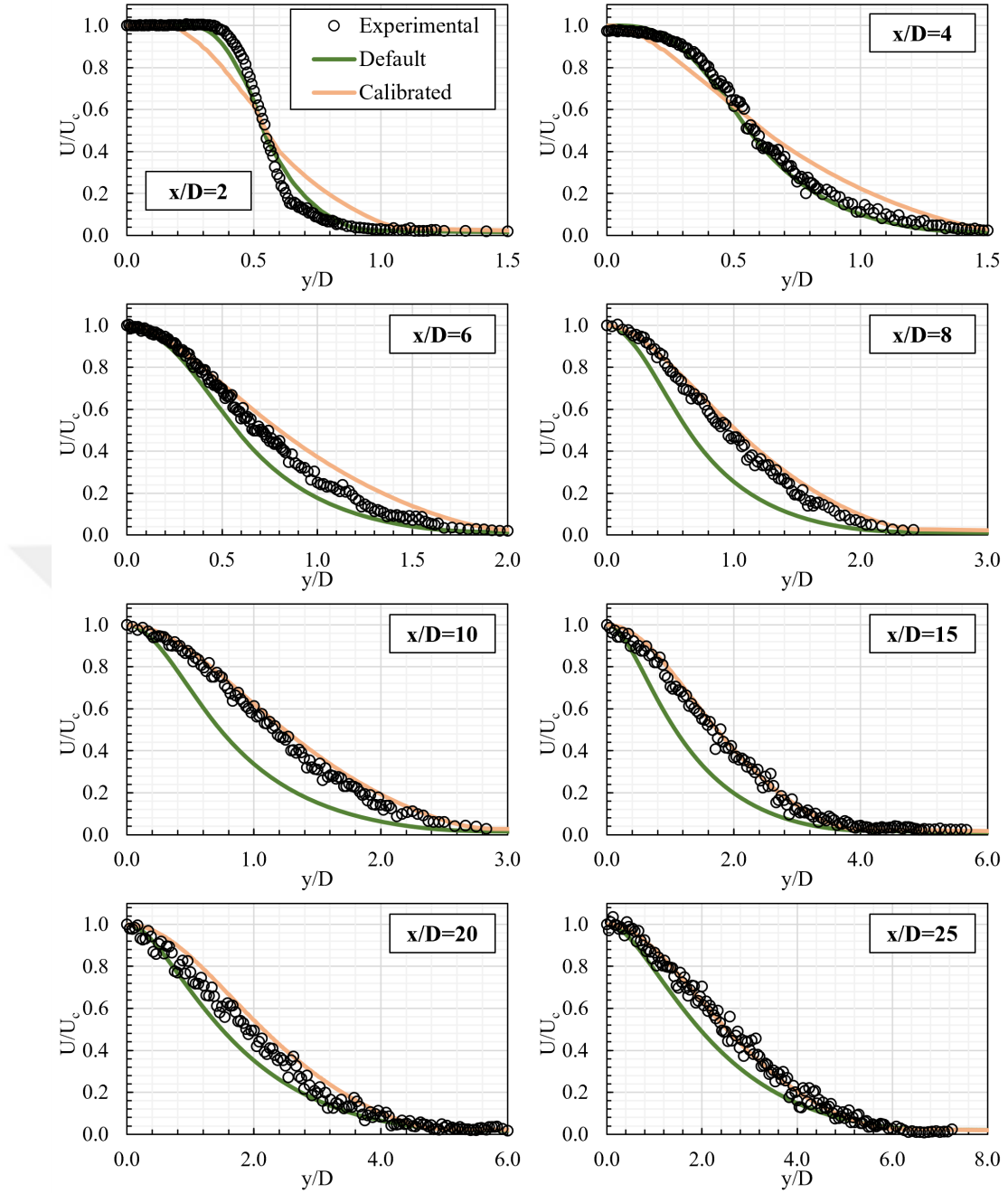


Figure 3.10 : Velocity profiles along jet axis for $Re=10000$.

coefficients deviate more from the experimental data in the region $x/D > 15$ along the jet center axis compared to the TI values obtained with the default coefficients.

The reason for the failure of the TI calculations along the jet centerline with calibrated coefficients in the region of $x/D > 15$, compared to the default coefficients, lies in the influence of these coefficients on the turbulence kinetic energy calculations. Similarly, this issue is observed in the comparison of TI profiles along the jet centerline. While the TI results obtained with the calibrated coefficient combination are more successful in the region close to the jet exit, the TI values calculated with the default coefficient

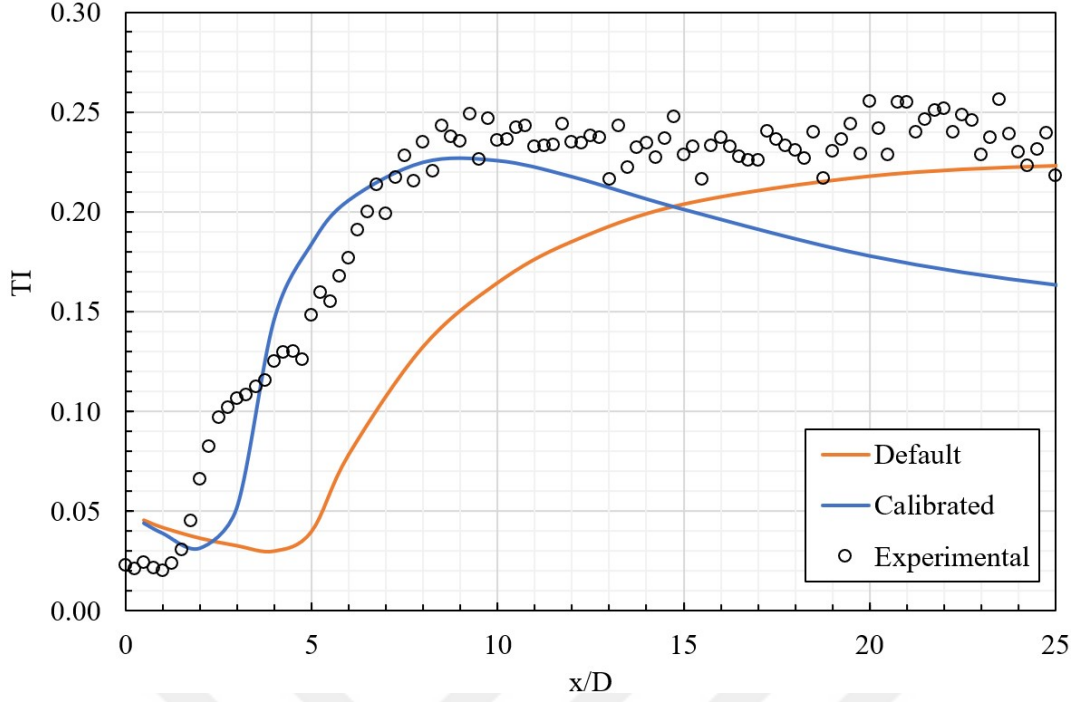


Figure 3.11 : TI along the jet centerline for Re=10000.

combination show better agreement with the experimental data in the $x/D > 15$ region.

Free jet flows exhibit self-similar behavior. In a turbulent free jet flow problem, the change in velocity along the jet centerline is referred to as the decay constant " B " and is calculated using the formula in equation 3.13. Pope [74] has shown that the formula in equation 3.13 is obtained from the curve of the jet center velocities along the jet axis.

$$\frac{1}{B} = \frac{\left(\frac{U_0}{U_{c(x)}} \right)}{\left(\frac{x}{D} - \frac{x_0}{D} \right)} \quad (3.13)$$

Here, U_0 is the jet exit velocity, $U_{c(x)}$ is the local velocity calculated along the jet centerline, and x_0 is the position defined as the virtual jet origin, obtained from the intersection of the curve in Figure 3.13 with the x-axis. Another critical parameter for free jet flows is the spreading rate " S ". In equation 3.14, the spreading rate formula, as defined by Pope [74], is given as the slope of the curve of jet half-widths along the jet centerline.

$$S = \frac{\left(\frac{r_{\frac{1}{2}}(x)}{D} \right)}{\left(\frac{x}{D} - \frac{x_0}{D} \right)} \quad (3.14)$$

where $r_{\frac{1}{2}}(x)$ is the jet half-width. Accurately calculating these parameters demonstrates that a turbulence model can reliably simulate real-world jet behavior, which is crucial in the design and optimization of various engineering systems such as propulsion systems, combustion problems, and industrial chimneys [98].

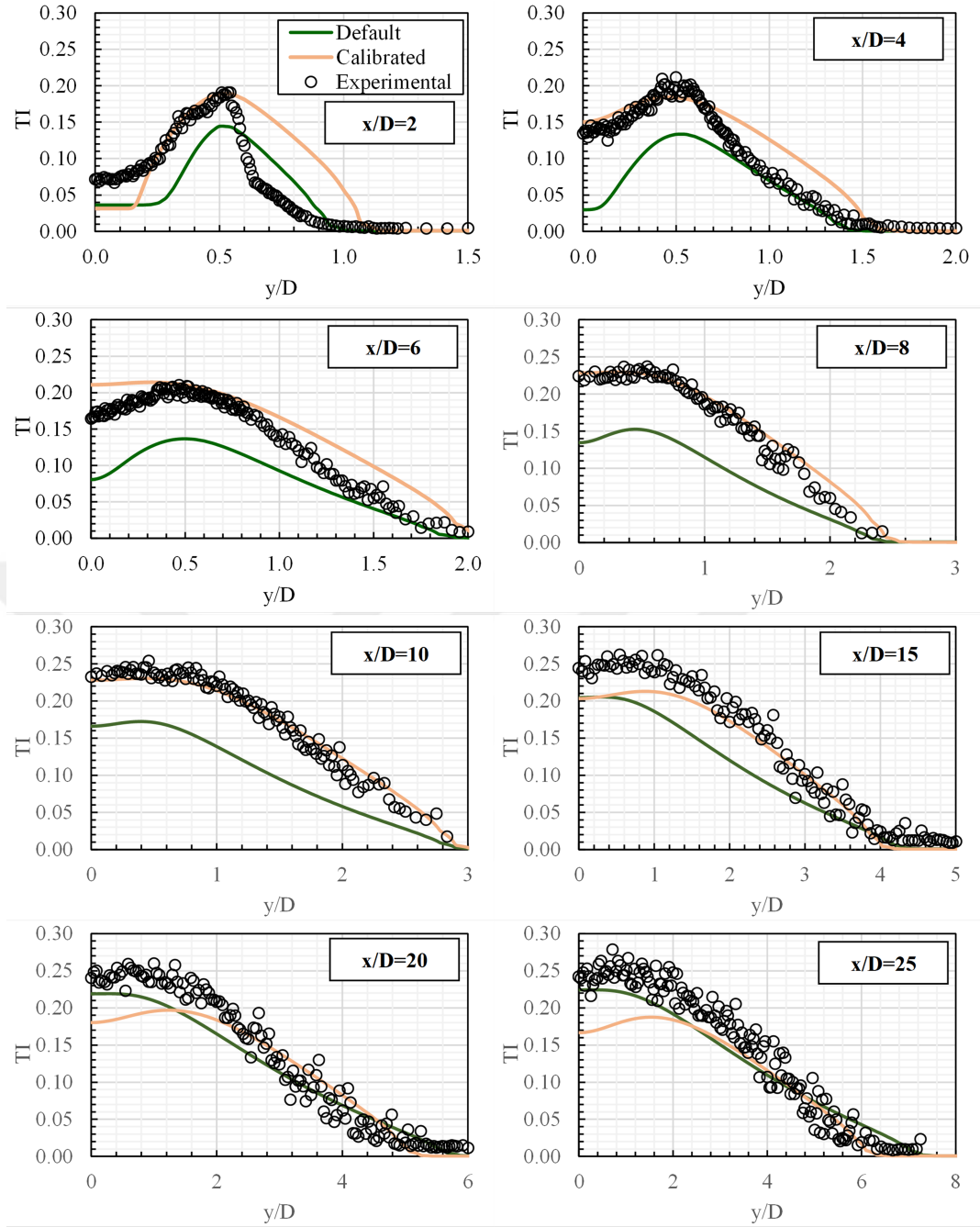


Figure 3.12 : TI profiles along jet axis for $Re=10000$.

In Figure 3.13, the graph of the change in jet center velocities for $Re = 10000$ shows a comparison between the results obtained through experimental methods, CFD results, and benchmark studies. Neither coefficient combination shows a successful match with the experimental results. To make the comparison more meaningful, decay constant and spreading rate values were calculated and shared in Table 3.4 along with results from benchmark studies.

When comparing the decay constant and spreading rate values calculated in this study,

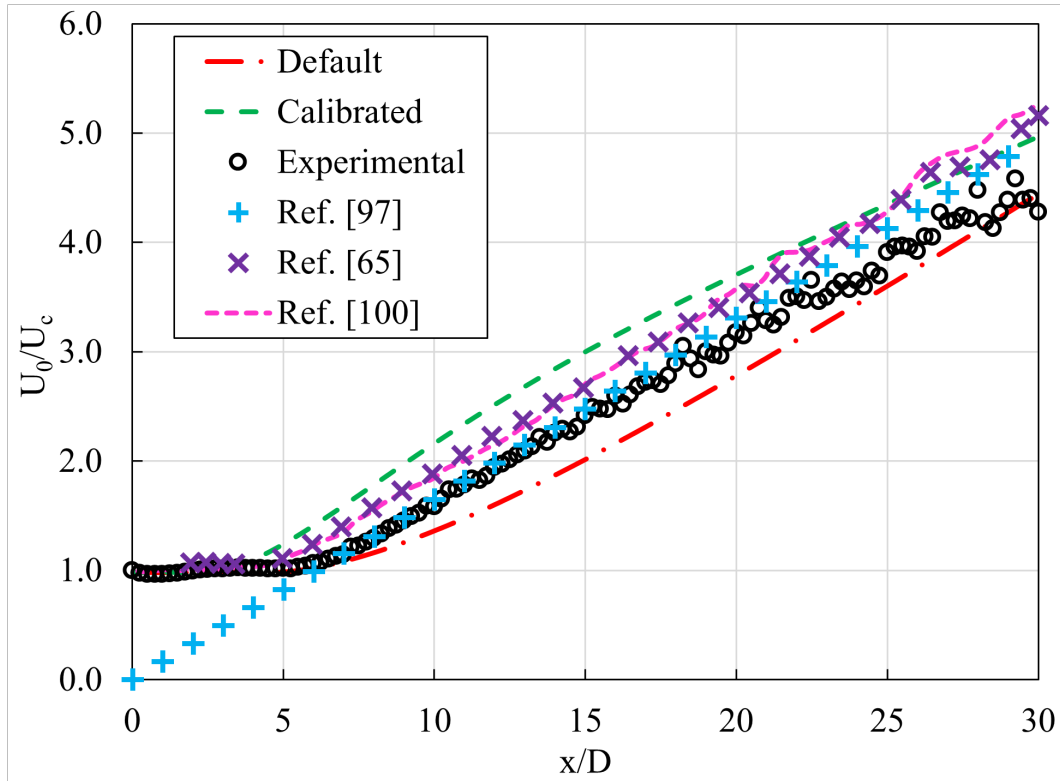


Figure 3.13 : Jet centerline velocity distribution for $Re=10000$.

the error margin for the decay constant was reduced from approximately 2% to 1%, and the error margin for the spreading rate was reduced from approximately 16% to 5.2% thanks to the calibration study. These results indicate that the uncertainty of the IP RSM turbulence model has been reduced, and the model predictions have been improved.

Table 3.4 : Decay constant and spreading rate results.

Name of the Study	Method	Re	B	S
Ref. [97]	Experimental	11000	6.06	0.096
Ref. [99]	Numerical (LES)	11000	6.40	0.087
Ref. [65]	Experimental	10750	6.40	0.083
Ref. [100]	Numerical (LES)	11000	5.79	0.099
Current Study	Experimental	10000	6.46	0.091
Current Study	Numerical	10000	6.61	0.076
Default Coeff.	(RANS)			
Current Study	Numerical	10000	6.52	0.086
Calibrated Coeff.	(RANS)			

3.3.3 Budget of turbulence kinetic energy

To gain a more detailed understanding of the impact of coefficient optimization on the flow field, the turbulence budget of the RSM model has been calculated.

The turbulence kinetic energy budget was derived by taking the trace ($i = j$) and multiplying by $1/2$ in the RSM equation, as given in equation 3.3, which results in the turbulence kinetic energy equation. Taking the trace causes the pressure-correlation term to vanish [15]. The equation used for the TKE budget is presented below.

$$0 = \overline{v_k} \frac{\partial k}{\partial x_k} + \left(\overline{v} \frac{\partial^2 k}{\partial x_k^2} + \frac{\partial}{\partial x_k} \left(\frac{2}{3} C_s \frac{\rho k^2}{\epsilon} \frac{\partial k}{\partial x_k} \right) \right) + \frac{P_{ii}}{2} - \epsilon \quad (3.15)$$

The first term on the right-hand side of the equation represents advection, the second term represents diffusion, the third term represents production, and the fourth term represents dissipation. In the true IP RSM equation, the diffusion term is calculated using viscous, turbulent, and pressure diffusion. However, since fluctuating variables are not computed in the model equation, the pressure diffusion and turbulent diffusion terms are modeled. In the RSM model, the pressure diffusion term is not calculated directly; instead, viscous and turbulent diffusion terms are used.

To ensure meaningful comparison of the budget calculation, the results obtained from the CFD analyses in this study are compared with those obtained through experimental methods by Panchapakesan and Lumley [97] and through LES calculations by Balajee and Panchapakesan [100], as shown in Figure 3.14. Additionally, in the LES conducted by Balajee and Panchapakesan [100], both turbulent diffusion and pressure diffusion terms were calculated. The arithmetic average of the separately calculated turbulent diffusion and pressure diffusion terms $((TD + PD)/2)$ is also shown in Figure 3.14. All calculations are normalized using $U_c^3/r_{1/2}$, and viscous diffusion terms are not presented in the figure since they are found to be negligible. Since these studies were also examined at $Re = 11000$ over a wider flow field, the flow domain was extended to $x/D = 100$, the computational grid is also doubled in the x direction, and CFD analyses were re-run for $Re = 11000$ using both the default and calibrated coefficients of the IP RSM model. The TKE budget obtained in this study was also calculated in the far field region ($50 < x/D < 90$) to align with benchmark studies.

When examining Figure 3.14, it can be observed that the default coefficients perform better in the calculation of advection and turbulent diffusion compared to the calibrated coefficients. However, the fact that the diffusion term calculated with the adjusted coefficients aligns well with the average diffusion term calculated by Balajee and Panchapakesan [100] gives significance to the calibration study. The results obtained with the adjusted coefficient combination have improved the overall diffusion

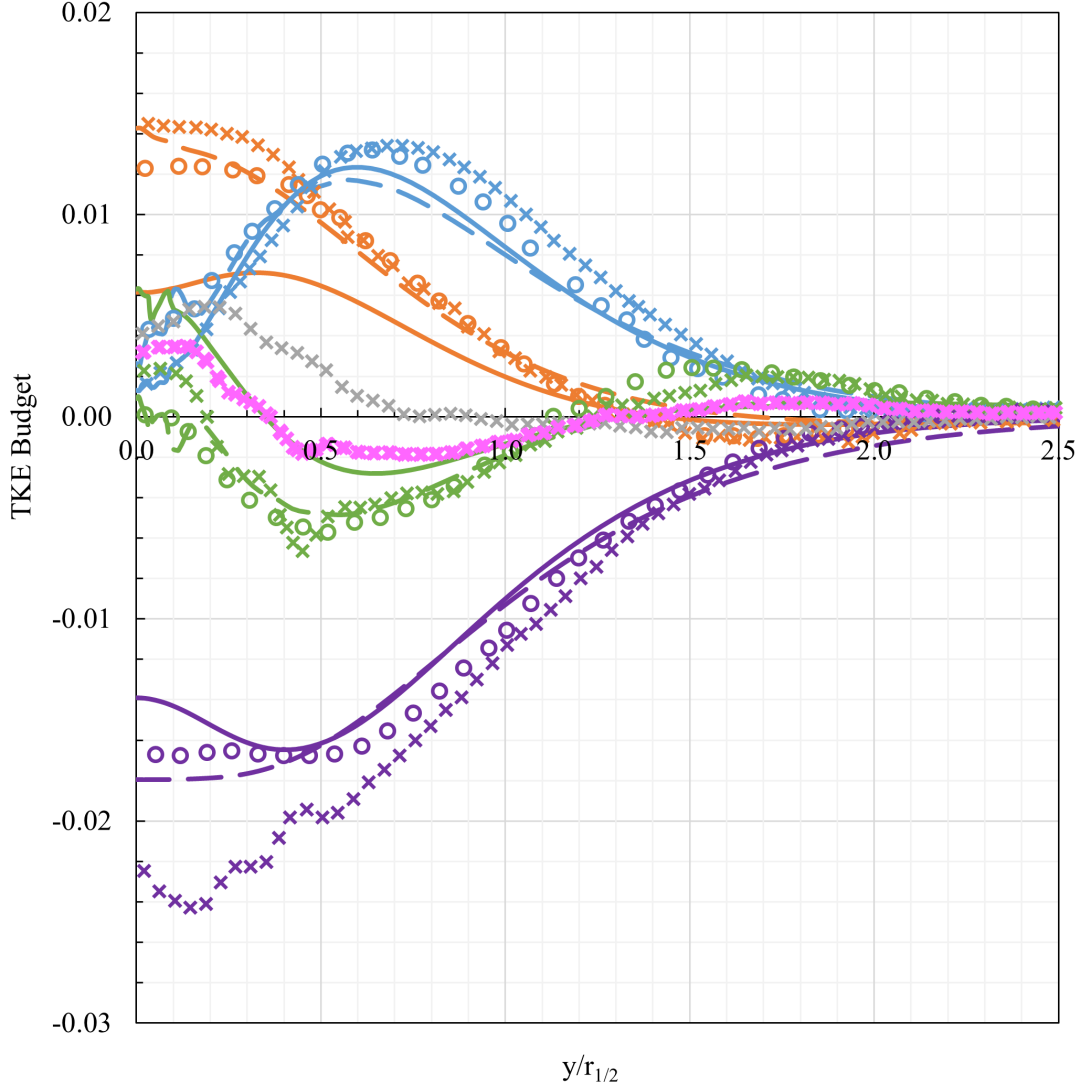


Figure 3.14 : TKE budget for $Re=11000$. Symbols: $\circ$, Panchapakesan and Lumley [97]; $\times$, Balajee and Panchapakesan [100], - - - (dashed lines), current numerical study with default closure coefficients; — (solid lines), current numerical study with calibrated closure coefficients. Colors: orange, advection; blue, production; green, diffusion; purple, dissipation; gray, pressure diffusion from Balajee and Panchapakesan [100]; pink, arithmetic average of pressure diffusion and turbulent diffusion calculated from Balajee and Panchapakesan [100].

modeling of the IP RSM turbulence model, albeit at the expense of the accuracy in the advection term.

3.3.4 Normalized dissipation rate and Kolmogorov length scale along jet centerline

In order to compare the calculations along the x-axis, the normalized dissipation rate ($\epsilon^* = \epsilon D / U_0^3$) and Kolmogorov length scale ($\eta / D = (\nu^3 / \epsilon)^{1/4} / D$) for $Re = 10750$, as determined in the experimental study by Mi et al. [65], were calculated using the

CFD results from this study and compared accordingly. As shown in the ε^* graph in Figure 3.15, it is evident that the dissipation rate calculations performed with the fine-tuned coefficients show better agreement with the experimental data than those obtained using the default coefficients. The Horizontal Line provided by Mi et al. [65] represents the independence of the dissipation rate values along the x-axis from the Reynolds number after the critical Reynolds number ($Re_{cr} \approx 10000$, according to Mi et al. [65]). These results not only provide insight into the validation of the TKE budget calculations but also reinforce the reliability of the dissipation rate calculations. As highlighted in the study by Mi et al. [65], after the critical Reynolds number is reached, the dimensionless results within the self-similar region remain consistent. Therefore, it was deemed necessary to conduct analyses for $Re = 20000$, assuming that the fine-tuned coefficients would also yield more accurate jet half-width calculations across different Reynolds numbers compared to the default coefficients.

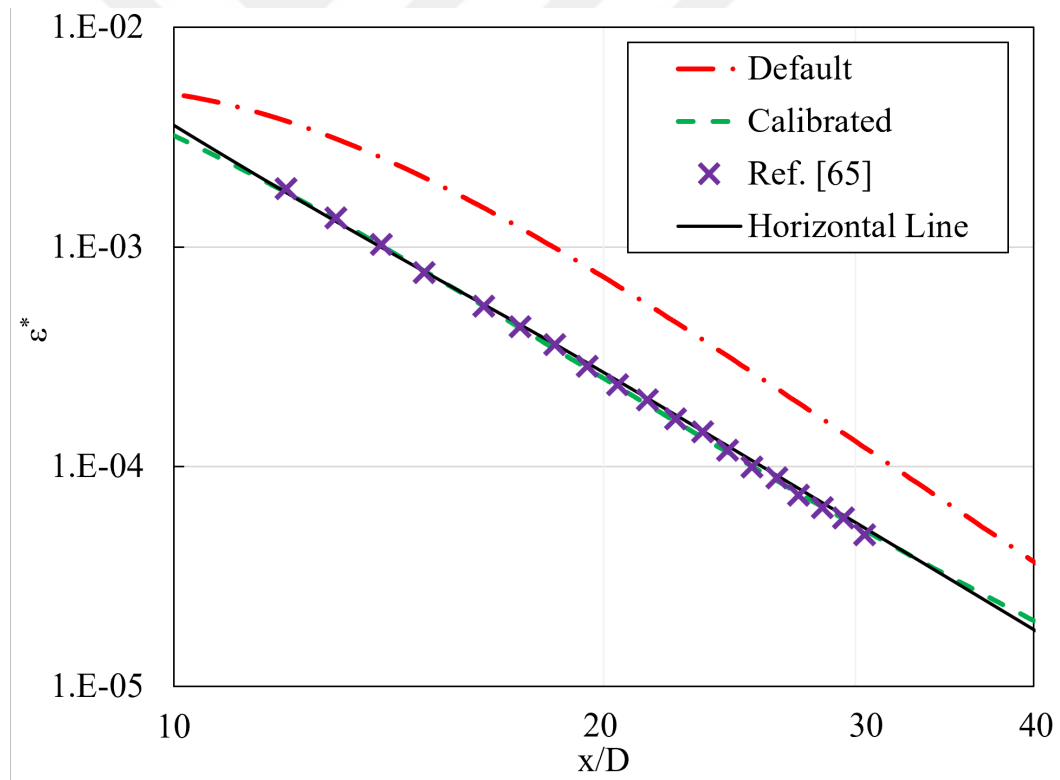


Figure 3.15 : Normalized dissipation rate along the jet centerline.

Mi et al. [65] also provided Kolmogorov length scale values along the jet centerline, demonstrating the independence of this value from the jet exit conditions. To further evaluate the success of the calibrated coefficient combination, Kolmogorov length scale values were calculated along the same axis and presented in Figure 3.16.

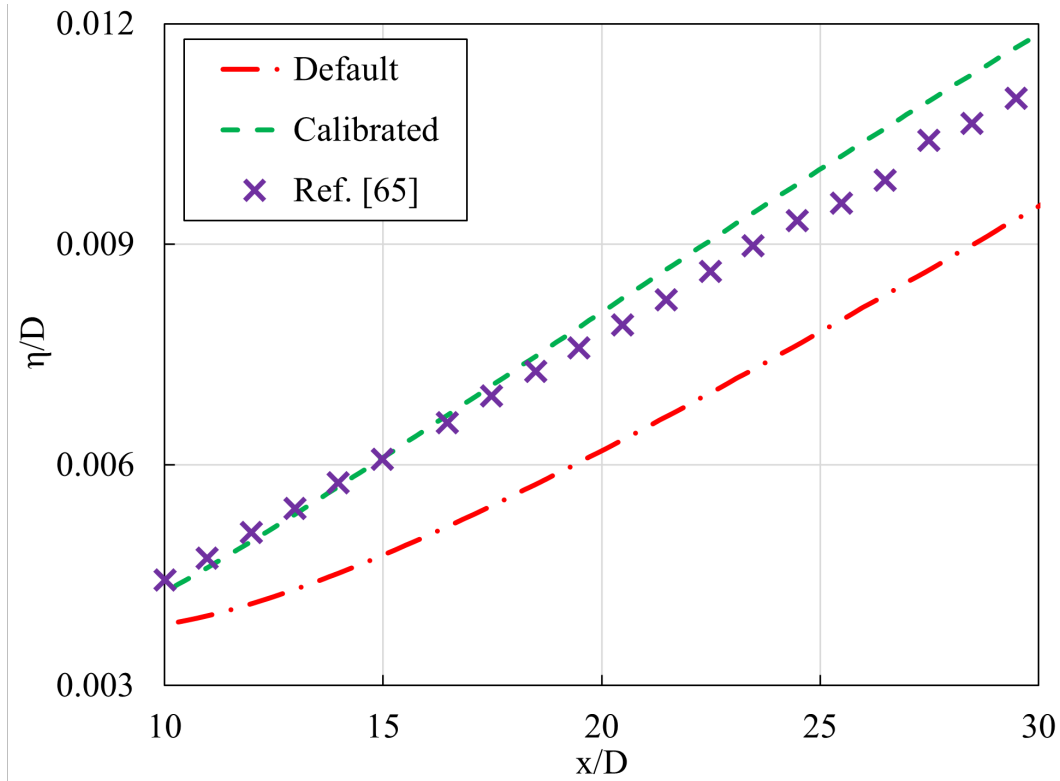


Figure 3.16 : Kolmogorov length scale results along the jet centerline.

These results further demonstrate that calculations performed with arranged coefficients yield more accurate outcomes compared to those using default coefficients. Despite the fact that the IP RSM RANS model does not directly calculate or model at the Kolmogorov length scale, the agreement between the Kolmogorov length scale values computed along the jet centerline and those experimental results indicates that model predictions can be significantly improved with the correct combination of coefficients. Encouraged by these findings, new CFD analyses were conducted for $Re = 20000$ using the arranged coefficients. Additionally, analyses with the default coefficients were carried out at the same Reynolds number, and experimental work was repeated for comparison at this Reynolds number.

3.3.5 Results for $Re=20000$

In studies conducted on round free jets at Reynolds numbers greater than the critical Reynolds number, Mi et al. [65] emphasized that the jet half-width, centerline velocity, and TI calculations do not change within the self-similar region. In this context, CFD analyses were rerun for $Re = 20000$ using both default and calibrated coefficient combinations, and the results were compared with those obtained through experimental methods at $Re = 20000$. Since the coefficient calibration was based

on jet half-width calculations, it is expected that the half-width values would align with the experimentally determined jet half-widths. As shown in Figure 3.17, the jet half-width values calculated along the jet centerline for $Re = 20000$ were compared with experimental results. As anticipated, the results obtained using the calibrated coefficient combination showed better agreement with the experimental data than those calculated with the default coefficient combination. Based on these findings, it can be confidently stated that the calibrated coefficient combination for the IP RSM turbulence model is successful in jet half-width calculations from the critical Reynolds number onward. The increase in Reynolds number leads to an increase in jet exit velocity, which in turn increases turbulence kinetic energy. Consequently, the calibrated coefficients, by improving the diffusion modeling, provide more accurate half-width calculations compared to the default coefficients.

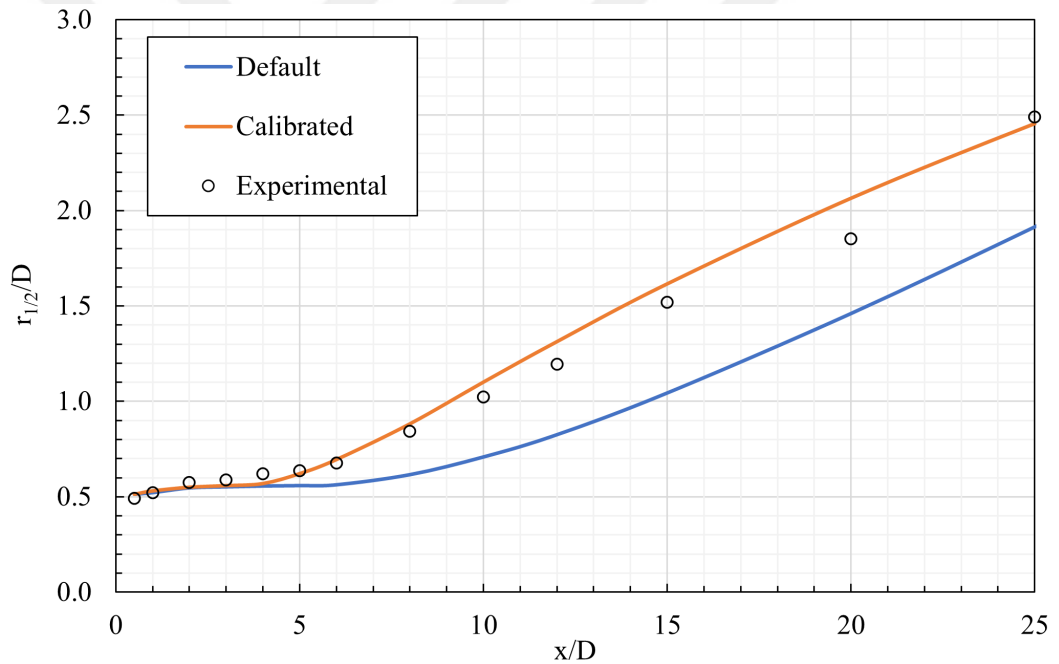


Figure 3.17 : Jet half-width results along the jet centerline for $Re=20000$.

The same cannot be said for the jet centerline velocities. Figure 3.18 shows the variation of the mean velocity distributions along the jet centerline for $Re = 20000$. The agreement between the jet centerline velocity profiles calculated with calibrated coefficients and the experimental results is not as successful as that observed for jet half-widths. Increasing the influence of the diffusion term to improve the half-width calculations reduces the effect of the advection term, which in turn decreases the turbulent kinetic energy and average velocities along the centerline. However, this

decrease occurs primarily at or near the downstream of the jet axis, and the agreement improves as the distance from the jet axis increases in the longitudinal direction. Figure 3.19 illustrates the velocity profiles on the longitudinal axis for selected stations. The discrepancy in the centerline velocities not aligning with the velocity profiles formed along the y-axis, i.e., where longitudinal axis velocity profiles are better predicted with calibrated coefficients, is again due to the increased influence of the diffusion term.

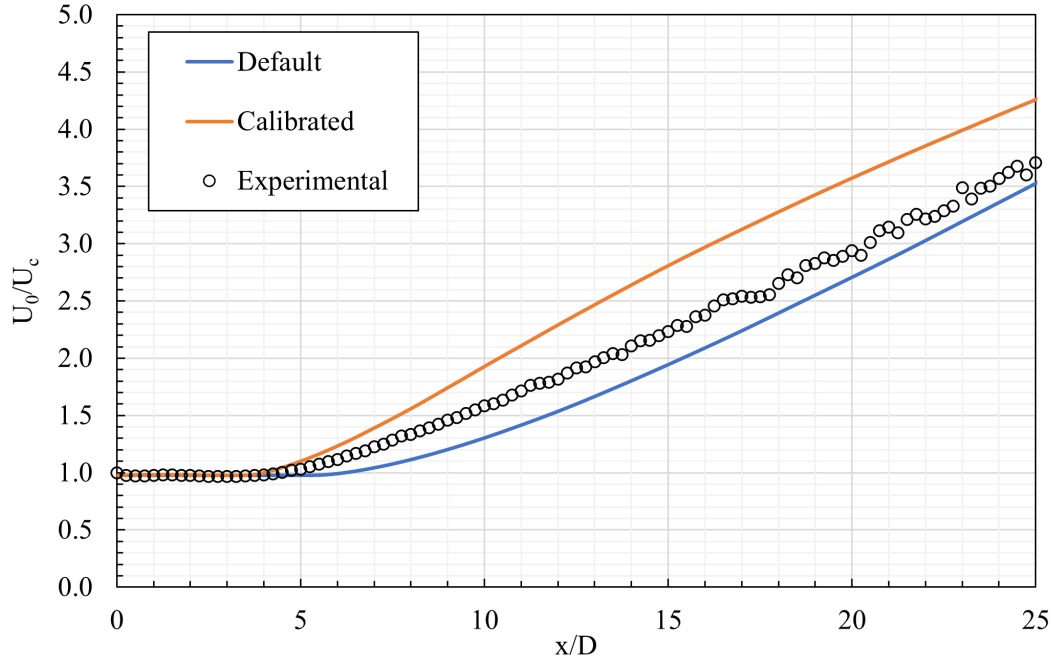


Figure 3.18 : Centerline velocity results along the jet centerline for $Re=20000$.

When comparing the velocity profiles generated along the longitudinal direction from the jet centerline with CFD analyses to experimental results, it is observed that, as with $Re = 10000$, the results in the development region ($x/D < 5$) show better agreement with the default coefficients. However, as we get further from the jet exit, the results obtained with corrected coefficients show better agreement with the experimental data. A closer examination of the jet half-widths in Figure 3.17 reveals that the region where the jet half-widths begin to develop with the default coefficients is approximately $x/D = 6$. In contrast, according to the experimental results, this development starts earlier ($x/D \approx 5$). It is evident that the calibration process has caused this development to commence earlier, leading to better alignment of the jet half-widths with the experimental results. However, the earlier onset of diffusion, particularly in the transition zone, has resulted in a reduction in average centerline velocities, thereby increasing them in the longitudinal direction. This is why the

centerline velocity results obtained with corrected coefficients appear higher than those obtained with default coefficients and experimental methods, as shown in Figure 3.19.

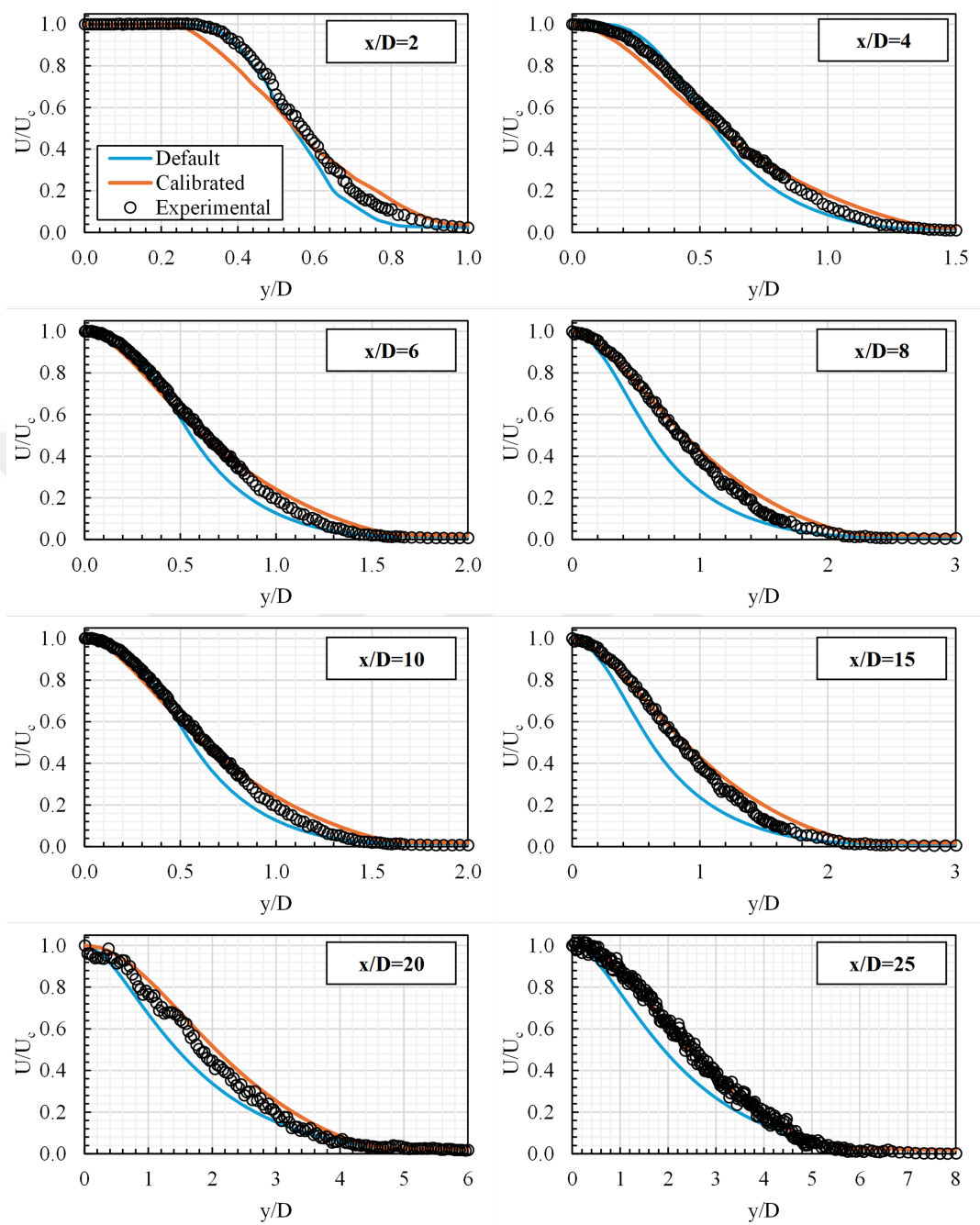


Figure 3.19 : Velocity profiles along jet axis for $Re=20000$.

It is important to note that the assumption made during TI calculation, i.e., equation 3.11, namely that "the flow is isotropic with respect to Reynolds stresses and the shear stress components are equal to zero," significantly impacts the TI calculation. Therefore, the TI graphs for $Re = 20000$ were not further investigated. Based on all these findings, it can be concluded that the jet half-widths and centerline velocity

variations calculated for $Re = 20000$ are very similar to those calculated for $Re = 10000$ as Mi et al. [65] mentioned before.

3.4 Discussion

In general, IP RSM turbulence model is most sensitive to the production constant of the dissipation equation ($C_{\epsilon 1}$), the turbulent diffusion constant of IP RSM equation (C_S), the turbulent diffusion constant of dissipation equation (C_ϵ), and the dissipation constant of the dissipation equation ($C_{\epsilon 2}$), especially when considered specifically for the first 13 stations. Calculating sensitivities in the transition region poses challenges due to the ongoing development of the flow, making it difficult to draw conclusive interpretations. In Figure 3.20, the sensitivity results are presented as bar charts on the TKE contours in the near field region after the transition (with default coefficients on the left and calibrated coefficients on the right, scaled according to station intervals). Trendlines were plotted using second-order polynomial functions to make the interpretation of TKE results more meaningful. Positive and negative sensitivity values, respectively, represent the increase and decrease in jet half-width at the corresponding station as the corresponding coefficient increases. In the near field ($5 < x/D < 25$), the sensitivity of the coefficients $C_{\epsilon 1}$, $C_{\epsilon 2}$, C_1 , and C_2 increases, while the sensitivity of C_ϵ and C_μ decreases, and C_S appears to have a nearly constant sensitivity.

The overhauled coefficient combination improved the diffusion effects to better estimate jet half-widths in the near field region. As a result, the jet half-widths, which were underestimated using default coefficients, increased to match the experimental results more closely. The higher sensitivity of the diffusion coefficients (C_S and C_ϵ) at the start of the developing region supports this finding, highlighting the importance of diffusion modeling in this region. While the diffusion coefficients are of critical importance in the developing region, where their effects are more pronounced compared to the other coefficients, towards the end of the near field the sensitivity of the production and dissipation coefficients of the dissipation equation becomes more important.

Considering the energy cascade principle, it is known that from the beginning of the near-field region along the latitudinal axis on the round jet, the effect of dissipation

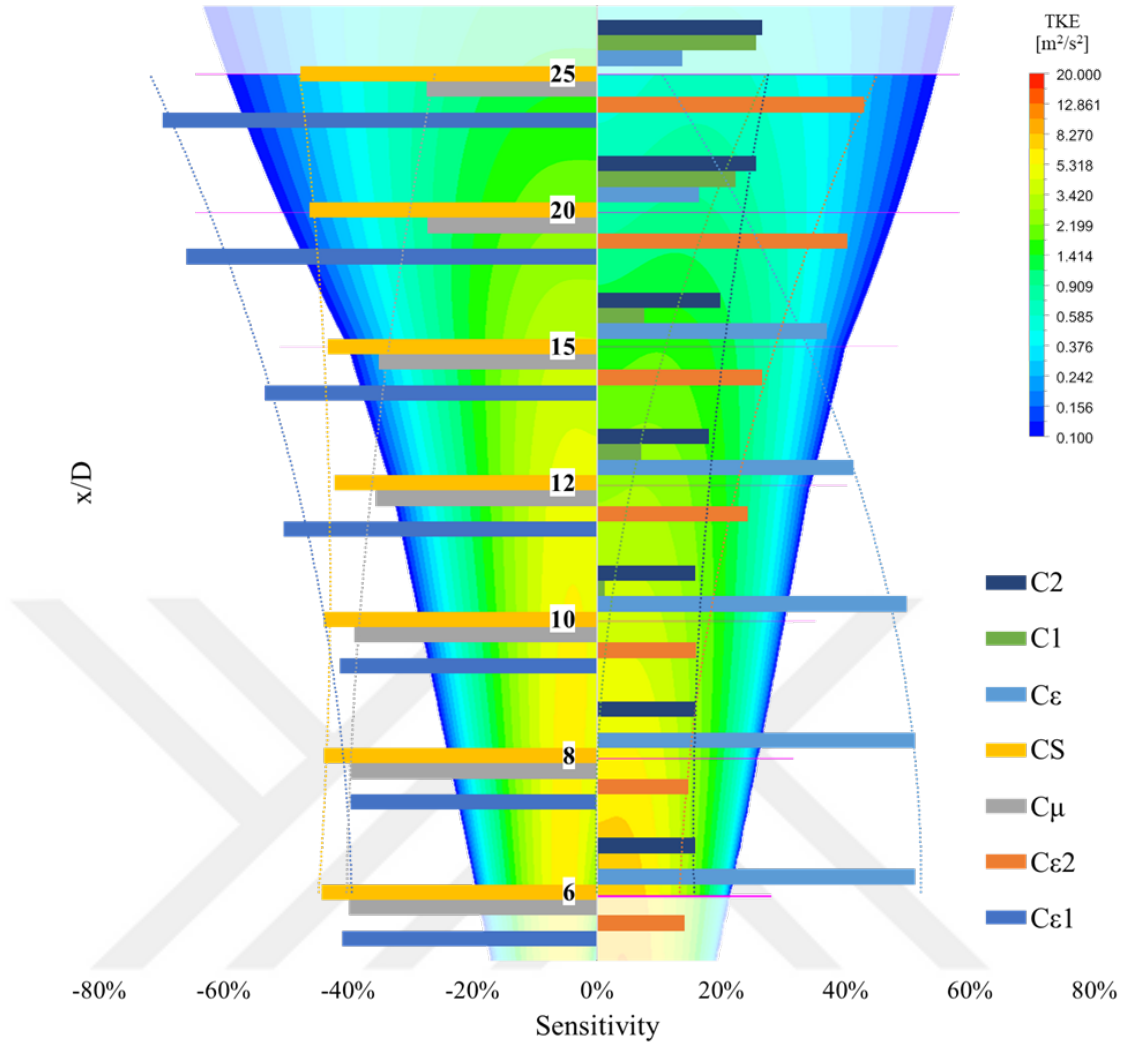


Figure 3.20 : Sensitivities of closure coefficients and TKE contour of default and calibrated coefficient results.

increases, with energy extracted from the mean flow being converted into internal energy through dissipation. The increasing sensitivity of the production and dissipation coefficients of the dissipation equation in the jet half-width calculations along the latitudinal axis emphasizes the growing influence of dissipation. As seen in the TKE budget graph in the self-similar region, the dissipation term is the largest in absolute value among the TKE equation terms, underscoring the importance of accurate modeling for both the overall flow field and jet half-width distribution. However, the modifications made to the diffusion coefficient of the dissipation equation, intended to improve diffusion modeling, also affect dissipation. To compensate for this change, the coefficients $C_{\epsilon 1}$, $C_{\epsilon 2}$, and C_μ were adjusted to refine the dissipation modeling. The trend in sensitivity value changes, particularly for the $C_{\epsilon 1}$ and $C_{\epsilon 2}$ coefficients, shows that these coefficients become the most sensitive towards the end of the

near-field region, reflecting the critical importance of accurate dissipation modeling as dissipation becomes the dominant term.

It is important to note that sensitivity calculations include interaction effects. The $C_{\varepsilon 1}$ coefficient, according to equation 3.5, is influenced by the C_{μ} coefficient. Based on the optimization results, the $C_{\varepsilon 2}$ coefficient was reduced to improve the dissipation model, which had a jet half-width reducing effect. To counteract this, the $C_{\varepsilon 1}$ coefficient was reduced to increase the jet half-width, while the C_{μ} coefficient was increased to decrease the jet half-width at a local station, thereby enhancing the dissipation model. The high sensitivity of the $C_{\varepsilon 1}$ coefficient relative to the others is attributed to its significant interaction effects.

As the flow gets closer to the self-similar region, the sensitivity of the turbulent viscosity coefficient (C_{μ}) for the jet half-width calculation decreases. The C_{μ} coefficient affects the turbulent viscosity (μ_t) calculation through its correlation with TKE and dissipation, affecting the turbulent production due to viscous forces in the dissipation equation. As we get further from the jet exit (nozzle), both TKE and dissipation diminish; however, TKE decreases much more rapidly than dissipation, leading to a significant reduction in turbulent viscosity. Consequently, the sensitivity of the turbulent viscosity coefficient also decreases.

The sensitivity of the slow part term coefficient (C_1) is minimal near the transition region but increases along the jet centerline axis. At the start of the near-field region, the high velocity gradient causes the effects of the slow part term to be negligible. However, as the dissipation effect becomes more dominant, the influence of the slow part term increases, leading to a rise in the sensitivity of the C_1 coefficient.

The rapid part term and its coefficient (C_2) affect the production term in the RSM equation and serve as a sink term for Reynolds shear stresses. A reduction in the C_2 coefficient tends to decrease the jet half-width. When examining along the jet centerline, it is expected that turbulence becomes increasingly isotropic from the start of the near-field region to the end of the self-similar region. Like the C_1 coefficient, the C_2 coefficient also contributes to making the turbulence more isotropic. The increasing sensitivity trendline of the C_2 coefficient throughout the near-field region can be attributed to this effect. Therefore, it seems reasonable to conclude that the sensitivity values calculated for each coefficient at various stations are parallel to the

dominance of the terms affected by these coefficients in the flow field at the same stations. The discrepancies between the jet half-widths calculated using CFD analyses with the default RSM model coefficients and those obtained experimentally cannot be solely attributed to the turbulence model or mesh structure. The use of upwind schemes in discretization can lead to numerical diffusion, resulting in inaccuracies in the jet half-width predictions. It has been shown through the mesh independence study that the calculation error is not due to the mesh structure or its insufficient refinement. To investigate the potential impact of numerical diffusion, the analyses of the mesh independency study were conducted using the highest discretization scheme available in Ansys[®]-CFX, the second-order upwind, and compared with results obtained using the high-resolution scheme. The high-resolution scheme aims to achieve discretization as close as possible to the second-order upwind scheme while maintaining stability [52]. Ansys[®]-CFX defines a special coefficient based on the boundedness principles used by Barth and Jespersen [101], striving to keep the specified blend factor close to 1. The specified blend factor is a value between 0 and 1, with 0 corresponding to first-order upwind discretization and 1 corresponding to second-order upwind discretization. Although higher-order schemes exist in the literature, the goal of this study is to enhance the RANS-based RSM turbulence model. Using a higher-order discretization scheme could make the problem computationally expensive, so it was not chosen for the current study.

The coefficient calibration process has redistributed the excess energy from the center along the longitudinal axis, leading to more accurate jet half-width calculations but resulting in a reduction in centerline velocity and turbulence intensity.

Overhauling the turbulence closure coefficients has improved the model of the diffusion term, enabling the RSM to better predict jet half-widths. As illustrated in the turbulence budget graph in Figure 3.14, the most significant differences between the results obtained with overhauled coefficients and those with default coefficients are observed in the advection and diffusion terms. By enhancing the diffusion term model, the accuracy of jet half-width calculations has been improved. However, this improvement comes at the expense of reducing the effects of the advection term, particularly in the jet center, leading to lower TKE values along the centerline. Consequently, the decreased TKE values have resulted in reduced TI values along the centerline. This effect is clearly evident when examining the TKE and TI contours

obtained from both default and adjusted coefficients, as shown in Figure 3.21.

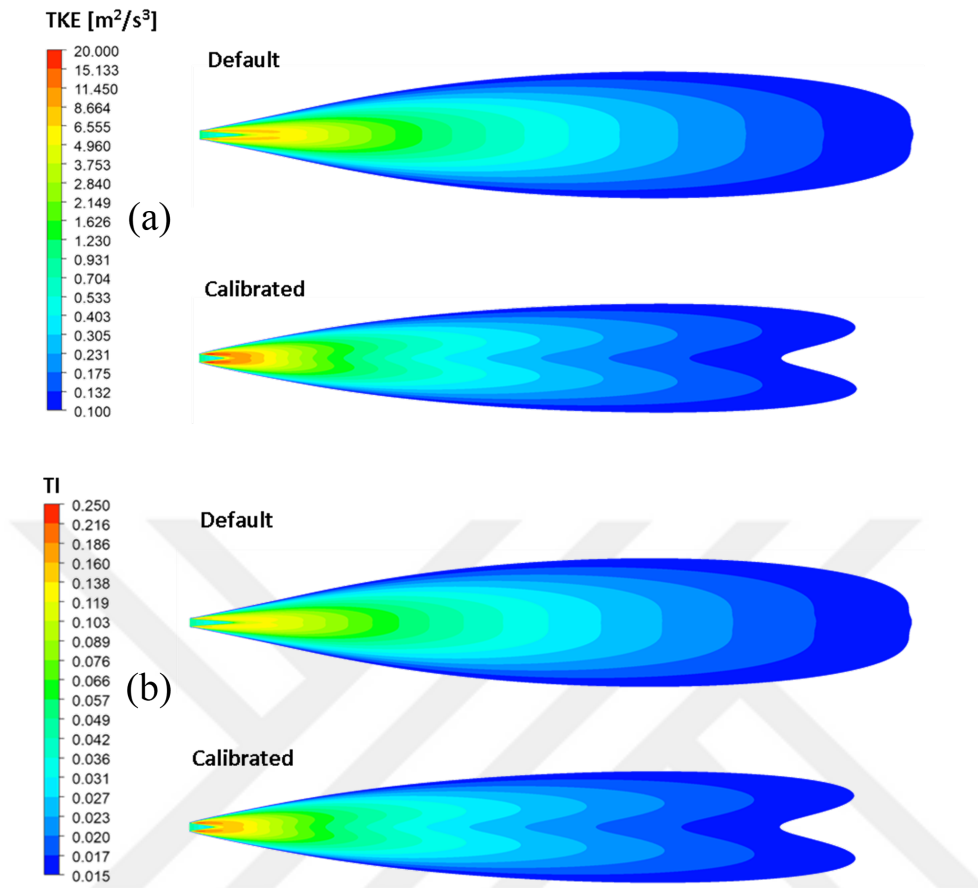


Figure 3.21 : Contours of default and calibrated coefficient results (a) TKE and (b) TI.

The reason why TI profiles calculated at different stations along the longitudinal axis with updated coefficients show relatively improved results in the near-field region compared to those calculated with default coefficients can be associated with the non-dimensionalization of the mean velocities at the centerline. Due to the modified advection effects, the velocities along the centerline are calculated to be lower with the updated coefficients, which has relatively improved the TI profiles obtained along the longitudinal axis. As observed in the velocity distribution contours in Figure 3.22, it is evident that the velocity profiles along the longitudinal axis vary significantly depending on the coefficient combination used, primarily due to the influence of advection effects.

The comparison graph of the normalized dissipation rate in Figure 3.15 further demonstrates that the optimized coefficient combination yields better results in dissipation modeling compared to the default coefficients. One of the reasons for

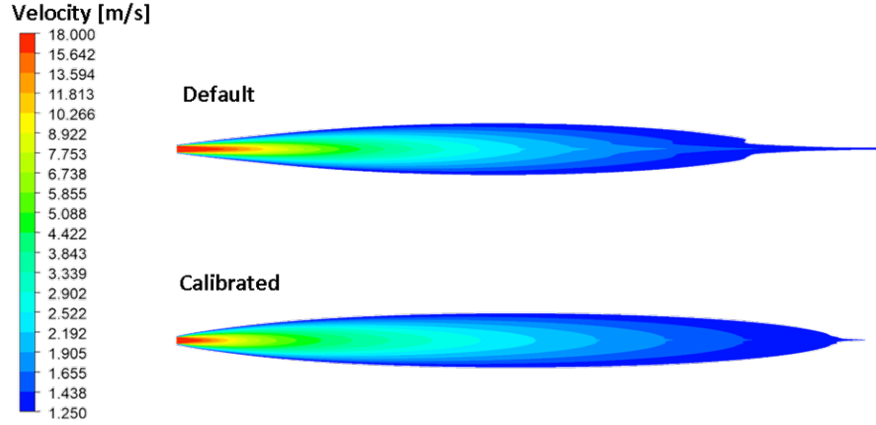


Figure 3.22 : Mean velocity contours of default and calibrated coefficients.

creating normalized dissipation rate and Kolmogorov length scale graphs based on the results of this study and comparing them with benchmark studies is to highlight that the calibration results are successful not only along the longitudinal axis (y-axis) but also along the latitudinal axis (x-axis). The optimized coefficients ensured almost perfect results, particularly in dissipation modeling. The success in calculating these dimensionless numbers in the latitudinal axis within the near-field region ($10 < x/D < 40$) can also be attributed to the calibration process, which was focused on jet half-width values in the near-field region.

The improvement of the turbulence model was mainly performed at a moderate Reynolds number of 10000; the findings in this study for the IP RSM model constants are also valid for higher Reynolds numbers. As Mi et al. [65] noted, when round free jets are at Reynolds numbers greater than Re_{cr} , the decay constant and spreading rate results do not change significantly with the Reynolds number. In Figure 3.17, it can be observed that, for calculating jet half-width in round free jets using the IP RSM turbulence model, with sufficient discretization for advection and turbulence equations and an adequate computational grid, the calibrated coefficients presented in this study will provide enhanced results compared to the default coefficients.

3.5 Conclusion

In this study, the AMO method was used to calibrate the model coefficients of the IP-RSM turbulence model, using jet half-widths obtained experimentally along the jet centerline in the near-field region as a reference. After calibration, the decay constant and jet spreading rate were calculated, and a TKE budget was generated to understand

the alterations occurring in the turbulence model. The overhauled closure coefficients yielded enhanced predictions of the decay constant, spreading rate, jet half-width, and velocity and TI profiles along the longitudinal axis for round free jet problems for Reynolds numbers at or above the threshold, although they did not remarkably improve the velocity and TI profiles along the jet centerline.

RANS-based turbulence models are widely preferred due to their lower computational efforts, but they can lead to high uncertainties depending on the flow problem. It has been shown that by adjusting the coefficients and choosing their combination correctly, these uncertainties are reduced or even eliminated in the circular free jet flow. It was shown that the IP RSM turbulence model was improved, particularly in terms of modeling diffusion and dissipation. With sufficient discretization for advection and turbulence equations and an adequate computational grid, the optimized coefficients provided improved results for calculating jet half-width in round free jets using the IP RSM turbulence model.

4. A NEW VARIANT OF RENORMALIZATION GROUP TURBULENCE MODEL FOR FLOWS IN AXIAL COMPRESSOR CASCADE³

4.1 Introduction

Turbulence mechanics -which we cannot fully explain and accurately calculate in most real-world problems, except through methods like direct numerical simulation (and even DNS requires high computational costs to solve everyday problems)- necessitates further simplifications and assumptions in solving fluid dynamics problems. Even large eddy simulation, one of the closest approaches to a direct solution, requires the use of models based on certain assumptions. In 1877, Boussinesq [8] introduced the eddy viscosity concept to the literature, paving the way for the development of turbulence models that utilize eddy viscosity instead of the Reynolds stresses term. Due to the instantaneous transient and three-dimensional nature of fluid motion, the Navier-Stokes equations were averaged by Reynolds [9], resulting in the Reynolds Averaged Navier-Stokes equations. In these equations, the stress and strain forces arising from the instantaneous motion of the fluid are referred to as Reynolds stresses. These forces, which, when directly resolved, allow us to calculate fluid motion with the closest approximation to reality, require high computational costs to solve directly in the relatively large-scale problems we encounter today.

In the early 1900s, the foundations of computational fluid dynamics were laid through the contributions of researchers such as le Roux [102], Richardson [13], Philips and Weiner [103], Lusternik [104], and Courant, Friedrichs, and Lewy [105] (known as CFL). This enabled the solution of fluid dynamics problems under various conditions and through certain assumptions. The first computer-aided solutions were achieved by Harlow and his team at Los Alamos, although their work became part of open literature around the 1960s. The study by Harlow and Nakayama [22] in 1967 laid the groundwork for what would later become the standard $k-\epsilon$ model,

³This chapter is based on the paper: Turutoglu, C. and Cadirci, S. (2025). A new variant of renormalization group turbulence model for flows in axial compressor cascade, *Physics of Fluids*, 37 (7), 075123.

introduced by Jones and Launder [25, 106] in 1972.

The $k - \varepsilon$ model allows the closure of the Navier-Stokes equations in CFD by introducing additional equations for turbulence kinetic energy and dissipation rate, thereby capturing the forces caused by turbulence. While various models were proposed before and after this period, this study focuses on the Renormalization Group $k - \varepsilon$ model, a variant of the $k - \varepsilon$ model first introduced by Yakhot and Orszag [107] in 1986. Following the development of the RANS equations by Reynolds [108] in 1987, the RNG model was further refined by Yakhot et al. [109] in 1992, evolving into the form widely used today.

To reduce computational costs and make the RANS equations solvable with the computational power available to us (easier than what is required for LES or DNS methods), several models have been developed to approximate these Reynolds stresses. The turbulence models are derived through a series of assumptions and own their closure coefficients that have yielded good results for many different conditions. However, it is well known that even minor changes in these coefficients can significantly alter the results obtained in flow field calculations. In fact, Menter [110] in his seminal 1994 paper introducing the SST and Baseline $k - \omega$ models, i.e., two of the most widely used turbulence models in both industry and academia for solving fluid dynamics problems, emphasized that even small adjustments to the model coefficients can noticeably affect the solutions obtained.

The modeling of intricate flow phenomena, such as turbulence, has advanced significantly alongside the development of CFD analysis methods. In contemporary RANS turbulence models, the closure coefficients employed are rooted in historical experimental, analytical, or theoretical methods [41, 74, 111]. Extensive studies have consistently affirmed the problem-specific nature of these coefficients. As CFD capabilities continue to evolve, recognizing the reliance on past experimental data for turbulence model coefficients underscores the ongoing need for refinement and innovation in capturing the complexities of turbulent flows with greater precision and generality, especially for turbomachinery problems.

Today, researchers recognize the significant impact of model coefficients in case-specific investigations of turbulence models. In various problems, studies have been conducted to determine the uncertainty introduced by these coefficients in results

obtained through CFD methods, and ongoing research continues to explore this area further [43, 86, 112–118]. The review study conducted by Xiao and Cinnella [119] in 2019 provides a detailed examination of uncertainty quantification in RANS models. Additionally, numerous studies have demonstrated that slight modifications to the model closure coefficients can lead to results that better align with experimental data or solutions obtained through methods like DNS [120–127].

As evident from before-mentioned studies, the optimization or calibration of coefficients for specific problems and boundary conditions has often been employed to improve CFD results based on certain comparisons, thereby reducing the uncertainty associated with turbulence models. For instance, there are studies in which the authors have enhanced the calculation of flow fields in free circular jets through optimization. Alternatively, instead of using model coefficients as fixed constants, an approach has been proposed, replacing these coefficients with variables, i.e., calculating these coefficients based on various flow parameters [128–133].

However, only a small fraction of these studies focused on turbomachinery problems. One of the studies on defining the uncertainty arising from turbulence models and their coefficients in pressure loss and flow field calculations was conducted by Banliat et al. [134] in 2019. In their study, the coefficients of different turbulence models for the V103 compressor cascade profile were obtained using a Bayesian approach under both design and off-design conditions. It was demonstrated that the combination of coefficients reducing uncertainty varies for each operating condition, highlighting the need for condition-specific optimization to minimize model-related uncertainty. Li et al. [135], also focusing on the V103 cascade profile, developed a data-driven transition turbulence model based on the Spalart-Allmaras turbulence model using machine learning methods. Their new approach enabled more accurate transition prediction compared to traditional intermittency factor equation-based transition models.

In studies conducted with RANS models, one of the most influential factors (perhaps the most influential) on computational cost is the number of computational grid cells. The cell count is directly proportional to the solution cost. However, reducing the number of cells to lower computational costs also decreases solution accuracy. Conversely, increasing the number of cells to achieve accurate results not only raises computational costs but also amplifies errors inherent in CFD problems, such

as round-off error, truncation error, discretization error, and numerical diffusion. Therefore, to demonstrate that a sufficient number of cells has been used, researchers show that the results are independent of the grid resolution by ensuring that the examined outputs -such as forces, moments, or other flow parameters (e.g., velocity, turbulence kinetic energy, pressure, or temperature distribution)- do not change or become uncertain as the number of cells increases. However, RANS turbulence models have specific expectations regarding grid resolution and cell structure. For instance, when using Menter's SST model, it is a widely accepted practice to ensure that the first cell height, represented by the dimensionless parameter y^+ , is in accordance with the viscous sublayer zone [110]. However, this requirement can lead to an increase in the number of cells and, consequently, computational costs, as well as a reduction in mesh quality for complex geometries. For this reason, researchers may employ various wall function models with different RANS models. Wall functions allow for higher y^+ values near the wall, modeling the flow in this region using the law-of-the-wall approach. This enables maintaining high mesh quality while reducing computational costs and minimizing numerical errors. The required y^+ values vary depending on the wall function and RANS model used [136]. Given the priority of cost-effective computation in this study, investigations of different wall functions and computational grid structures with varying y^+ values have also been included.

Turbomachinery designs are typically conducted on a component-by-component basis. Compressor stages, turbine stages, or combustion chamber stages are often examined individually, although it is also possible to study one or more of these components together. However, as more components are included, geometric complexity increases, making flow field analysis more challenging, i.e., leading to higher computational costs. For this reason, analyzing a single component, or even a single stage of a component, is preferred, as it reduces both computational cost and analysis time. Furthermore, to simplify the problem further, a two-dimensional or quasi-two-dimensional approach is often favored over a full three-dimensional analysis. While the two-dimensional approximation is valid under certain conditions, it may hinder realistic analysis for some components.

Compressor and turbine stages of turbomachines are relatively suitable for two-dimensional analyses, but this approach -also known as the quasi-two-dimensional approach- still requires certain assumptions. For instance, in compressor stages,

geometries in two-dimensional problems must be tapered from the inlet to the outlet to account for three-dimensional tip leakage and root corner separation effects, such as the axial velocity density ratio (AVDR). This allows for more realistic results in flow field calculations. In the design of compressors, particular attention is paid to accurately calculating certain flow field outputs, i.e., total pressure loss, inflow and outflow velocity profiles, and Mach numbers. Especially, loss calculation in compressor stages is one of the fundamental criteria for determining stage performance. Numerous studies from the past to the present have emphasized the importance of loss calculation and proposed various methods for its computation [137–140].

In turbomachinery CFD analyses, in addition to profile and three-dimensional effects, the applied boundary conditions also play a crucial role in performance calculations. Studies such as Roberts [141] and Zaki et al. [142] have shown that the turbulence intensity applied at the inlet boundary condition can influence flow separation, either triggering or preventing it, shifting the separation point forward or backward, and reducing or narrowing the separation zone. Therefore, it is essential to apply boundary conditions that accurately reflect real operating conditions.

While there are numerous studies on flow field analysis in turbomachinery problems, very few focus on model improvement. One of the primary reasons for this is that the results obtained with current turbulence models are often sufficiently accurate for certain conditions. For example, Menter's SST model is one of the most widely used models for turbomachinery problems, and its computational success has been demonstrated repeatedly [110, 143].

Another reason is the importance of boundary conditions used to study turbomachinery problems under conditions closest to their operational state, as these problems are highly sensitive to such boundary conditions. Changes in model coefficients can alter the flow field, which in turn affects the operating conditions. Therefore, it becomes challenging to perform optimization or calibration studies, such as Bayesian methods, on model coefficients while maintaining fixed boundary conditions. Variations in coefficients can lead to changes in local pressure calculations, potentially impacting critical conditions like the Reynolds number or Mach number, which are often desired to remain a specific value at the inlet. Therefore, this study focuses on compressor

cascade problems, with the objective of improving the calculation of total pressure loss in a computationally cost-effective manner.

In turbulence model improvement studies, accurate and reliable data form the foundation of progress, as the uncertainty introduced by turbulence models can only be assessed through comparison with measurements. While DNS data serves as the primary reference for validation and model enhancement, experimentally derived data are also frequently employed. Given that the DNS method demands substantial computational resources for many industrial-scale problems, experimental results are often the more practical choice.

Since the focus of the current study is on improving the total pressure loss calculation in a compressor cascade, the Double Circular Arc (DCA) V2 compressor cascade profile, whose reliability was established by The Advisory Group for Aerospace Research and Development (AGARD) Working Group 18 (WG18) [144] in 1990 and shared on the European Research Community on Flow, Turbulence and Combustion (ERCOTAC) Knowledge Base database (KBwiki) [145], has been examined. The availability of necessary data, such as total pressure loss, outlet Mach number, and wake profiles for multiple boundary conditions, made the E/CA2-V2 cascade profile the preferred choice for this study. Additionally, CFD results obtained using the RNG model are provided and compared with experimental data. However, it is observed that the CFD results obtained using the RNG $k - \epsilon$ model do not align sufficiently well with the experimental data across many parameters. Although satisfactory results have been obtained with the RNG model for problems such as backward-facing step, blunt flat plate, and flow past a 2D car model, the results for the axial compressor cascade problem were deemed insufficient by the authors, leading to the conclusion that the model could be further improved [146].

The RNG model is based on a scale elimination procedure. Small-scale eddies, assumed to be isotropic, are systematically removed until the remaining scales are resolvable with available computational resources [107]. Wilson [147], the pioneer of renormalization group methods for field theory, emphasized that when the computational grid cannot be refined enough to directly apply the fundamental equations, the renormalization group allows you to increase the grid spacing beyond what a standard numerical method would permit, while still maintaining the accuracy

and trustworthiness that comes from using the fundamental equations. This approach aligns with the authors' goal of achieving computationally cost-effective solutions. Precisely for this reason, it was believed that better results could be obtained using the RNG model introduced by Yakhot et al. [109].

However, in this study, instead of optimizing coefficients for one or multiple specific conditions and improving calculations only under certain constraints, the aim is to enhance the model coefficients based on the flow field conditions. Additionally, Yakhot and Orszag [107] emphasized that the RNG model does not include any experimentally adjustable parameters. Initially, the authors attempted coefficient optimization for some operating conditions but found that the results did not sufficiently improve under different operating points. Consequently, the focus shifted from local coefficient optimization to developing solutions that could enhance performance across all conditions.

The starting point of this study is to reduce the error rate of the RNG $k - \varepsilon$ turbulence model with low computational cost, specifically for axial compressor cascade investigations. To this end, a new function was integrated into the RNG $k - \varepsilon$ turbulence model, introducing what the authors refer to as the new generation RNG $k - \varepsilon$ turbulence model. The performance of this model was thoroughly examined across different grid structures, wall functions, CFD solvers, axial compressor cascade profiles, and operating conditions. Comparisons with experimental results and those obtained from other turbulence models demonstrate that the new model exhibits satisfactory computational capability and shows promise for the application to various flow problems.

While the computational capability of the new generation RNG model, as demonstrated through detailed analyses of axial compressor cascade problems, appears satisfactory, it is essential to investigate its performance in different types of fluid flow problems. Therefore, the authors have made the model's code and integration methods to various CFD solvers available on GitHub, inviting researchers to explore its computational performance in diverse flow scenarios.

In the subsequent sections of the study, the methods used are explained in detail, and the problems and models are introduced. Results obtained with different well-known turbulence models with their nominal coefficients are presented, followed by the

effects of the modifications made to the RNG model under selected conditions and its successful outcomes across various operating conditions. To extend the study and examine its applicability to different blade profiles, results were also analyzed for the V103 compressor cascade, and the advantages and limitations of the newly introduced model are discussed.

4.2 Methodology

4.2.1 V2 compressor cascade

The V2 highly loaded compressor cascade features a DCA profile with a 7% thickness, 56.8° camber angle, and a 0.45 pitch-chord ratio (t/l) [145]. The main geometric parameters are summarized in Table 4.1. The design condition for the V2 cascade is defined by an inlet Mach number (Ma) of 0.85, an inlet Reynolds number of 5×10^5 (based on chord length), an inflow angle (β_1) of 139.5° , and an outflow angle (β_2) of 90.5° (where $\alpha_1 = \beta_1 - 90^\circ$, $\alpha_2 = \beta_2 - 90^\circ$). According to the experimental measurements performed by Hoheisel [144], the AVDR corresponding to the design condition is 1.2. Calvert [148] further noted in his studies that this condition represents a "choked flow" scenario.

Table 4.1 : Geometric parameters of V2 cascade.

Parameter	Value
Chord length [m]	0.08
Pitch-chord ratio (t/l)	0.45
Stagger angle (β_s) [$^\circ$]	109.2

The tests were conducted by AGARD WG18 in the High Speed Cascade Wind Tunnel of DFVLR Braunschweig and include investigations of different inlet Mach numbers, inflow angles, and AVDR values for the same inlet Reynolds number [144]. The experimental data used for comparison were obtained from the ERCOFTAC KBwiki database under the Application Challenge (AC) 6-10 section. Investigated test cases are listed in Table 4.2, and the original naming conventions have been retained for consistency. More information about the tests i.e. test equipments, methodology, and uncertainties of measured data, can be found in Ref. [145].

The profile geometry, measurement planes, and computational domain are presented in Figure 4.1. The three-dimensional computational domain was constructed using

Table 4.2 : Operating conditions of test cases.

Case Name	Re	TI [%]	Ma_1	α_1 [°]	$AVDR$
Exp. 1	5×10^5	5	0.3	49.5	1.2
Exp. 2	5×10^5	5	0.6	49.5	1.2
Exp. 3	5×10^5	5	0.8	49.5	1.2
Exp. 4	5×10^5	5	0.85	49.5	1.2

a quasi-two-dimensional approach, with axial variation defined by an approximately linear variation between the inlet and exit measuring planes. This was achieved using the stream tube thickness (STT) calculation provided in KBwiki [145], which has also been validated in previous studies [144, 149].

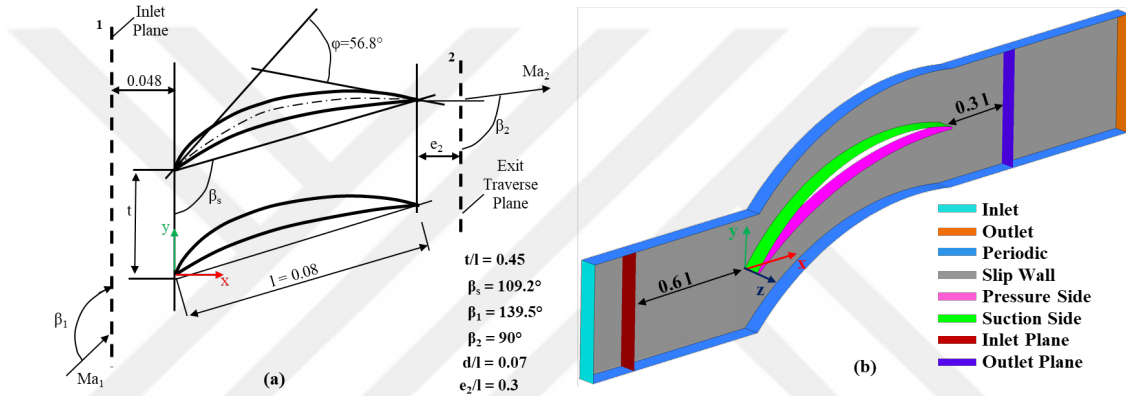


Figure 4.1 : (a) V2 cascade geometry and locations of measurement planes (b) computational domain with boundary conditions.

4.2.2 Numerical method

The physical description of the problem corresponding to the test cases is defined as continuum flow, three-dimensional, steady-state, compressible, viscous, and turbulent motion of Newtonian fluid with body forces neglected. Corresponding mathematical expressions of mean flow RANS equations solved by the Ansys[®]-CFX are given in tensor notation as [52]

$$\frac{\partial}{\partial x_i}(\rho \bar{v}_i) = 0 \quad (4.1)$$

$$\frac{\partial}{\partial x_i}(\rho \bar{v}_i \bar{v}_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}[\tau_{ij} - \rho \overline{v'_i v'_j}] \quad (4.2)$$

The term $-\rho \overline{v'_i v'_j}$ on the right-hand side of the equation is referred to as the Reynolds stress term, representing the stress caused by turbulence. This term is modeled using the Boussinesq approach (also known as the eddy viscosity hypothesis), as shown in

equation 4.3, by relating it to the mean velocity gradients.

$$-\rho \overline{v'_i v'_j} = \mu_t \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left(\rho k + \mu_t \frac{\partial \bar{v}_k}{\partial x_k} \right) \quad (4.3)$$

Subject to the eddy viscosity hypothesis, the momentum transfer equation given in equation 4.2 is rewritten as

$$\frac{\partial}{\partial x_j} (\rho \bar{v}_i \bar{v}_j) = -\frac{\partial p'}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu_{\text{eff}} \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) \right] \quad (4.4)$$

where effective viscosity (μ_{eff}) = dynamic viscosity (μ) + turbulent viscosity (μ_t). The turbulent viscosity term is calculated from the equation 4.5.

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \quad (4.5)$$

Note that fluctuating pressure, p' , is approximated as being equal to static pressure, p , in CFX (which is done by using the expert parameter “pressure value option = 2”). The Reynolds averaged energy equation is given in equation 4.6.

$$\frac{\partial}{\partial x_j} (\rho \bar{v}_j h_{\text{Tot}}) = \frac{\partial}{\partial x_j} \left[\lambda \frac{\partial T}{\partial x_j} + \frac{\mu_t}{\text{Pr}_t} \frac{\partial h}{\partial x_j} \right] + \frac{\partial}{\partial x_j} \left[\bar{v}_i (\tau_{ij} - \rho \overline{v'_i v'_j}) \right] \quad (4.6)$$

where $h_{\text{Tot}} = h + \frac{1}{2} \bar{v}_i \bar{v}_i + k$, and the last term in the equation is called the viscous work term.

In this study, the results obtained with the proposed RNG model variant were compared not only with experimental results but also with those obtained using well-known and widely used turbulence models such as the standard $k - \varepsilon$, RNG $k - \varepsilon$, and SST $k - \omega$ models. The differential transport equations for the standard $k - \varepsilon$ model, introduced by Jones and Launder [106], are as

$$\frac{\partial}{\partial x_j} (\rho \bar{v}_j k) = \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + P_k - \rho \varepsilon \quad (4.7)$$

$$\frac{\partial}{\partial x_j} (\rho \bar{v}_j \varepsilon) = \frac{\varepsilon}{k} (C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon) + \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right) \quad (4.8)$$

where P_k is the production term of turbulence kinetic energy and it is defined as $\mu_t \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) \frac{\partial \bar{v}_i}{\partial x_j} - \frac{2}{3} \frac{\partial \bar{v}_k}{\partial x_k} \left(\mu_t \frac{\partial \bar{v}_k}{\partial x_k} + \rho k \right)$ for equation 4.7, which is the turbulence kinetic energy, k , equation, and for equation 4.8, which is the turbulence dissipation rate (TDR), ε , by Jones and Launder [106] in 1973. However, in this study, P_k is defined in equation 4.9.

$$P_k = \mu_t \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) \frac{\partial \bar{v}_i}{\partial x_j} - \frac{2}{3} \frac{\partial \bar{v}_k}{\partial x_k} \left(\mu_t \frac{\partial \bar{v}_k}{\partial x_k} + \rho k \right) \quad (4.9)$$

Table 4.3 : Model coefficients.

Model	C_μ	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	σ_k	σ_ε	η_0	β_{RNG}
standard $k - \varepsilon$	0.09	1.44	1.92	1	1.3	—	—
RNG $k - \varepsilon$	0.085*	1.42	1.68	0.7179*	0.7179*	4.38	0.012
RNG F_c	0.085	1.44	1.92	1	1.3	4.38+ F_c	0.012

* $C_\mu = 0.0845$, σ_k and σ_ε are calculated using inverse effective Prandtl number formula in Fluent [150].

The coefficients used in the transport equations were adopted similarly to those in the study by Launder and Sharma [30]. The coefficients used in the models are provided in Table 4.3.

The RNG $k - \varepsilon$ model, also known as a variant of the standard $k - \varepsilon$ model and introduced by Yakhot et al. [109], uses the same TKE transport equation as the standard $k - \varepsilon$ turbulence model given in equation 4.7, while the TDR transport equation of RNG $k - \varepsilon$ model differs from the standard $k - \varepsilon$ model's TDR transport equation. The TDR transport equation of the RNG $k - \varepsilon$ model is given in equations 4.10-4.13.

$$\frac{\partial}{\partial x_j} (\rho \bar{v}_j \varepsilon) = \frac{\varepsilon}{k} (C_{\varepsilon 1 RNG} P_k - C_{\varepsilon 2} \rho \varepsilon) + \frac{\partial}{\partial x_j} \left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right) \quad (4.10)$$

$$C_{\varepsilon 1 RNG} = C_{\varepsilon 1} - f_\eta \quad (4.11)$$

$$f_\eta = \frac{\eta \left(1 - \frac{\eta}{\eta_0} \right)}{1 + \beta_{RNG} \eta^3} \quad (4.12)$$

$$\eta = \sqrt{\frac{P_k}{\rho C_\mu \varepsilon}} \quad (4.13)$$

When conducting CFD analysis, selecting an appropriate turbulence model is mostly based on experience. It is also important to use a wall treatment method suitable for the model and grid structure. There are fundamentally two approaches for calculations in the near-wall region. The first is the near-wall model approach, where the flow in the viscosity-affected region near the wall is fully resolved. The second involves using wall functions instead of resolving the flow in this region. For more detailed information on the near-wall region and the law-of-the-wall, readers may refer to Ref. [15]. The first method used in this study is the scalable wall function (Sc. WF), an extension of the standard wall function theory proposed by Launder and Spalding [151]. Unlike the standard wall function, the dimensionless distance to the wall (y^*) is limited to ensure it does not fall below approximately 11 due to mesh refinement;

otherwise, the standard wall function method is applied. The second method employed is enhanced wall treatment (EWT). If the mesh is fine enough requiring the first near-wall node to be placed at $y^+ \approx 1$, the viscous sublayer is resolved using a two-layer zonal model. This model solves the mean flow equations in the fully turbulent region of the boundary layer and employs Wolfstein's [152] one-equation model. If the mesh is not fine enough, the enhanced wall function method proposed by Kader [153] is used. This approach is summarized as the near-wall treatment method available in Ansys[®]-Fluent [150]. For more detailed information on the methods and equations used, the relevant references can be consulted.

The mean flow equations are integrated over each control volume. The resulting volume integrals are then transformed into surface integrals by applying Gauss's divergence theorem. The volume and surface integrals are reconstructed linearly with second-order accurate discretization schemes to reduce numerical errors (e.g., truncation errors, diffusive discretization errors) more effectively with mesh refinement. The finite-volume-based CFD solver, CFX, uses finite-element shape functions to approximate integration points for evaluating flux terms (i.e., mass flux, pressure gradient, and diffusion) at each hexahedral volume element. This methodology is followed in this study, with the fourth-order Rhie-Chow method selected for pressure-velocity coupling [154]. Since the problem involves compressible flow, related products are linearized implicitly using the Newton-Raphson approach. The constructed matrix of the system of equations is then iteratively solved using the COUPLED solver until the convergence criterion of the Root-Mean-Square falls below 1×10^{-6} . Further details about the computational methodology can be found in Refs. [52, 150].

Additionally, an interrupt condition was established to stop the solution when the ratio of the change in the C_L and C_D coefficients (based on chord length and inlet dynamic pressure) of the cascade profile over the last 25 iterations to the change over the last 50 iterations falls below 1×10^{-3} . This ensures that the solution is terminated even if the RMS values are below 1×10^{-5} but do not reach below 1×10^{-6} , allowing convergence to be assumed in such cases.

4.2.3 Initial and boundary conditions

In numerical studies of turbomachinery applications, the boundary conditions must accurately reflect the operating conditions [155]. At the inlet surfaces, the total pressure corresponding to the specified Ma and Re was applied, and the inflow angle (α_1) was defined through velocity components, while the turbulence intensity was set to 5% and the length scale to $8 \times 10^{-5} m$, as provided by ERCOFTAC [145]. Adiabatic boundary conditions were applied to all walls, with no-slip conditions on the cascade surfaces. The end walls in the spanwise direction were defined as slip walls to simulate end wall effects using the STT method, while periodicity was defined in the pitchwise direction. At the outlet surface, the static pressure was specified to ensure consistency with the test conditions for the Re and Ma numbers calculated using the mass-average method at the inlet plane. The total temperature at the inlet was set to 313 K for all analyses.

A well-estimated initial condition can significantly reduce the number of iterations required to reach a convergent solution in numerical analysis. Moreover, it can help prevent divergence issues that may arise in the first few iterations, especially in problems such as compressible high subsonic turbomachinery flows. In this study, for all cascade investigations, the initial conditions were defined using the relevant total pressure and Ma values, along with static pressure and static temperature derived from isentropic flow equations, as well as velocity components corresponding to the inflow angle, TI , and length scale. This approach allowed many analyses to meet convergence criteria in nearly 300 iterations. For steady-state, COUPLED solutions, the physical time step, such as the acceleration parameter, was defined as a function: $1 \times 10^{-5} s$ for the first 49 iterations, $2.5 \times 10^{-5} s$ for iterations 50–99, and $1 \times 10^{-4} s$ from the 100th iteration onward.

4.2.4 Grid structure

To assess the adequacy of the mesh structure used in the V2 cascade analyses, several investigations were conducted. Special care was taken to ensure that the computational grid was compatible with the turbulence model and wall treatment approach. Thus, it did not affect numerical stability, allowed for robust solutions, had low skewness and aspect ratio, and minimized computational cost. However, it is not feasible to create a unique mesh structure that simultaneously satisfies all these criteria for

different conditions, turbulence models, and wall treatment approaches. If evaluated individually, the optimal mesh structure for each parameter would differ. In this study, to enable a meaningful comparison of the proposed turbulence model variant with other models in the literature and to keep storage costs low, it was decided to run all CFD analyses using a single mesh structure and ensure that the comparisons were not influenced by the mesh structures. In this context, analyses were conducted with three different meshes, varying the number of nodes in the streamwise, pitchwise, and spanwise directions. These meshes were designed to achieve y^+ values below 1 even at an inlet Mach number of 0.8. The total pressure loss coefficients obtained from these analyses at an inlet Mach number of 0.6 were compared. Table 4.4 presents the results obtained using the SST turbulence model, as it provided the lowest error compared to experimental results. The mesh used in the V2 analyses is illustrated in Figure 4.2.

Table 4.4 : Results of the mesh independence study using SST turbulence model.

Grid	Node Number	Relative Error [%]	Deviation from Test Result [%]
Coarse	18500	22.216	23.022
Medium	278500	-1.505	-0.453
Fine	692900	—	1.036

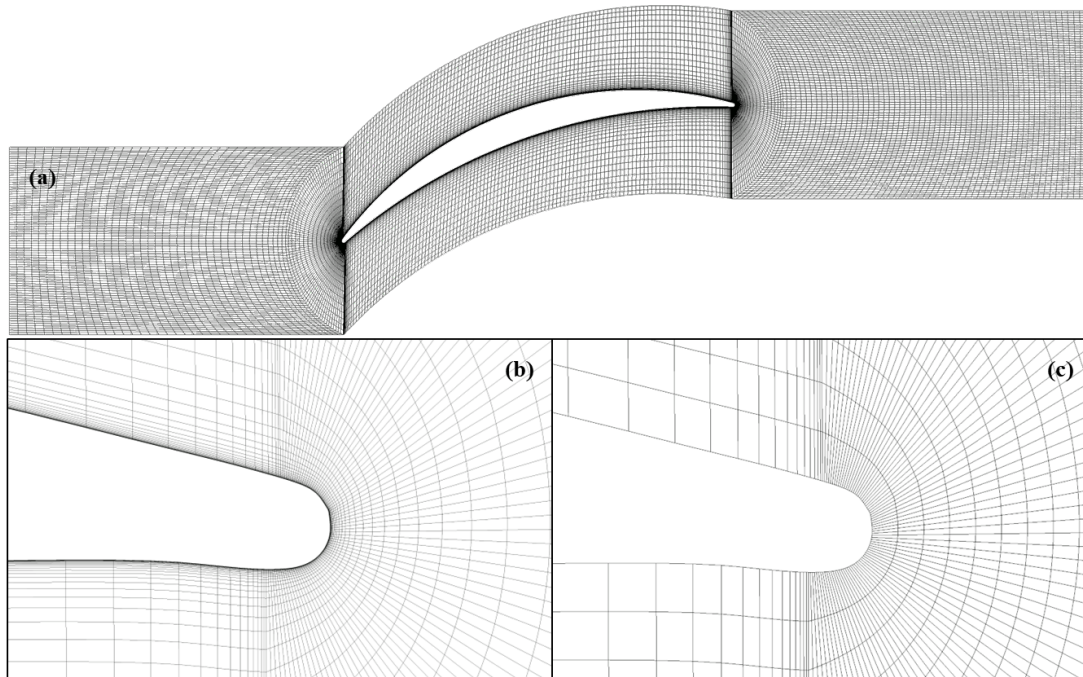


Figure 4.2 : (a) Grid structure of V2 cascade defined as medium (b) low y^+ medium grid trailing edge detail (c) high y^+ medium grid trailing edge detail.

In axial compressor cascade performance analyses, the total pressure loss coefficient ($\bar{\omega}$), widely used as a key evaluation indicator, is calculated as follows:

$$\bar{\omega} = \frac{P_{01} - P_0}{P_{01} - P_1} \quad (4.14)$$

4.2.5 Definition of RNG $k - \varepsilon$ modification

The TKE transport equation in Yakhot et al.'s [109] RNG model is nearly identical to that developed by Jones and Launder's [106] standard $k - \varepsilon$ model, except for the closure coefficients. A similar observation can be made for the TDR equations. The difference in the RNG model's TDR equation lies in its modification of the production term in the standard ε equation through the addition of a source term, R , derived from the renormalization theory in equation 4.15.

$$R = \frac{C_\mu \eta^3 \left(1 - \frac{\eta}{\eta_0}\right) \varepsilon^2}{1 + \beta_{RNG} \eta^3} \frac{1}{k} \quad (4.15)$$

where $\eta = Sk/\varepsilon$. When η^2 is substituted into equation 4.15, equation 4.11, and consequently the TDR equation in equation 4.10, is obtained. Additionally, since the equation coefficients were derived using RNG theory for high-Re flows, they are based on theoretical calculations, unlike other turbulence models. Among these coefficients, the most critical one is η_0 , which was calculated as 4.38 by Yakhot et al. [109] for homogeneous shear flow (where $\nabla^2 k = \nabla^2 \varepsilon = 0$). As pointed out by Yakhot et al. [109], the adjusted production term in the TDR equation has notable effects.

As anticipated, it reduces the rate at which TKE is produced. More interestingly, it also decreases the rate at which ε decays. This dual reduction leads to lower values of μ_t , as shown in equation 4.5. While the source term R can slightly increase μ_t in regions where η is small, μ_t generally remains lower than in standard turbulence models. However, when η exceeds 4.38, R changes sign and μ_t decreases further. Based on this fact, it is believed that this coefficient should also be modified for flow types such as boundary layer flow by relating η_0 to a new function given in equation 4.16.

$$F_c = (-\eta_0 Ma) \tanh \left\{ \max \left[0, \left(\frac{1}{\arg_1} - 0.09 \right) 500 \right]^2 \right\} \quad (4.16)$$

Here, $\arg_1$ is calculated using an expression that bears a resemblance to the general expression of the low-Re interpolation formula in RNG [146], is equation 4.17.

$$\arg_1 = \frac{\rho C_\mu k^2}{\kappa \mu \varepsilon} \quad (4.17)$$

where $\kappa = 0.41$ is the von Kármán constant. When using this model modification, it is recommended to employ the scalable wall-function developed by Launder and Spalding [151], as also suggested in studies by Thangam and Speziale [156] and Yakhot et al. [109]. Additionally, due to the possibility of very low or even vanishing TKE in cases of low free-stream turbulence intensity, a u^* limitation should be applied. In the current study, the RNG model variant, referred as the “new generation RNG,” is integrated into finite-volume CFD solvers as a source term in the TDR equation of the standard $k - \varepsilon$ model, as given in equation 4.18.

$$\text{Source Term} = -f_\eta \frac{\varepsilon}{k} P_k \quad (4.18)$$

where the f_η in equation 4.18 is redefined for RNG F_c as in equation 4.19.

$$f_\eta = \frac{\eta \left(1 - \frac{\eta}{\eta_0 + F_c}\right)}{1 + \beta \eta^3} \quad (4.19)$$

The closure coefficients of the RNG F_c model are a mix of standard and original RNG $k - \varepsilon$ model coefficients provided in Table 4.3. The values of $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k , and σ_ε are taken from the standard $k - \varepsilon$ model while C_μ and β_{RNG} are taken from the original RNG $k - \varepsilon$. Lastly, the η_0 coefficient of the RNG F_c model is assumed to be locally varying by adding the F_c function to its original value 4.38, while in the original RNG $k - \varepsilon$ model, it is assumed to be constant at 4.38.

4.3 Results and Discussion

4.3.1 V2 cascade off-design conditions

The V2 cascade is investigated for a total of four different operating conditions (three off-design conditions and one design condition) using four different turbulence models: standard $k - \varepsilon$, RNG $k - \varepsilon$, SST, and the presented RNG variant (RNG F_c). To demonstrate that the correct modifications to the turbulence model can be implemented in commercial CFD solvers, both the built-in RNG $k - \varepsilon$ model (referred to as RNG in the results) and a modified standard $k - \varepsilon$ model with the RNG source term and coefficients (referred to as sk ε RNG in the results) were used for comparison.

As shown in Table 4.5, the comparison between the results obtained with the RNG $k - \varepsilon$ model and the standard $k - \varepsilon$ model with the RNG source term (RNG - sk ε RNG) reveals that the modified standard $k - \varepsilon$ model (sk ε RNG) is highly consistent

with the results from the default RNG $k - \varepsilon$ model in CFX. It is worth noting that minor differences may arise when adjusting the outlet static pressure to ensure that the mass-weighted averaged Re and Ma numbers at the inlet plane match the test conditions. For example, the mass-weighted average Re at the inlet plane might be 499770 with a Ma of 0.79985 for sk ε RNG, while for the default RNG, it could be 500132 with a Ma of 0.801018, instead of the exact test conditions of $Re = 500000$ and $Ma = 0.8$. However, in analyses conducted with Fluent, the error rate is observed to be higher. This discrepancy arises because Fluent calculates the values of σ_k and σ_ε using the inverse effective Prandtl number equation rather than using fixed values [150]. Despite this, the comparison charts show that the sk ε RNG and default RNG curves are in close agreement, validating that the desired adjustments to the turbulence model can be achieved by adding source terms.

Table 4.5 : RNG $k - \varepsilon$ and standard $k - \varepsilon$ with RNG source term (RNG - sk ε RNG) results comparison.

Case	Results	Wall Treatment	Near-Wall Resolution	$\Delta\bar{\omega}$ [%]	ΔMa_2 [%]	$\Delta\alpha_2$ [%]
Exp. 1	CFX	Scalable WF	Low y^+	-1.57	0.23	1.03
Exp. 1	CFX	Scalable WF	High y^+	-2.69	0.07	187.10
Exp. 2	CFX	Scalable WF	Low y^+	-1.53	-0.03	1.06
Exp. 2	CFX	Scalable WF	High y^+	-1.53	0.06	47.61
Exp. 3	CFX	Scalable WF	Low y^+	-0.52	0.10	2.52
Exp. 3	CFX	Scalable WF	High y^+	0.17	0.18	27.30
Exp. 3	Fluent	Scalable WF	Low y^+	0.77	0.35	8.28
Exp. 3	Fluent	Scalable WF	High y^+	1.42	0.53	14.92
Exp. 3	Fluent	EWT	High y^+	5.36	2.49	101.30

In this study, the approximate relative error values provided are calculated according to the relative error equation given in equation 4.20, unless otherwise specified.

$$\text{Approx. Rel. Err.} = \frac{\text{Exp. Value} - \text{CFD Value}}{\text{Exp. Value}} \times 100\% \quad (4.20)$$

It has been mentioned that the CFD analysis results shared by ERCOFTAC were obtained using the STAR-CD v3.150 solver [145]. However, detailed information about the wall treatment methodology is not provided. The uncertainty levels for the measurements shared by WG18 are given as ± 0.001 for ω , ± 0.005 for C_p , ± 0.003 for Ma , and ± 0.2 for α . In Figure 4.3, the C_p values are calculated using the CFX solver with a low y^+ mesh and the scalable wall function approach on the V2 cascade profile

are plotted. Overall, it can be concluded that the boundary conditions were correctly applied for the three different off-design conditions studied.

It should be noted that the automatic wall treatment approach applied for the SST $k - \omega$ model, referred to as y^+ insensitive wall treatment, employs an enhanced wall treatment approach for the k -equation while ensuring the use of appropriate low- Re boundary conditions for the ω -equation between the laminar sublayer and the logarithmic region [52, 150]. The ability to achieve low y^+ values with the mesh used explains the superior performance of the SST model, particularly in total pressure loss calculations, compared to other models, as shown in the off-design condition results in Table 4.6. For this reason, the SST model is recommended when a very fine near-wall mesh can be constructed.

Table 4.6 : Results comparison of low y^+ grid and errors with respect to experimental measurements.

Case	Results	Model	Wall Treatment	$\bar{\omega}$	Ma_2	α_2 [°]	$\Delta\bar{\omega}$ [%]	ΔMa_2 [%]	$\Delta\alpha_2$ [%]
Exp. 1	AGARD WG18 Exp.			0.0335	0.229	0.50	—	—	—
Exp. 1	Ref. [145]	RNG	Inexact	0.0310	0.231	0.52	7.46	-0.87	-3.40
Exp. 1	CFX	sk ϵ	Sc. WF	0.0311	0.231	1.21	7.07	-1.00	-142.67
Exp. 1	CFX	RNG	Sc. WF	0.0294	0.231	1.06	12.13	-1.07	-111.46
Exp. 1	CFX	SST	Auto.	0.0340	0.232	-0.14	-1.40	-1.17	127.03
Exp. 1	CFX	RNG Fc	Sc. WF	0.0354	0.230	-0.09	-5.57	-0.47	117.20
Exp. 2	AGARD WG18 Exp.			0.0374	0.423	0.40	—	—	—
Exp. 2	Ref. [145]	RNG	Inexact	0.0327	0.431	0.66	12.57	-1.89	-64.00
Exp. 2	CFX	sk ϵ	Sc. WF	0.0335	0.432	1.39	10.46	-2.08	-246.37
Exp. 2	CFX	RNG	Sc. WF	0.0310	0.431	1.28	17.15	-1.84	-220.91
Exp. 2	CFX	SST	Auto.	0.0376	0.434	0.39	-0.45	-2.70	3.32
Exp. 2	CFX	RNG Fc	Sc. WF	0.0381	0.429	-0.36	-1.81	-1.53	189.94
Exp. 3	AGARD WG18 Exp.			0.0468	0.515	1.60	—	—	—
Exp. 3	Ref. [145]	RNG	Inexact	0.0411	0.530	1.27	12.18	-2.91	20.44
Exp. 3	CFX	sk ϵ	Sc. WF	0.0430	0.529	1.77	8.08	-2.66	-10.33
Exp. 3	CFX	RNG	Sc. WF	0.0383	0.528	1.81	18.19	-2.54	-12.87
Exp. 3	CFX	SST	Auto.	0.0467	0.538	1.22	0.15	-4.51	23.80
Exp. 3	CFX	RNG Fc	Sc. WF	0.0455	0.522	-0.48	2.71	-1.41	129.99
Exp. 3	Fluent	RNG	Sc. WF	0.0412	0.536	2.94	12.00	-4.00	-83.75
Exp. 3	Fluent	SST	Auto.	0.0481	0.565	2.02	-2.87	-9.64	-26.06
Exp. 3	Fluent	RNG Fc	Sc. WF	0.0467	0.525	0.14	0.29	-1.87	91.41
Exp. 3	Fluent	sk ϵ	EWT	0.0506	0.535	-0.31	-8.22	-3.90	119.66
Exp. 3	Fluent	RNG	EWT	0.0465	0.536	0.05	0.72	-4.02	96.83
Exp. 3	Fluent	RNG Fc	EWT	0.0466	0.526	-0.03	0.44	-2.16	102.13

When evaluating the off-design condition results obtained with the low y^+ mesh, it is observed that the ERCOFTAC CFD results exhibit lower error margins compared to

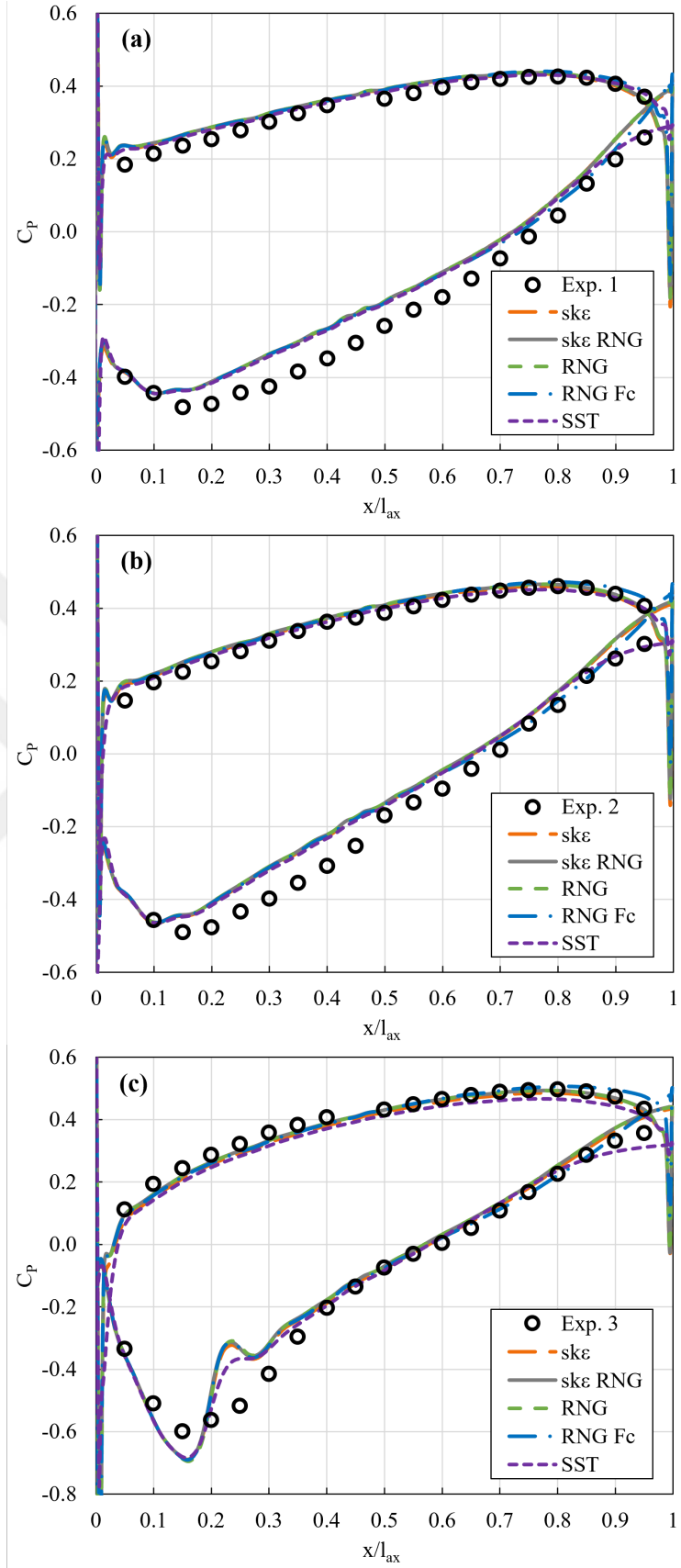


Figure 4.3 : C_p distribution on V2 cascade surfaces for low y^+ grid (a) Exp. 1 results (b) Exp. 2 results (c) Exp. 3 results.

the results calculated with the RNG model in Ansys CFX across almost all evaluation criteria, particularly in total pressure loss calculations, for all inlet Mach number conditions. However, when compared to the results of Exp. 3 using the RNG model and scalable wall function in Ansys Fluent, nearly identical total pressure loss values are obtained. Additionally, Fluent shows lower error margins in outlet Mach number and outflow angle calculations. Nevertheless, when the enhanced wall treatment is used for the RNG model in Fluent, the total pressure loss calculations are computed to be better than the ERCOFTAC CFD results. However, this success does not extend to the outlet Mach number and outflow angle calculations. The most notable aspect of the results is that the turbulence model presented in this study, referred to as RNG F_c in Table 4.6, demonstrates lower error margins in the total pressure loss and the outlet Mach number compared to both the standard $k - \epsilon$ and default RNG models, regardless of the flow solver and the scalable wall function or enhanced wall treatment approach. In the outflow angle calculations provided in Table 4.6, the relative error rates appear to be quite high. However, upon examining the wake profiles obtained using the CFX with the scalable wall function approach for the low y^+ mesh in Figure 4.4, it becomes evident that the CFD results in this study do not exhibit large errors. Outflow angle calculations were performed using a probe placed at the same location as the measurement point specified in the report, similar to the wedge-type probe used in the experiments. If the outflow angle were calculated using the mass-weighted average method at the same axial distance as the measurement plane, the values could vary. However, to ensure consistency with the experimental setup, outflow angle and outlet Mach number calculations were performed using the probe method, while total pressure loss calculations were based on the mass-weighted average at the measurement plane. A careful examination of Figure 4.4 reveals that the wake profiles obtained with the RNG F_c model show better agreement with the experimental profiles compared to those obtained with other turbulence models.

In addition to comparisons of different operating conditions and turbulence models, comparisons of different CFD solvers and wall treatment approaches were also presented. To make the comparison of different wall treatment approaches more meaningful, the mesh structure was reconstructed to maintain the same number of elements while increasing the y^+ value above 30 (averaging 60 for 0.8 inlet Mach number). The same analyses were then rerun using this mesh structure. Table 4.7

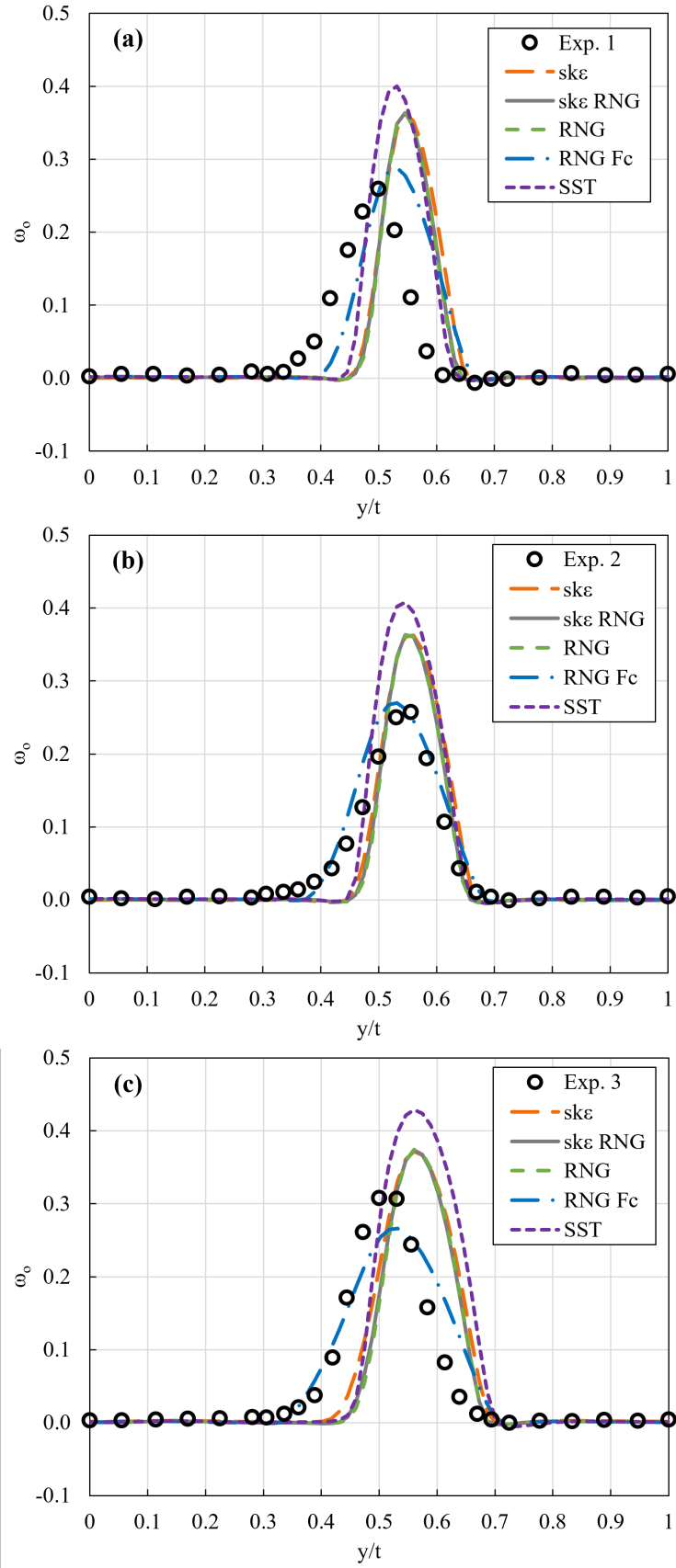


Figure 4.4 : Wake profiles for low y^+ grid (a) Exp. 1 results (b) Exp. 2 results (c) Exp. 3 results.

presents the results obtained with the coarse near-wall resolution grid structure. One of the objectives of this study, in line with RNG theory, is to provide a computationally cost-effective solution. Increasing the near-wall mesh resolution to create a low y^+ mesh structure leads to an increase in the total number of elements, thereby raising computational costs. Additionally, generating a low y^+ grid for complex geometries can reduce element quality (e.g., high aspect ratio, high skewness). For this reason, researchers often prefer using wall functions in the viscosity-affected region over fully resolving the near-wall region, especially for industrial problems. $k - \varepsilon$ - based models are compatible with wall function approaches, allowing for reasonably good results. As mentioned before, it is important for wall function methods to be compatible with the computational grid structure. To this end, results from near-wall modeling approaches such as scalable wall function and enhanced wall treatment were compared using a high y^+ grid.

It can be anticipated without further analysis that the high y^+ mesh will not be suitable for the near-wall treatment preferred by the SST model and will not yield successful calculations. Nevertheless, analyses were conducted for comparison purposes, and as shown in Table 4.7, the SST model exhibits the highest error margin in total pressure loss calculations in analyses performed both by using Ansys CFX and Ansys Fluent. This observation is also evident in the C_p charts presented in Figure 4.5. When examining the RMS error margins of the numerical C_p calculations at the points corresponding to the static pressure tapings in the experiment, it is clear that the C_p distribution obtained with the SST model has a higher error margin compared to the other models.

When using a high y^+ mesh with the scalable wall function, the capability of the total pressure loss prediction of the standard $k - \varepsilon$ model is improved. In fact, it has been found to perform even better than the default RNG model. Similarly, for outlet Mach number prediction, the standard $k - \varepsilon$ model outperforms both the RNG and SST models, regardless of the solver used. However, the RNG F_c model, which locally adjusts the η_0 coefficient, demonstrates superior prediction performance compared to other $k - \varepsilon$ models, even with a high y^+ mesh, just as it does with a low y^+ mesh. More importantly, it stands out due to its superior prediction capability over the SST model when the near-wall mesh resolution is coarse. Although the error rate in outflow angle calculations still appears high, the wake profile comparisons in Figure 4.6 suggest that

Table 4.7 : Results comparison of high y^+ grid and errors with respect to experimental measurements.

Case	Results	Model	Wall Treatment	$\bar{\omega}$	Ma_2	α_2 [°]	$\Delta\bar{\omega}$ [%]	ΔMa_2 [%]	$\Delta\alpha_2$ [%]
Exp. 1	AGARD WG18 Exp.			0.0335	0.229	0.50	—	—	—
Exp. 1	Ref. [145]	RNG	Inexact	0.0310	0.231	0.52	7.46	-0.87	-3.40
Exp. 1	CFX	sk ϵ	Sc. WF	0.0326	0.231	-0.11	2.69	-0.90	122.82
Exp. 1	CFX	RNG	Sc. WF	0.0298	0.232	0.06	10.96	-1.10	88.84
Exp. 1	CFX	SST	Auto.	0.0258	0.230	-0.98	22.98	-0.60	296.65
Exp. 1	CFX	RNG Fc	Sc. WF	0.0347	0.230	-1.15	-3.46	-0.43	329.66
Exp. 2	AGARD WG18 Exp.			0.0374	0.423	0.40	—	—	—
Exp. 2	Ref. [145]	RNG	Inexact	0.0327	0.431	0.66	12.57	-1.89	-64.00
Exp. 2	CFX	sk ϵ	Sc. WF	0.0355	0.431	0.05	4.95	-1.99	88.42
Exp. 2	CFX	RNG	Sc. WF	0.0334	0.432	0.33	10.58	-2.09	17.95
Exp. 2	CFX	SST	Auto.	0.0282	0.429	-0.56	24.72	-1.45	240.03
Exp. 2	CFX	RNG Fc	Sc. WF	0.0389	0.430	-1.45	-4.14	-1.55	462.96
Exp. 3	AGARD WG18 Exp.			0.0468	0.515	1.60	—	—	—
Exp. 3	Ref. [145]	RNG	Inexact	0.0411	0.530	1.27	12.18	-2.91	20.44
Exp. 3	CFX	sk ϵ	Sc. WF	0.0449	0.531	0.64	4.16	-3.08	60.16
Exp. 3	CFX	RNG	Sc. WF	0.0444	0.533	1.18	5.23	-3.46	26.29
Exp. 3	CFX	SST	Auto.	0.0370	0.526	0.26	20.90	-2.09	83.50
Exp. 3	CFX	RNG Fc	Sc. WF	0.0489	0.524	-1.52	-4.43	-1.79	194.75
Exp. 3	Fluent	sk ϵ	Sc. WF	0.0465	0.540	1.23	0.71	-4.90	23.24
Exp. 3	Fluent	RNG	Auto.	0.0484	0.545	1.81	-3.52	-5.75	-13.03
Exp. 3	Fluent	SST	Sc. WF	0.0646	0.645	0.06	-38.08	-25.33	96.32
Exp. 3	Fluent	RNG Fc	EWT	0.0469	0.524	-1.32	-0.13	-1.71	182.47
Exp. 3	Fluent	sk ϵ	EWT	0.0530	0.543	0.92	-13.20	-5.39	42.34
Exp. 3	Fluent	RNG	EWT	0.0562	0.549	0.03	-20.13	-6.51	98.25
Exp. 3	Fluent	RNG Fc	EWT	0.0561	0.527	-0.03	-19.97	-2.35	101.57

the RNG F_c approach is more consistent with the experimental results.

In a general evaluation of off-design conditions, the RNG F_c model yields more successful results than the standard $k - \epsilon$ model. More importantly, it also outperforms the default RNG model. Remarkably, while it demonstrates comparable success to the SST model for fine near-wall mesh resolution, it exhibits superior prediction capability over the SST model for coarse near-wall mesh resolution. Additionally, it should be noted that the RNG F_c model performs well with both scalable wall function and enhanced wall treatment approaches when using a low y^+ grid. However, when the mesh has a high y^+ value, better results are achieved with the scalable wall function, as expected.

Ansys Fluent allows the calculation of effective viscosity for the default RNG model in low- Re and near-wall flows using the differential relation given in equation 4.21,

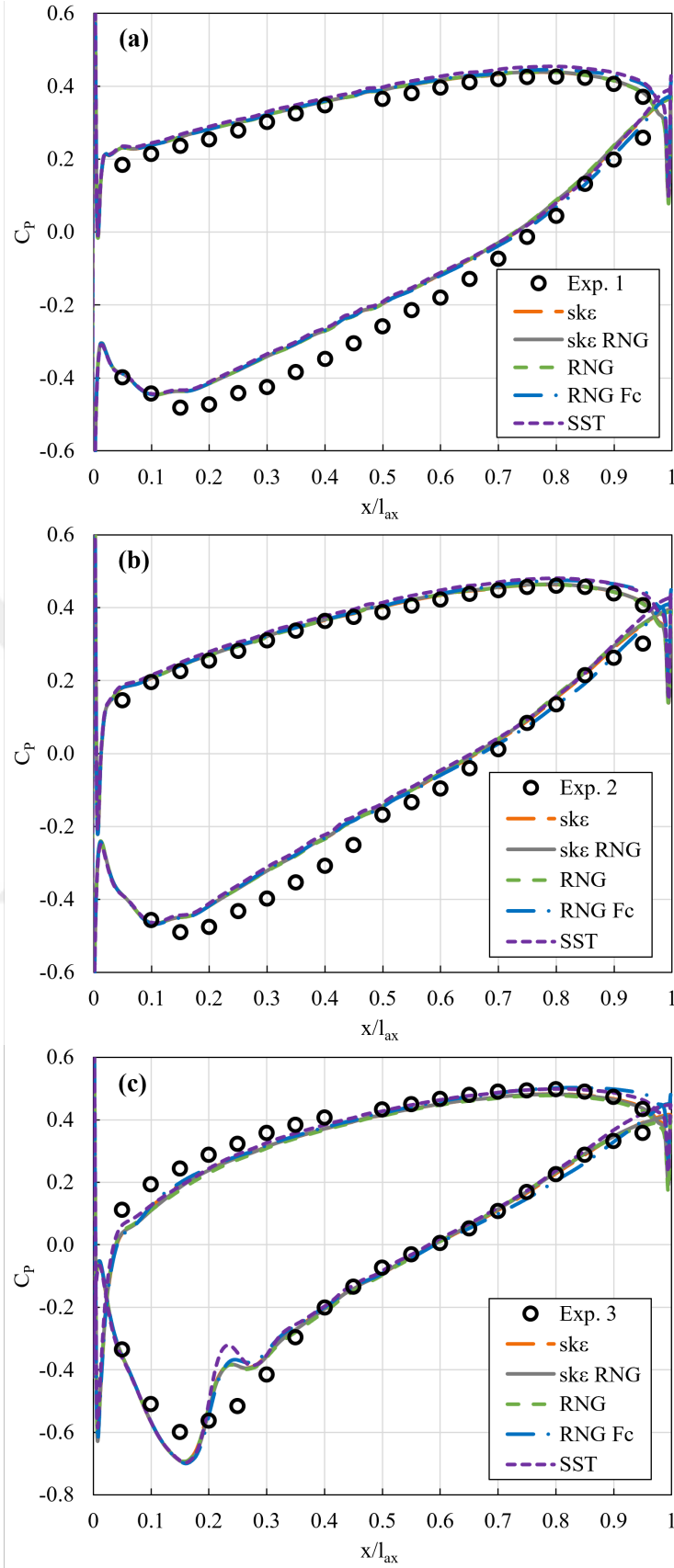


Figure 4.5 : C_p distribution on V2 cascade surfaces for high y^+ grid (a) Exp. 1 results (b) Exp. 2 results (c) Exp. 3 results.

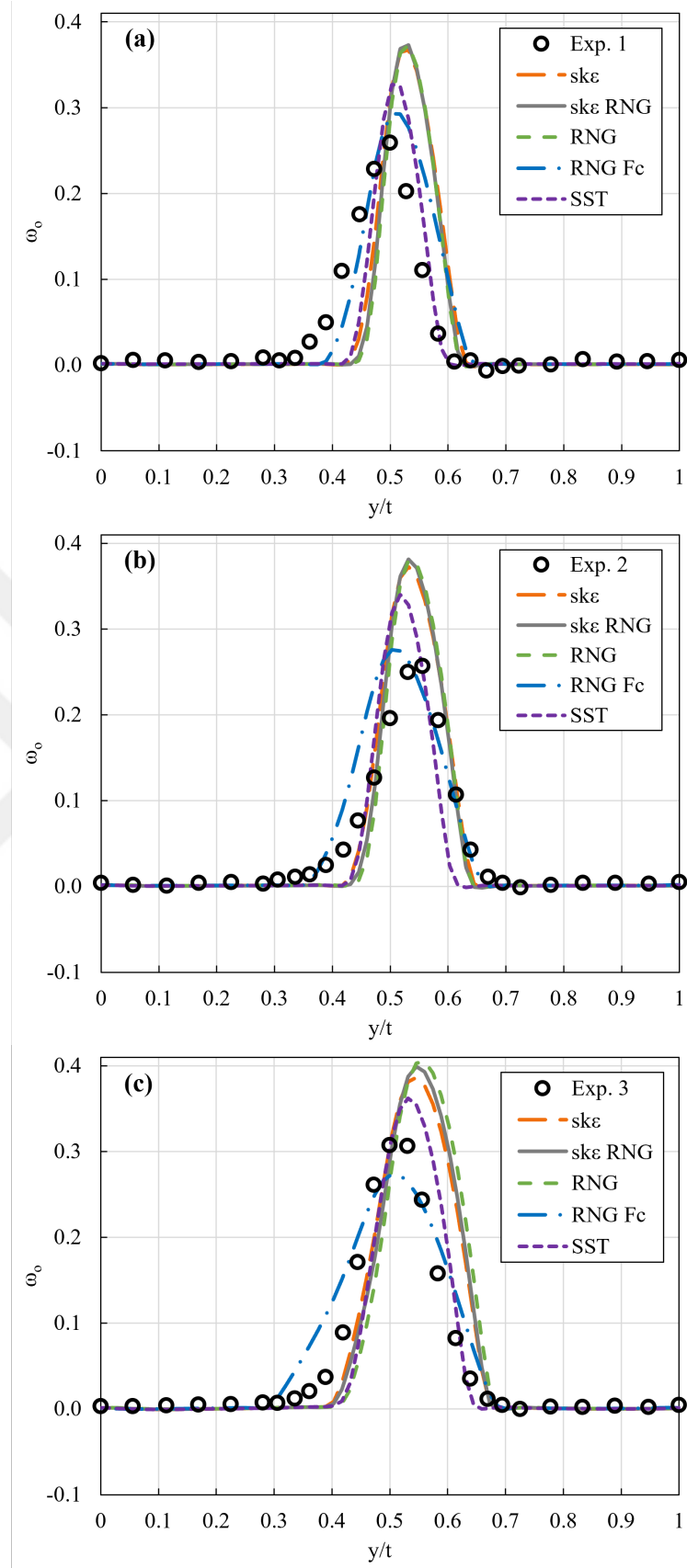


Figure 4.6 : Wake profiles for high y^+ grid (a) Exp. 1 results (b) Exp. 2 results (c) Exp. 3 results.

derived from the scale elimination procedure of RNG theory [150].

$$d \left(\frac{\rho^2 k}{\sqrt{\varepsilon \mu}} \right) = 1.72 \frac{\hat{\nu}}{\sqrt{\hat{\nu}^3 - 1 + C_v}} \quad (4.21)$$

where $\hat{\nu} = \mu_t/\mu$, and $C_v \approx 100$. To provide a comprehensive evaluation, analyses were conducted for Exp. 3 using both mesh structures and different wall treatment approaches with differential viscosity calculation given in equation 4.21. When using the low y^+ mesh with the scalable wall function, the CFD analysis became numerically unstable, preventing convergence and yielding no comparable data. However, with the enhanced wall treatment, the results compared to the experimental data showed relative errors of -0.32% for total pressure loss and -4.49% for outlet Mach number. In contrast, with the high y^+ mesh (coarse near-wall resolution), these errors increased to -22.72% and -7.35% , respectively. When the scalable wall function was applied to the high y^+ mesh, the errors were -9.45% and -6.86% , respectively. These results highlight the success of the RNG F_c model once again.

Another point that may raise questions is whether this model is numerically stable and converges well. To clarify this, the RMS values of the governing equations for the analyses run in CFX for Exp. 3 using two different mesh structures are provided in Figure 4.7. It can be confidently stated that the modification made by adding a source term to the TDR equation does not negatively impact numeric stability and convergence.

It can be said that the η function is directly proportional to the production-to-dissipation ratio. This ratio varies across different flow types, e.g., it is approximately 5 in free mixing layers at equilibrium but around 3 in the logarithmic layer. The assumption of $\eta_0 = \text{constant} = 4.38$ causes the source term to change sign due to the calculation of the f_η function as $1 - \eta/\eta_0$. In Figure 4.8, the contours of the f_η function values for the default RNG and RNG F_c models, calculated using CFX, are provided along the midplane. It is observed that the value of the f_η remains nearly unchanged across the flow domain, regardless of variations in near-wall resolution. In the default RNG model, the f_η function ranges between ± 1 . In the RNG F_c model, however, it increases up to 1.5, with its effect particularly pronounced in the wake region. Since the function appears with a negative sign in the TDR equation, it results in lower production of dissipation. Consequently, higher turbulent diffusion occurs, explaining why the wake profiles obtained with RNG F_c are wider.

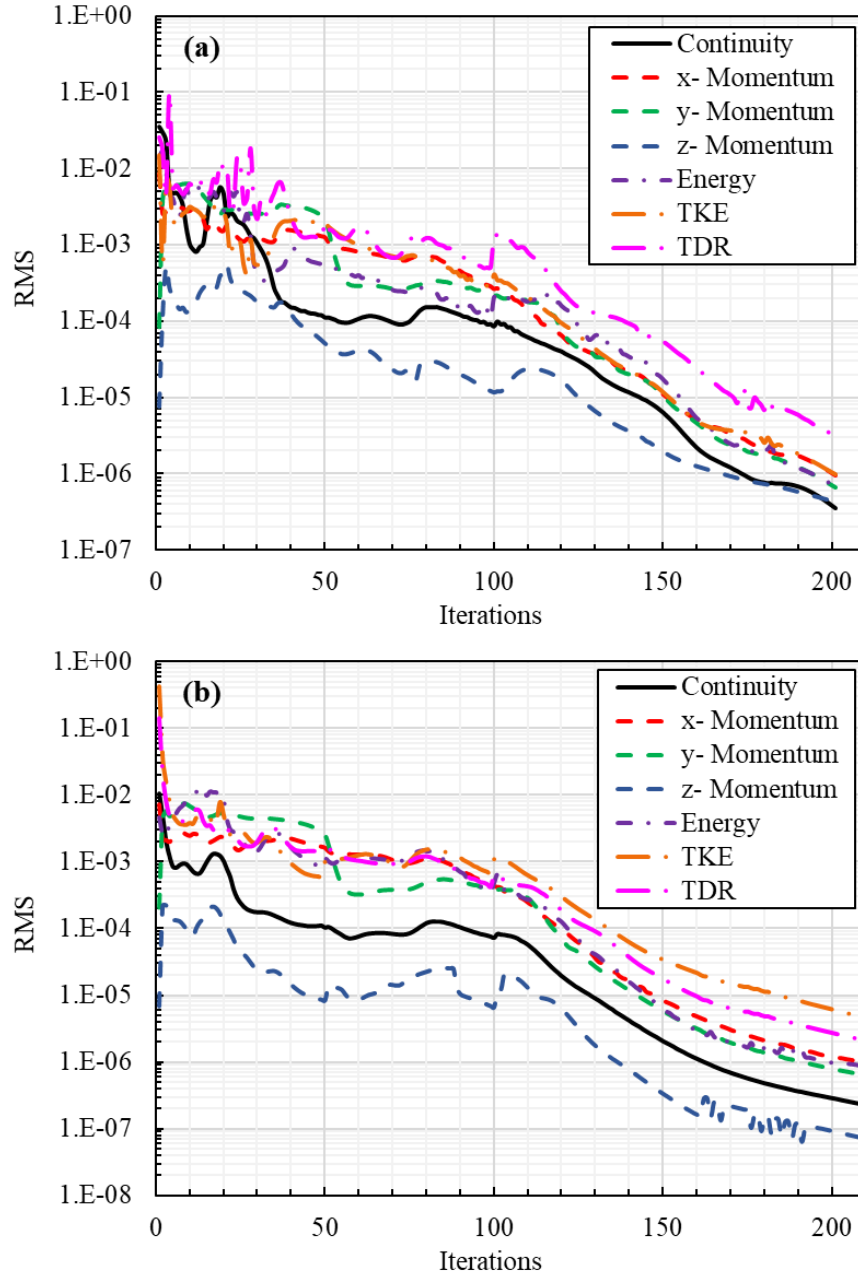


Figure 4.7 : RMS of governing equations for Exp. 3 with RNG F_c model (a) low y^+ grid results (b) high y^+ grid results.

The effects on the pressure and suction surfaces of the cascade differ from those in the flow domain, and as seen in Figure 4.9, the results of the RNG F_c model vary significantly depending on the near-wall resolution. This demonstrates that the RNG F_c model can be successful with both grids. Of course, it is important to note that the use of the scalable wall function also contributes to this success. In the default RNG model, the source term values are similar for both mesh structures. In the analysis using the high y^+ mesh, the default RNG model shows a significant drop in the source term on the pressure surface within the range $0.05 < x/l_{ax} < 0.2$. In this range, the

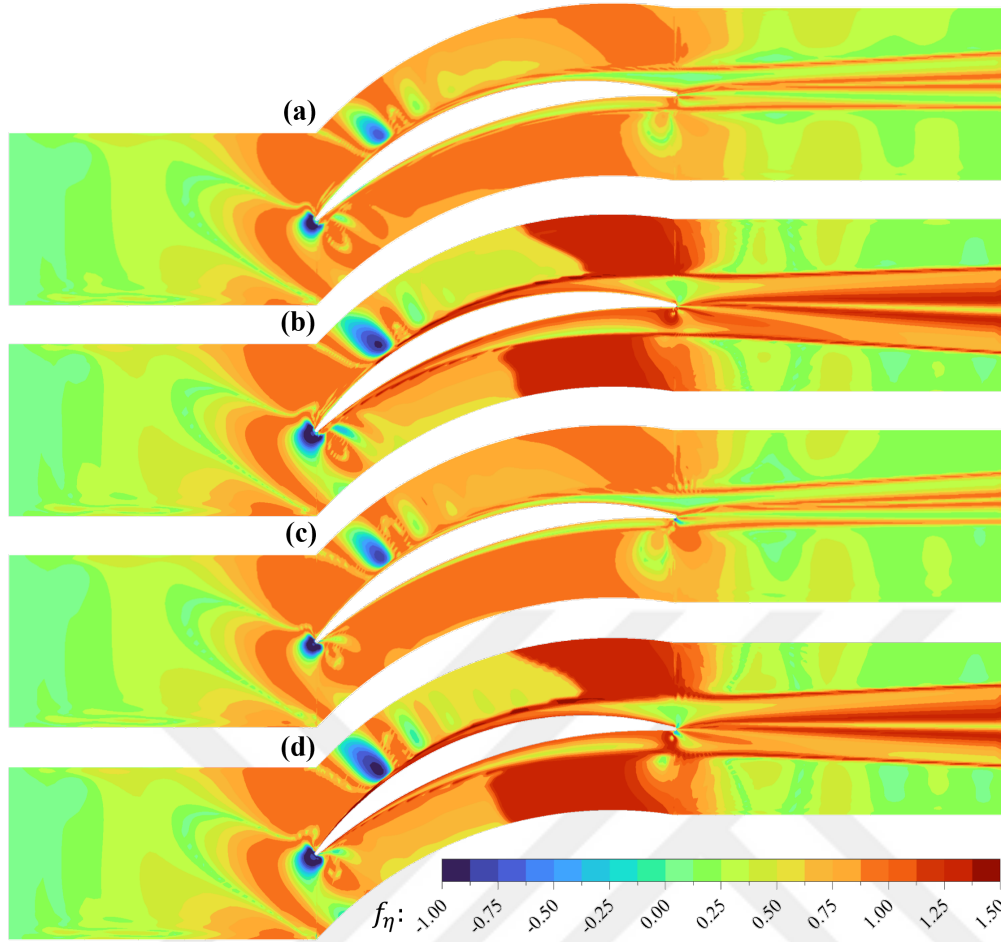


Figure 4.8 : f_η contours of Exp. 3 test case (a) RNG with high y^+ grid (b) RNG F_c with high y^+ grid (c) RNG with low y^+ grid (d) RNG F_c with low y^+ grid.

η value increases, and when it exceeds 4.38, the source term changes sign. With the RNG F_c modification, η_0 is increased in this range, preventing the turbulent viscosity from dropping unnecessarily. As emphasized by Orszag et al. [146], RNG theory has the interesting consequence of decreasing both the rate of production of TKE and the rate of dissipation of ε , leading to smaller values of μ_t according to equation 4.5. In regions of small η , the source term tends to increase μ_t somewhat. However, in regions of large η , the sign of the source term changes, and μ_t decreases even further. However, this effect is not observed when near-wall mesh refinement is increased. Therefore, it is interpreted that the scalable wall function approach in the high y^+ mesh artificially increases the η value beyond what is physically reasonable, leading to an intervention that contradicts the problem's physics.

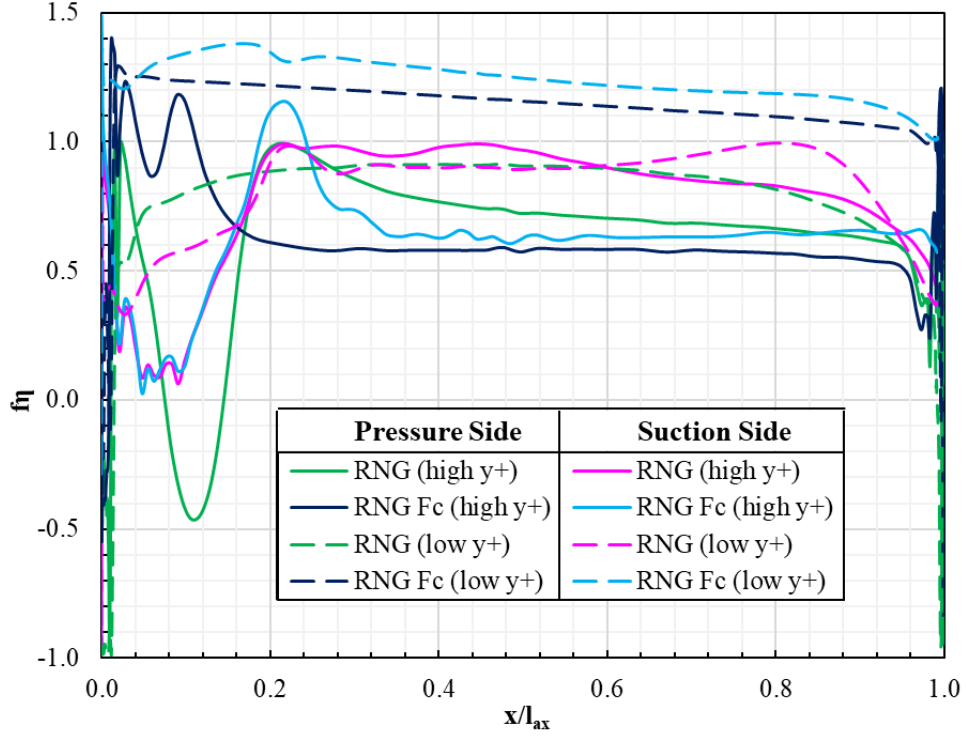


Figure 4.9 : f_η distribution on cascade surfaces for Exp. 3 test condition.

4.3.2 V2 cascade design condition

The comparison results with the test condition referred to as Exp. 4, which Calvert [148] emphasized as a “choked flow,” were examined separately due to the slightly different physics of the problem compared to off-design conditions (e.g., shock wave-boundary layer interaction effects). Although the boundary conditions corresponding to the test conditions were applied at the inlet surface using isentropic flow equations for a Mach number of 0.85, the Reynolds number and Mach number calculated using the mass-weighted average method at the inlet plane could not be adjusted to match the test conditions of $Re = 500000$ and $Ma = 0.85$, regardless of how much the outlet static pressure was reduced. Since no CFD results were shared in the ERCOFTAC databases for this condition, it is not included in the comparisons in Table 4.8. It is believed that results for this condition were not included because choked flow occurs at $AVDR = 1.2$ in test case Exp. 4, making it impossible to obtain results for this condition. Since the inlet total pressure is defined as the inlet boundary condition, using Mach numbers higher than 0.85 in the isentropic flow equations leads to changes in the inlet flow angle due to choking. To enable a meaningful comparison of turbulence models, the outlet static pressure values were adjusted until the total pressure loss values matched the experimental results. Therefore, the total pressure loss values in Table 4.8 are

similar and have low relative errors.

Table 4.8 : Results comparison of Exp. 4 and errors with respect to experimental measurements.

Grid	Results	Model	Wall Treatment	$\bar{\omega}$	Ma_2	α_2 [°]	$\Delta\bar{\omega}$ [%]	ΔMa_2 [%]	$\Delta\alpha_2$ [%]
Low y^+	AGARD WG18	Exp.		0.0779	0.540	1.70	—	—	—
Low y^+	CFX	sk ϵ	Sc. WF	0.0777	0.556	2.65	0.21	-2.91	-55.89
Low y^+	CFX	RNG	Sc. WF	0.0781	0.559	2.97	-0.24	-3.49	-74.87
Low y^+	CFX	SST	Auto.	0.0780	0.568	1.72	-0.13	-5.12	-1.06
Low y^+	CFX	RNG F_c	Sc. WF	0.0781	0.542	-0.08	-0.31	-0.32	104.61
High y^+	CFX	sk ϵ	Sc. WF	0.0777	0.558	1.67	0.28	-3.36	1.51
High y^+	CFX	RNG	Sc. WF	0.0783	0.569	2.58	-0.46	-5.30	-51.84
High y^+	CFX	SST	Auto.	0.0774	0.564	2.60	0.69	-4.43	-53.19
High y^+	CFX	RNG F_c	Sc. WF	0.0783	0.541	-1.04	-0.48	-0.21	161.37

However, the C_p distributions on the cascade in Figure 4.10 reveal that the SST model fails to capture the shock wave-boundary layer interaction effect on the suction side when using a low y^+ grid. In the region where $x/l_{ax} > 0.8$, low C_p values due to flow separation are observed. On the pressure side, the SST model exhibits the lowest C_p distribution. A similar trend is observed in the high y^+ grid analyses. However, in the high y^+ grid case, the SST model correctly captures the shock wave-boundary layer interaction effect but predicts it more intensely than the other models, resulting in a lower minimum C_p value.

Based on the wake profile comparisons in Figure 4.11, it is fair to conclude that the SST model is not successful with the low y^+ mesh. However, in the high y^+ mesh analyses, it achieves results close to those of the other default models. The RNG F_c results, on the other hand, appear significantly successful both in terms of C_p distributions and wake profiles. The C_p distributions on the pressure side are closer to the measured data for both mesh structures. On the suction side, it achieves C_p distributions similar to those of the default $k - \epsilon$ models (i.e., standard $k - \epsilon$ and RNG $k - \epsilon$). In the wake profile comparisons, although the RNG F_c model does not match the maximum pressure loss value, it provides more accurate flow angle predictions. Based on these results, it can be concluded that the RNG F_c model is generally successful in axial compressor cascade analyses. However, it should be noted that the model presented in this study was developed specifically to improve total pressure loss calculation capability. Fluid dynamics problems, such as flow choking, involve complex physical phenomena in

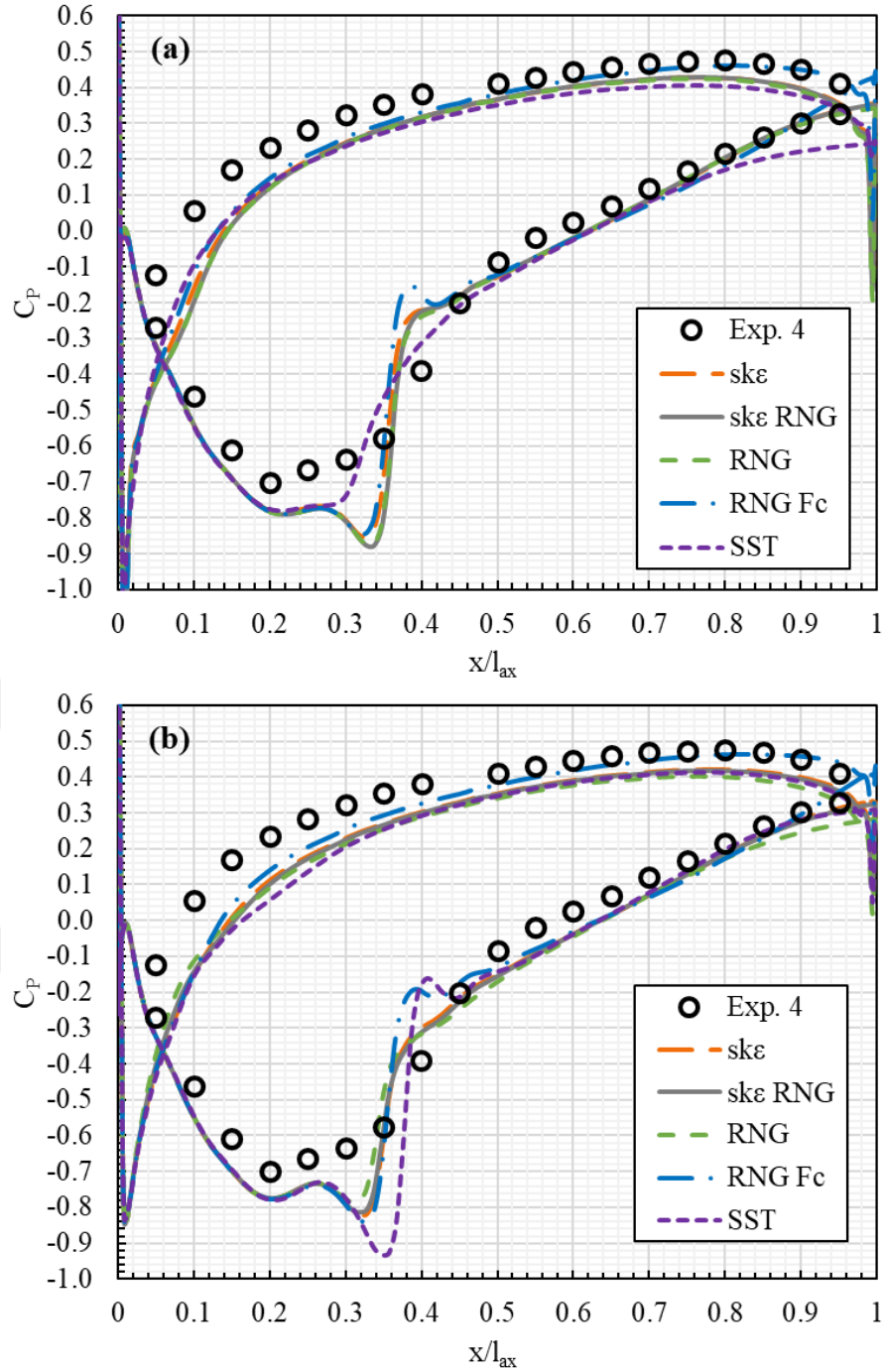


Figure 4.10 : Design condition (Exp. 4) results of (a) low y^+ grid C_p distribution (b) high y^+ grid C_p distributions.

terms of flow regimes and solution approaches, and it is not claimed that the model can fully address these complexities.

4.3.3 V103 cascade

To extend the validation study, CFD analyses were conducted using the RNG model variant presented in this study for a different cascade profile with distinct design

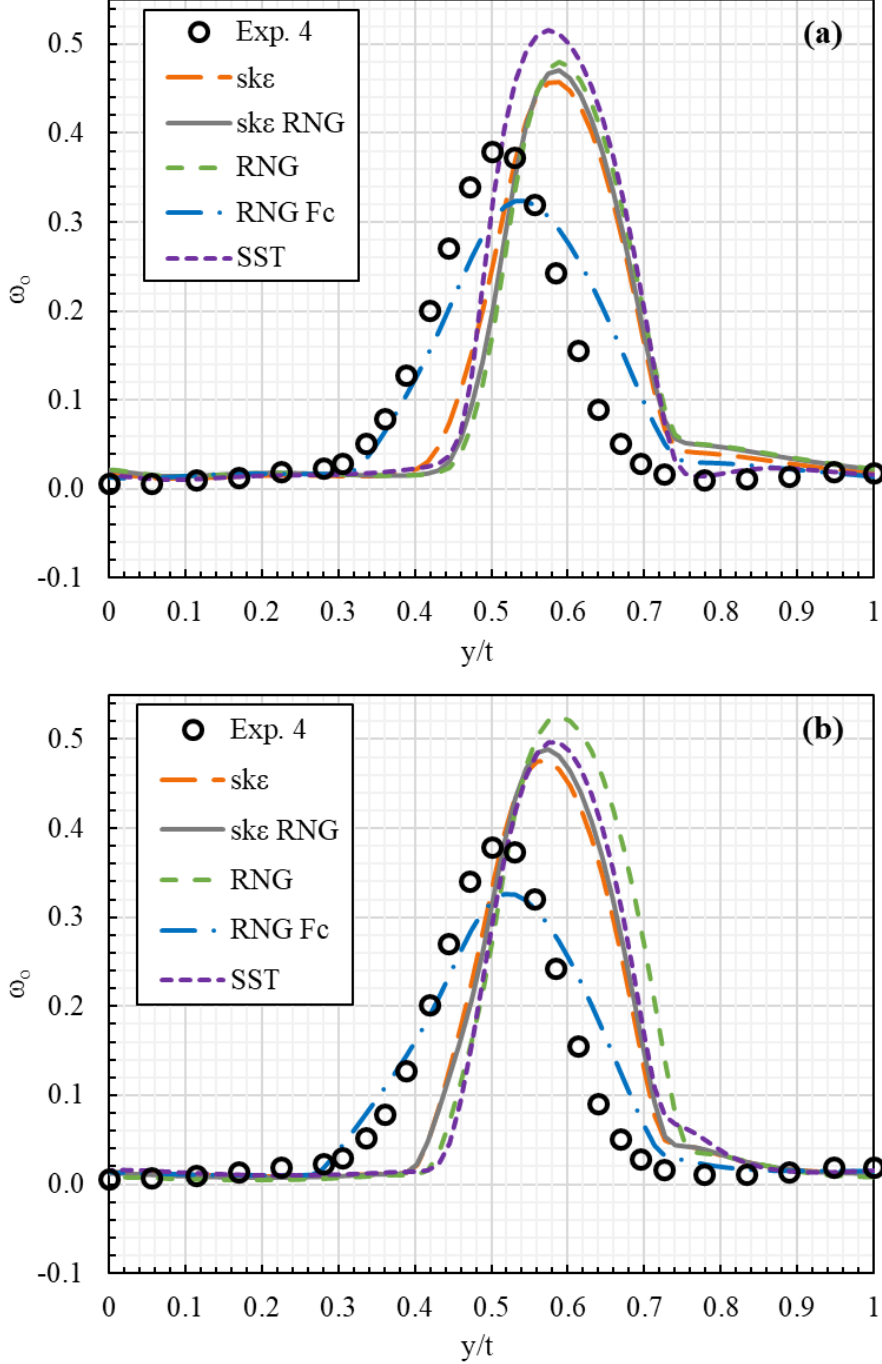


Figure 4.11 : Wake profiles of Exp. 4 (a) low y^+ grid results (b) high y^+ grid results.

conditions. As a benchmark study, the widely used V103 cascade profile, which exhibits different physical phenomena compared to the V2 cascade profile, was analyzed under design and off-design conditions using Yakhot et al. [109] RNG $k - \varepsilon$, Menter's SST $k - \omega$ [110], and the RNG F_c model introduced in this study [157–159]. The design parameters for the V103 highly loaded compressor cascade are provided in Table 4.9.

Numerous experimental data sets from various researchers are available in the

Table 4.9 : Design parameters of V103 axial compressor cascade.

Parameter	Value
Chord length [m]	0.18
Pitch-chord ratio (t/l)	0.55
Stagger angle (β_s) [$^\circ$]	109.2
Inflow angle (β_1) [$^\circ$]	132
Outflow angle (β_2) [$^\circ$]	96
Inlet Ma	0.67
Inlet Re	450000

literature for this profile, and the availability of data for different inlet Reynolds number conditions was a key factor in its selection. One of the most significant differences from the V2 cascade flow is the formation of a laminar separation bubble on the suction surface under design conditions. Although phenomena such as the flow separation point, the onset of transition, and separation length are complex and difficult to predict with standard RANS models, they were included for comparison purposes. The calculations were performed using the Ansys CFX with the scalable wall function and a computational grid (with 377000 hexahedral cells) generated to achieve $y^+ < 1$. The profile coordinates were taken from the study by Bell and Fottner [160], and the measurement planes were placed according to the coordinates specified in the same study. The inlet turbulence intensity was set to 2%, as Leipold et al. [161] reported this value for the distribution of TI outside the boundary layer under design conditions. The inlet length scale boundary condition was defined as 10% of the axial chord length. For the design condition, the $AVDR$ was given as 1.06 by Bell and Fottner [160] and the computational domain was constructed with axial variation from the leading edge to the trailing edge, following the approach of Li et al. [135] with an inlet spanwise extent of 0.16 chord length.

Boese and Fottner [162] calculated total pressure loss coefficients for off-design conditions at inlet Reynolds numbers varying from 150000 to 1200000. The isentropic Mach number (Ma_{is}) distribution on the profile for the design conditions is shown in Figure 4.12.

When compared to the experimental results, it is evident that none of the three models capture the onset of transition at around $x/l \approx 0.5$. In fact, for this reason, the use of RANS models with transition computation is recommended in turbomachinery CFD analyses to address such issues [158]. Nevertheless, this comparison is considered

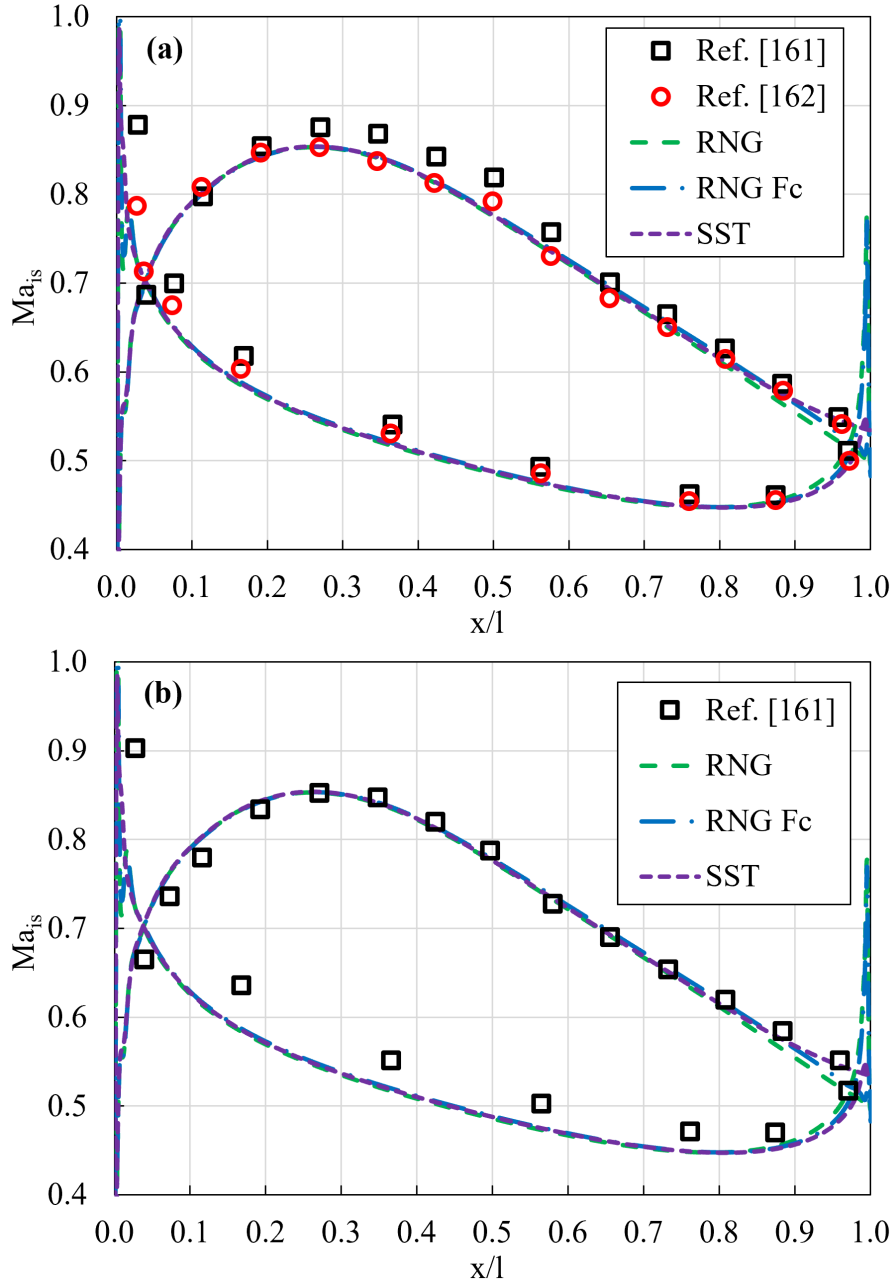


Figure 4.12 : Isentropic Mach number distributions on V103 cascade (a) inlet $Re=450000$ result (b) inlet $Re=800000$ result.

to be valuable for a detailed examination of the model and its total pressure loss calculation capability.

In Figure 4.13, the total pressure loss coefficients calculated under design and off-design conditions using different turbulence models are compared with the results obtained from experimental methods. Under the design condition, the RNG F_c model achieved the most accurate total pressure loss calculation with 5% approximate relative error. The SST model showed a relative error of about 25%, while the RNG $k - \varepsilon$ model calculated the total pressure loss with approximately 17% relative error compared to

measurements. For the outflow angle, the SST, RNG, and RNG F_c models exhibited relative errors of 9.54%, 31.54%, and 15.18%, respectively.

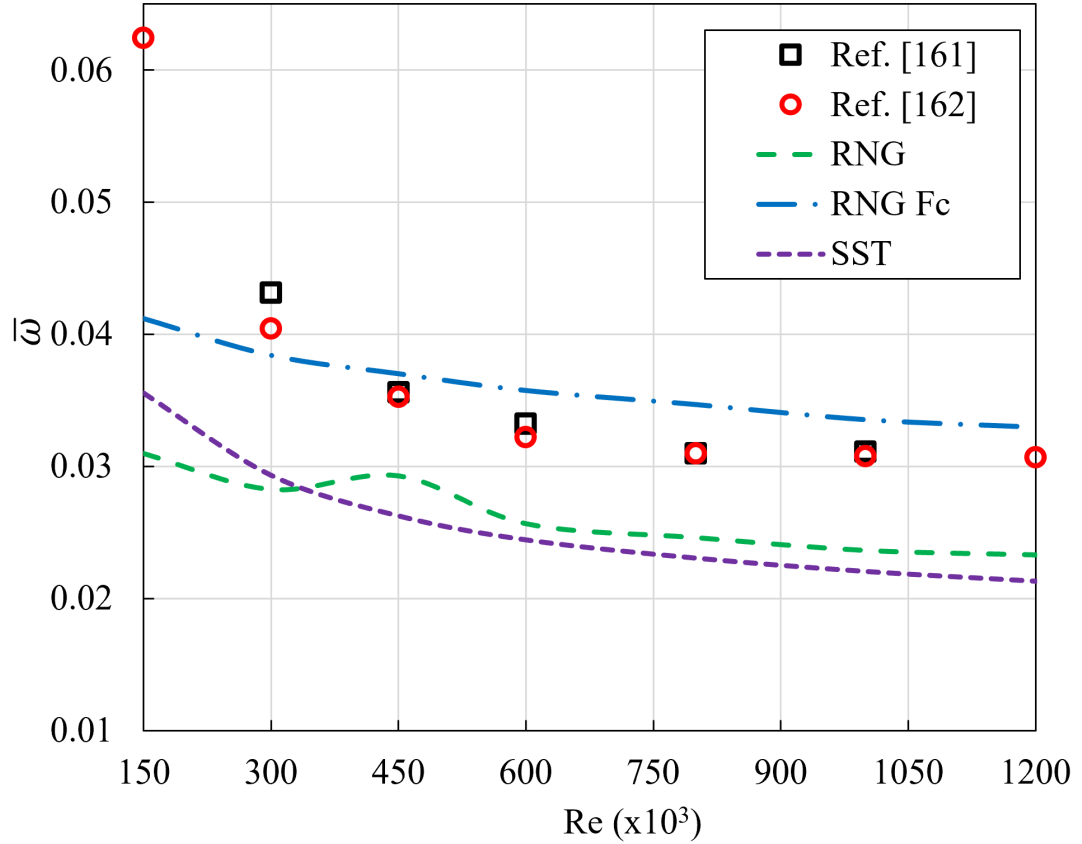


Figure 4.13 : Total pressure loss coefficient results of V103 cascade for different operating conditions.

In Figure 4.14, the Mach number contours on the midplane are compared with the results of the LES study conducted by Li et al. [163]. It is noteworthy that the RNG F_c model shows identical Mach number distribution around the blade profile with close resembles to the LES results. The results obtained for off-design conditions by varying the inlet Reynolds number are not satisfactory due to the increase in relative errors. This can be attributed to the fact that the AVDR ratios may differ for off-design conditions, while all analyses were conducted using the known single value of 1.06 for the design condition.

Considering all these evaluations, the results obtained using the RNG F_c model are highly satisfactory and require further investigation. It has been demonstrated that the RNG F_c model improves upon the Yakhot et al. [109] RNG model in numerical investigations of flows in axial compressor cascades. The RANS model presented as RNG F_c has been carefully fine-tuned and investigated for various compressor

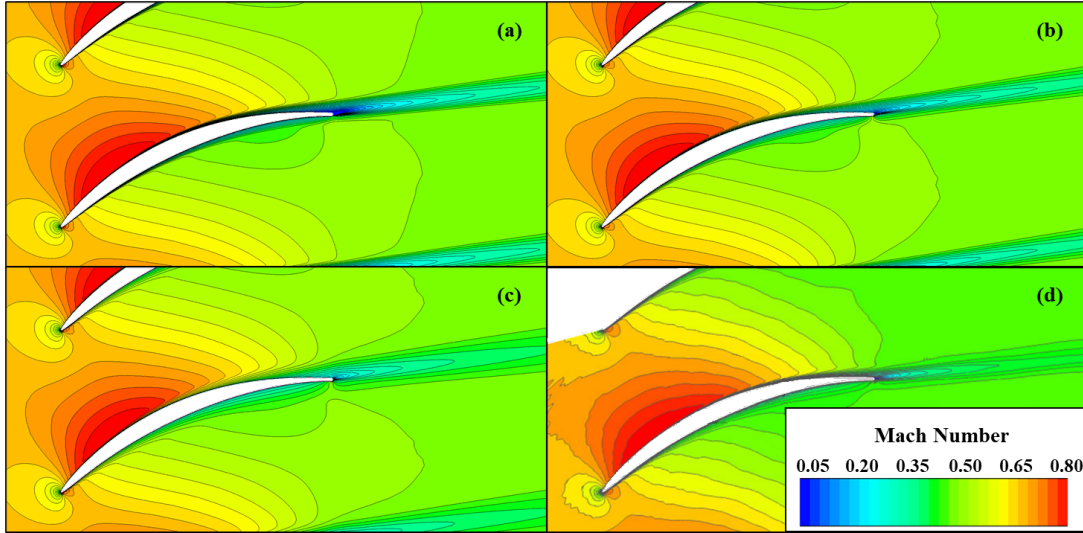


Figure 4.14 : Mach number distribution on midplane for V103 cascade at design condition (a) SST result (b) RNG result (c) RNG F_c result (d) LES result.

operating conditions. It is highly advantageous due to its minimal programming effort, numerical stability, consistent performance across computational grids with different near-wall resolutions, and its computationally cost-effective configuration, making it a promising approach when the local adjustment of the η_0 coefficient is carefully carried out.

4.4 Conclusion

In this study, a numerically stable and computationally cost-effective method has been proposed, specifically for axial compressor cascade analyses, by allowing the η_0 coefficient to take locally varying values within the flow domain based on a function. A new function is derived and added to the RNG $k - \varepsilon$ model, resulting in a new variant named Renormalization Group Function Modification (RNG F_c). By defining a new function resembling to the general expression of the low-Re interpolation formula in Yakhot et al.'s [109] RNG $k - \varepsilon$ turbulence model source term, the calculation of the total pressure loss in axial compressor cascade flows is precisely improved regardless of the grid topology, wall treatment and CFD solver since the developed function is dependent on turbulent viscosity, local Mach number, and von Kármán constant.

This model was tested on two different axial compressor cascades under a wide range of operating conditions, and the results were tested with both experimental data and widely used turbulence models. The RNG F_c model demonstrated superior performance in calculating total pressure loss and wake prediction for axial compressor

cascades compared to the standard RNG $k - \varepsilon$ model. This performance holds across grid structures with different near-wall resolutions, wall treatment approaches, and CFD solvers, achieving results comparable to those of the SST model.

While initial studies of the new model at varying Mach numbers show great promise and compare well to the SST model, further validation across diverse flow types, including unsteady flows and incompressible flows, is essential. To accelerate and diversify these investigations, the implementation code for the model modification is shared as open access for various CFD software. Researchers are encouraged to work on the model, evaluate its performance on complex problems, and contribute to its further development.





5. ENHANCING THE $k-\omega$ SHEAR STRESS TRANSPORT TURBULENCE MODEL THROUGH RENORMALIZATION GROUP THEORY FOR FLOWS SEPARATED FROM A CONTINUOUS SURFACE⁴

5.1 Introduction

Turbulent flow dynamics remains one of the most challenging part of engineering problems, characterized by its inherent chaotic nature and random fluctuations that produce a broad spectrum of energy scales through dissipation processes driven by turbulent eddies. While there is no universally accepted definition of a turbulent eddy, Kolmogorov's hypothesis asserts that large scale turbulent eddies extract energy from the mean flow and transfer this energy progressively to smaller structures through the cascade process until it is dissipated by viscosity at the smallest scales [14]. The cascade process defines the core mechanism of turbulence and underpins modern turbulence modeling. Over the past fifty years, enormous effort has been dedicated to improve the numerical accuracy of Computational Fluid Dynamics simulations for predicting complex flows. High-fidelity approaches such as Large Eddy Simulation and Direct Numerical Simulation can resolve a wide range of turbulent scales, but their prohibitive computational cost renders them impractical for most industrial applications. Hybrid methods such as RANS-LES coupling and wall-modeled LES provide promising alternatives but still rely heavily on turbulence modeling. Consequently, Reynolds-Averaged Navier-Stokes models continue to dominate both industrial and academic practice. Their enduring relevance is primarily due to their low computational cost and ability to deliver consistent results across a wide variety of turbulent flow types.

The evolution of RANS modeling has followed a trajectory shaped by both theoretical ambition and practical necessity. From Prandtl's early mixing-length hypothesis in the 1920s to the development of algebraic models like Baldwin-Lomax in the

⁴This chapter is based on the paper: Turutoglu, C., Yildizeli, A., Cadirci, S., Kuzuguden, C. and Erdem, D. (2026). Enhancing the $k-\omega$ shear stress transport turbulence model through renormalization group theory for flows separated from a continuous surface, *Physics of Fluids*, 38 (1), 015165.

1970s and transport-equation-based closures such as the Spalart-Allmaras (SA) model in the 1990s, each new generation has attempted to balance physical fidelity with computational feasibility [11, 12, 164, 165]. The two-equation family, particularly the $k - \varepsilon$ and $k - \omega$ models, has marked a milestone by offering general purpose formulations applicable to various flows [17, 25, 106].

Despite their widespread use, RANS models have significant limitations in addressing certain problems due to their founding assumptions. The assumption of isotropic turbulent fluctuations and the Boussinesq approximation of turbulent viscosity decrease the accuracy of the RANS models, especially in cases of large-scale rotational flows, where the anisotropy of turbulence is important, as velocity fluctuations are non-uniformly directed [8, 166]. The Boussinesq hypothesis, while mathematically convenient, treats Reynolds stresses as proportional to mean strain through a scalar eddy viscosity, effectively discarding the tensorial nature of turbulence. As a result, this assumption breaks down in flows with strong anisotropy, rotation, streamline curvature, or secondary motion, such as the strong flow separation regions encountered in aerospace, turbomachinery, automotive, and environmental engineering [167]. More advanced turbulence models, such as Reynolds Stress Models defined by considering the second moments, provide better treatment of the anisotropic nature of turbulence but at the cost of robustness and efficiency [70, 94, 168, 169].

Industrial demands are often time-critical, requiring accurate solutions within strict deadlines, while academic projects frequently operate under limited budgets. Although high-performance computing resources are increasingly available to well-funded corporations, they remain inaccessible to many researchers and smaller enterprises. Consequently, improving the effectiveness and reliability of RANS models remains extremely important. These models persist not because they are perfect, but because they represent the most viable option for bridging academic research and industrial design workflows.

As Duraisamy [16] has pointed out, the field is experiencing a “renaissance in turbulence modeling” (p. 11), driven by convergent trends such as machine learning, the proliferation of high-fidelity turbulent flow databases, and the ongoing demand for cost-efficient turbulence closures. This persistence is further emphasized by the NASA CFD Vision 2030 study, which underscored that RANS models will

remain indispensable well beyond 2030 [170]. Engineering design processes naturally progress from low to high-fidelity methods as resources allow. The continued use of inviscid solvers in preliminary design phases illustrates that even lower-order models remain relevant. Thus, reducing uncertainties and error margins in RANS predictions is just important today as it was in the past.

Throughout CFD history, several revolutionary RANS models have been introduced involving a variety of complexity from one equation Spalart-Allmaras to the Reynolds Stress Models. Among these, the Renormalization Group model initially generated high expectations due to its strong theoretical foundation. Developed by Yakhot and Orszag [107] and based on renormalization group theory pioneered by Wilson [147], the model employs a systematic scale-elimination procedure, i.e., small isotropic eddies are successively removed until only computationally resolvable scales remain while keeping the characteristics of small scales. This theoretically rigorous process modifies turbulence model constants in a mathematics-based manner, enabling the use of coarser grids without heavy reliance on empirical tuning. Notably, the RNG-based $k - \varepsilon$ model demonstrates superior performance over the standard $k - \varepsilon$ model in predicting reattachment, e.g., in backward-facing step flows, lending credibility to Yakhot and Orszag's assertion that the RNG formulation contains empirically determined parameters and may therefore be more universally applicable [30, 109]. However, despite its elegance, the RNG methodology has not been successfully embedded into RANS frameworks beyond the $k - \varepsilon$ model, leaving much of its potential untapped.

In contrast, the $k - \omega$ Shear Stress Transport model developed by Menter has emerged as the most widely adopted RANS model [110]. Built from a blending of the $k - \omega$ model of Wilcox [17] and the $k - \varepsilon$ model of Launder and Sharma [30], its Baseline formulation is further refined with limiting and blending functions to form the SST model. This model has been extensively validated and is widely regarded as one of the most reliable closures available today. Its wide adoption is evidenced by its status as the default turbulence model in major commercial and open source CFD software such as ANSYS Fluent, STAR-CCM+, and OpenFOAM, making it the first preference. Practitioners often value its robustness, particularly in external aerodynamics and turbomachinery. However, its performance deteriorates significantly in flows involving strong separation from continuous surfaces. Technical

reports such as those by Jakirlić [171] within the Advanced Turbulence Simulation for Aerodynamic Application Challenges (ATAAC) project and Manceau [172] in the ERCOFTAC Special Interest Group 15 (SIG-15) Workshop on Refined Turbulence Modelling have shown that conventional RANS methods misrepresent the physics of continuous-surface separation, i.e., the separation point is not fixed but oscillates intermittently over extended wall regions, producing zones of forward and reverse motion. Time-averaged flow fields in these regions are dominated by intermittently shed vortices, where turbulence is non-local and significantly stronger than in attached boundary layers. Benchmark studies, including the periodic hill flow, have consistently shown that the SST model excessively suppresses turbulence activity, leading to over-prediction of recirculation zones. Researchers have attempted to address these deficiencies through various empirical modifications and calibration approaches by incorporating partial differential equation (PDE) frameworks [158, 173–177]. However, such methods often require extensive tuning across different flow regimes, leading to high uncertainty and limited generality.

Machine learning has recently been explored as an alternative path, with efforts ranging from field inversion and machine learning (FIML) to physics-informed neural networks (PINN). These methods show promise but face issues of interpretability, generalization, and reliance on large, high-quality datasets. In contrast, incremental physics-based modifications remain attractive because they can enhance prediction accuracy without compromising transparency or industrial usability.

The present study builds upon two key insights. First, Yakhot and Orszag’s argument that RNG model is based on a theory and eliminates the need for adjustable parameters offers a path towards a more universal modeling approach [107]. Secondly, Menter’s observation that even minor adjustments to turbulence-model constants can drastically affect predictions highlights the need for dynamic, rather than static, tuning [110]. Initial attempts to statically optimize SST coefficients for flow over curved backward-facing step benchmark problem proved insufficient. This prompted a strategic shift toward a more universally applicable approach, i.e., embedding RNG corrections selectively within the SST framework.

The primary aim of this research is to improve predictions of separation and reattachment on continuous surfaces with minimal additional computational cost while

keeping accuracy and numerical stability. Inspired by the superior reattachment capability of the RNG-based $k - \varepsilon$ model, this work introduces an RNG-enhanced SST $k - \omega$ model, enabled through a separation-detection function that targets flow regions where conventional RANS methods often fail. To ensure rigorous evaluation, the proposed model is benchmarked against LES and experimental datasets across canonical cases, including the backward-facing step, periodic hills, and a curved backward-facing ramp. Examined test cases are selected to cover diversity of flow separation and reattachment. For instance, in backward-facing step problem, separation occurs at the step location independently from the flow conditions, whereas pressure gradient and topology determines the flow separation over periodic hills and curved backward-facing ramp. Furthermore, flow over periodic hills and curved backward-facing step problems differ from each other as incoming flow properties are different. Additionally, comparisons at different Reynolds numbers for each flow type expanded the portfolio of wake structures examined.

A particular emphasis is placed on accurate boundary-condition treatment of turbulent quantities, as inflow specifications (e.g., length scale, viscosity ratio, eddy dissipation, or eddy frequency) are known to strongly influence separation prediction and reattachment [178–180]. To minimize arbitrariness, the dissipation profile at the inflow is computed from turbulence kinetic energy budgets derived from LES data [181, 182]. Comparisons with alternative formulations confirmed that the boundary conditions are correctly specified, ensuring errors could be attributed to the model itself rather than to setup inconsistencies.

In this thesis, a dedicated experimental campaign is conducted in an open-channel wind tunnel using a modified ramp geometry. This configuration differs from existing ERCOFTAC benchmarks by employing a higher freestream TI, a different Reynolds number, and the absence of confinement effects typical of closed-circuit tunnels. This deliberate deviation ensured that the resulting dataset would extend the parameter space available to the community, offering a more challenging benchmark for model validation. By situating the campaign in an open-channel facility, the study reduces wall-induced biases and enables a cleaner interpretation of large-scale flow separation dynamics. The results from this study demonstrate that the coupled RNG-SST framework not only delivers satisfactory computational performance but also shows considerable promise for a broader range of engineering applications.

5.2 Formulation of RNG-SST Model

The fundamental concept of the RNG-SST coupling is to retain the proven strengths of the SST model while remedying its known deficiencies in strongly separated flows by leveraging the advantages of the RNG theory. A prerequisite is a thorough understanding of the respective benefits of each model. Menter's SST model is conceived to merge the superior near-wall computational performance of the $k - \omega$ model with the robustness of the $k - \varepsilon$ model in the freestream. Wilcox's original $k - \omega$ equations are given in equations 5.1 and 5.2.

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \bar{v}_j k) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_{k1}} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \beta^* \rho k \omega \quad (5.1)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \bar{v}_j \omega) = \alpha_1 P_k \frac{\omega}{k} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_{\omega 1}} \right) \frac{\partial \omega}{\partial x_j} \right] - \beta_1 \rho \omega^2 \quad (5.2)$$

The relationship between the turbulence dissipation frequency (ω) and the turbulence dissipation rate (ε) is $\varepsilon = \beta^* \omega k$, where k is the turbulence kinetic energy. Utilizing this relation, Menter transformed the Jones and Launder $k - \varepsilon$ equations into a $k - \omega$ formulation, simultaneously redefining the model coefficients. It is noteworthy that the coefficients in the contemporary standard $k - \varepsilon$ model are those established by Launder and Sharma [30]. The transformed $k - \varepsilon$ equations are presented in equation 5.3 and 5.4, with the model coefficients defined as $\beta^* = 0.09$, $\beta_1 = 0.075$, $\beta_2 = 0.0828$, $\sigma_{k1} = 2$, $\sigma_{k2} = 1.176$, $\sigma_{\omega 1} = 2$, $\sigma_{\omega 2} = 1.168$, $\alpha_1 = 5/9$, $\alpha_2 = 0.44$, and $a_1 = 0.31$. The production term, P_k , in the equation 5.3 is given in equation 5.5.

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \bar{v}_j k) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_{k2}} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \beta^* \rho k \omega \quad (5.3)$$

$$\begin{aligned} \frac{\partial(\rho \omega)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \bar{v}_j \omega) &= 2\rho \frac{1}{\sigma_{\omega 2} \omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \\ &+ \alpha_2 P_k \frac{\omega}{k} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_{\omega 2}} \right) \frac{\partial \omega}{\partial x_j} \right] - \beta_2 \rho \omega^2 \end{aligned} \quad (5.4)$$

$$P_k = \mu_t \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) \frac{\partial \bar{v}_i}{\partial x_j} - \frac{2}{3} \frac{\partial \bar{v}_k}{\partial x_k} \left(\mu_t \frac{\partial \bar{v}_k}{\partial x_k} + \rho k \right) \quad (5.5)$$

The function F_1 is a blending function designed to separate the near-wall region from the freestream. It takes a value of one near the wall and decays to zero away from it. By blending the model equations using this function, a new set of equations, named the Baseline model, is formulated. The F_1 blending function is given in equation 5.6.

$$F_1 = \tanh(arg_1^4) \quad (5.6)$$

where

$$arg_1 = \min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right), \frac{4\rho k}{CD_{kw} \sigma_{\omega 2} y^2} \right] \quad (5.7)$$

$$CD_{kw} = \max \left(2\rho \frac{1}{\sigma_{\omega 2} \omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 1 \times 10^{-10} \right) \quad (5.8)$$

However, Menter [110] emphasized that a key difference between eddy-viscosity and Reynolds-stress models is the latter's account of turbulent shear stress transport. Consequently, in the SST model, he redefined the eddy viscosity using Bradshaw's assumption that the shear stress is proportional to the turbulence kinetic energy in boundary layers and formulated it as shown in equation 5.9.

$$\mu_t = \rho \frac{a_1 k}{\max(a_1 \omega, SF_2)} \quad (5.9)$$

The function F_2 , defined in equation 5.10, is employed to distinguish between boundary layer flows and free shear layers. Calculations proceed according to boundary layer flow when F_2 is equal to one, and zero according to free shear layer treatment.

$$F_2 = \tanh(arg_2^2) \quad (5.10)$$

where

$$arg_2 = \max \left(\frac{2\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{y^2 \omega} \right) \quad (5.11)$$

The final steady-state k and ω equations of the SST model, obtained through the use of blending functions, are presented in equation 5.12 and 5.13. In the version of the SST model examined in this study, the production term corresponds to the exact production formulation as shown in equation 5.5. However, since this term is neglected in the momentum equation, the model is referred to as SST-2003e [143, 183].

$$\underbrace{\frac{\partial}{\partial x_j} (\rho \bar{v}_j k)}_{\text{Convection}} = \underbrace{\frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_{k3}} \right) \frac{\partial k}{\partial x_j} \right]}_{\text{Total Diffusion}} + \underbrace{P_k}_{\text{Production}} - \underbrace{\beta^* \rho k \omega}_{\text{Dissipation}} \quad (5.12)$$

$$\begin{aligned} \frac{\partial}{\partial x_j} (\rho \bar{v}_j \omega) &= (1 - F_1) 2\rho \frac{1}{\sigma_{\omega 2} \omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} + \\ &\alpha_3 P_k \frac{\omega}{k} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_{\omega 3}} \right) \frac{\partial \omega}{\partial x_j} \right] - \beta_3 \rho \omega^2 \end{aligned} \quad (5.13)$$

Coefficients of the model can be derived using equation 5.14, where ϕ is the corresponding coefficient.

$$\phi_3 = F_1 \phi_1 + (1 - F_1) \phi_2 \quad (5.14)$$

The SST model has enabled highly successful simulations in numerous studies, particularly in aerodynamics. A long-standing and well-documented weakness, highlighted by Johnson et al. [184] and reiterated by Menter [143] is that the RANS models underpredict the magnitude of the turbulent stresses in the shear layer, which is the reason for the incorrect reattachment prediction. To address separation and reattachment shortcomings in SST, Langtry and Menter developed the $\gamma - Re_\theta$ transition model [173]. This model requires solving two additional transport equations for intermittency (γ) and the momentum-thickness Reynolds number (Re_θ). It relies on empirical correlations derived from extensive experimental data. However, correlation-based equations tend to lose efficacy when applied to problems, boundary conditions, or Reynolds numbers outside their calibration range. At this juncture, the author elected to investigate the effects of integrating the RNG model into the SST equations instead of employing correlation-based methods.

Renormalization group theory provides a mathematical framework for examining a system's effective behavior across different energy scales. It systematically eliminates small scales while preserving their characteristic effects on larger scales. This procedure exhibits remarkable synergy with the cascade theory of turbulence as described by Kolmogorov [14]. Although the integration of RNG theory into the $k - \epsilon$ model by Yakhot et al. [109] yielded improved performance for some problems, it failed to deliver generally enhanced results for many flow types.

The RNG $k - \epsilon$ model differs from the standard Jones and Launder model primarily through the subtraction of the function f_η given in equation 5.15, multiplied by $(P_k \epsilon / k)$ in the turbulence eddy dissipation equation. Calculation of the η variable is given in equation 5.16.

$$f_\eta = \frac{\eta \left(1 - \frac{\eta}{\eta_0}\right)}{1 + \beta_{RNG} \eta^3} \quad (5.15)$$

$$\eta = \sqrt{\frac{P_k}{\rho C_\mu \epsilon}} \quad (5.16)$$

Theoretically, this modification can be directly integrated into the transformed ϵ equation of the SST framework by multiplying it by $1 - F_1$. However, this would allow the f_η function to also affect the ϵ equation solved in the freestream. To confine the function's influence on the boundary layer and prevent interference with the near-wall (low Reynolds number) ω equation, the decision is made to multiply it not by $1 - F_1$,

but by a function based on the turbulent Reynolds number, analogous to the $F_{Reattach}$ function used by Langtry and Menter [173]. Thus, the model presented in this study is formulated by adding the source term shown in equation 5.17 to the ω equation of the SST model. The function f_η is computed using the formulation given in equation 5.18.

$$\text{Source Term} = -f_\eta(1 - F_{Reattach})\frac{\omega}{k}P_k \quad (5.17)$$

$$f_\eta = \frac{\eta \left(1 - \frac{\eta}{\eta_0}\right)}{1 + \eta^3/\omega} \quad (5.18)$$

$$\eta = \sqrt{\frac{P_k}{\rho\beta^*\epsilon}} \quad (5.19)$$

$$F_{Reattach} = e^{-\left(\frac{\rho k}{\omega\mu^{20}}\right)^4} \quad (5.20)$$

A key distinction from the original RNG model by Yakhot et al. [109] in this formulation is the replacement of the constant C_μ in the expression for η with the SST model's coefficient β^* . Since both C_μ and β^* share the same value (i.e., 0.09) in the present implementation, this replacement can be regarded as a notational modification introduced by Menter [110]. The last and major difference from the original RNG model is the β_{RNG} coefficient. Yakhot et al. [109] defined it as a constant value of 0.012. In the present study, β_{RNG} gets value from $1/\omega$ by non-dimensionalizing it with 1 Hz. The rationale for this modification stems from an empirical observation made during model development. While calibrating the model, values other than the original value for the β_{RNG} coefficient were also examined. In the investigations conducted for β_{RNG} , it was determined that there is a similarity between the value that best predicts the reattachment position and the inverse of ω applied for the inflow boundary condition. Since the value of ω boundary condition varies for each case, it was considered that using this coefficient as a variable rather than a constant value would provide a broader application area for the model.

5.3 Methodology

5.3.1 Numerical model

The turbulence model presented in this study is evaluated across multiple test cases at various Reynolds numbers. Meticulous care is taken to minimize all errors extrinsic to the model itself. Consequently, the same finite-volume solver, Ansys[®]-CFX [52],

is employed for all investigated cases. While the geometry, flow type, computational grid, and boundary conditions were specific to each problem, all cases shared common feature i.e. the flow regimes within the incompressible limit since there are no thermal effects and relatively low velocity fields, and the working fluid is air with constant thermophysical properties. The computational grids and boundary conditions, which vary depending on the case, are detailed the following section. The corresponding RANS equations solved in numerical investigations are provided in equation 5.21 and 5.22.

$$\frac{\partial}{\partial x_i} (\rho \bar{v}_i) = 0 \quad (5.21)$$

$$\frac{\partial}{\partial x_j} (\rho \bar{v}_i \bar{v}_j) = -\frac{\partial p'}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu_{eff} \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) \right] \quad (5.22)$$

In equation 5.22, the modified pressure, p' , is assumed to be the static pressure, p . The effective viscosity, μ_{eff} , is the sum of the molecular viscosity and the turbulent viscosity. The turbulent viscosity is derived from the eddy diffusivity hypothesis as shown in equation 5.9, which posits a linear relationship between Reynolds scalar transport and the mean scalar gradient. All of the governing equations are integrated over each control volume and discretized using second-order Gaussian integration with linear interpolation (achieved by setting the blending factor to one in CFX) to mitigate numerical diffusion. Finite-element shape functions are utilized to approximate integration point values for mass flux, pressure, and diffusion terms. For turbulent fluxes, a first-to-second order blending scheme is used, which approximates a second-order upwind discretization taking into account the boundedness principles of Barth and Jespersen [101] particularly for structured grids. The resulting system of equations is solved iteratively using the COUPLED algorithm presented by Rhie and Chow [154] and enhanced by Majumdar [185] until the Root-Mean-Square of residuals converges to 1×10^{-6} for each equation. The pseudo time-step size is determined based on the characteristic length and mean flow velocity of each corresponding problem. Further computational details are available in Ref. [52]. A low-Reynolds number approach is employed at all no-slip walls for the turbulence kinetic energy and turbulence eddy frequency boundary condition, ensuring a y^+ value below unity. Zero-gradient boundary condition is applied at the outlet surfaces to prevent influencing the upstream flow field. The boundaries of the computational domain are set to be sufficiently far from the region of interest.

5.3.2 Description of test cases

The investigation is focused on three canonical cases of separated flow: the curved backward-facing step (CBFS), the backward-facing step (BFS), and periodic hills. These problems are characterized by strong flow separation with combined adverse and favorable pressure gradients and are known as problems under which the SST model under-predicts turbulent stresses in shear layers. Given the documented sensitivity of such flows to boundary conditions, by which results can be easily manipulated, rigorous measures are employed to possess the pure model effect [186]. Numerical errors from discretization and solver algorithms are minimized, and well-established test cases from NASA and ERCOFTAC are utilized to suppress boundary condition biases. The selection of these specific cases is further justified by the availability of independent studies conducted by other researchers using SST model, which facilitates robust method verification and cross-comparison. Details of investigated benchmark problems are tabulated in Table 5.1.

Table 5.1 : Details of the benchmark cases.

Case	Step Angle [$^{\circ}$]	Step Height [mm]	Re_H	Source
BFS-a	90	12	47000	Ref. [187]
BFS-b	90	12	64000	Ref. [187]
BFS-c	20	12	47000	Ref. [187]
BFS-d	20	12	64000	Ref. [187]
CBFS	N/A	31.5	13700	Ref. [182]
Periodic Hills-a	N/A	28	10595	Ref. [188]
Periodic Hills-b	N/A	28	37000	Ref. [189]
Experiment	N/A	76.6	48570	—

The CBFS case is selected as the reference case, sourced from the NASA Langley Research Center’s Turbulence Modeling Resource [190], which provides data from both experimental campaign and LES simulation. This case is also included in the ERCOFTAC KBwiki database under the index of Underlying Flow Regime (UFR) as UFR 3-31, studied by Lardeau [191] using various RANS models, including SST. The CBFS case involves a turbulent boundary layer separating from a curved surface at the Reynolds number of 13700 based on 0.0315 m step height and inflow velocity. Mean flow quantities (u, v) and turbulence kinetic energy profile obtained from LES data are directly imposed as the inflow boundary condition. The specification of the ω or length scale (l) boundary condition is investigated through different methodologies.

The periodic hills case, a popular benchmark case used in RANS model comparisons, was chosen to demonstrate model performance across different turbulent flow structures (separation, reattachment, and recirculation zones) triggered by the favorable and adverse pressure gradients following the sudden expansion near the hill crest. NASA and ERCOFTAC databases provide access to both experimental and LES results. Furthermore, the ATAAC workshop report includes results from the application of various RANS models [171]. An important distinguishing feature of this problem is the streamwise periodicity defined at the inflow and outflow boundaries. Due to streamwise periodic boundary condition, k and ω specification at the inlet plane is not required. Thus, the effect of imposed boundary condition at the inflow boundary is eliminated. Instead, total mass flow rate is set to achieve the corresponding Reynolds number, calculated using the mean velocity over the crest of the first hill.

The computational meshes for all cases were constructed to ensure high resolution and low numerical dissipation. A symmetry boundary condition is applied on the spanwise surfaces for all benchmark studies, as the analyses are conducted in two dimensions. Across all simulations, a near-wall $y^+ < 1$ was strictly maintained, with a maximum cell growth rate of 1.05 in the wall-normal direction. To ensure mesh quality, an orthogonality check was performed using the methodology of Tan et al. [192]. The analysis confirmed that the orthogonality metrics of the employed grids do not contribute significantly to the numerical error. Consequently, the detailed orthogonality results are not presented here for brevity. For the CBFS case, the LES-based mesh consists of 768×160 cells, resulting in approximately 122,000 cells, with particular attention paid to ensuring that there are more than 280 cells within the separated region to adequately resolve the recirculation zone [182]. The periodic hills cases are investigated by using a grid consisting of 254×128 cells, totaling 32,512 cells with a similar cell count to that used in LES simulations [193]. Figure 5.1 represents the corresponding computational grids used in this study for the CBFS and periodic hills investigations.

Flow separation from a curved surface is substantially more intricate and includes higher unsteadiness compared to a sharp-edged step such as 90° BFS, primarily due to its strong dependence on the state of the incoming boundary layer. Statistically, this unsteadiness correlates with exceptionally high levels of production of TKE and shear stress observed near the separation point. Furthermore, the separation zone is notably

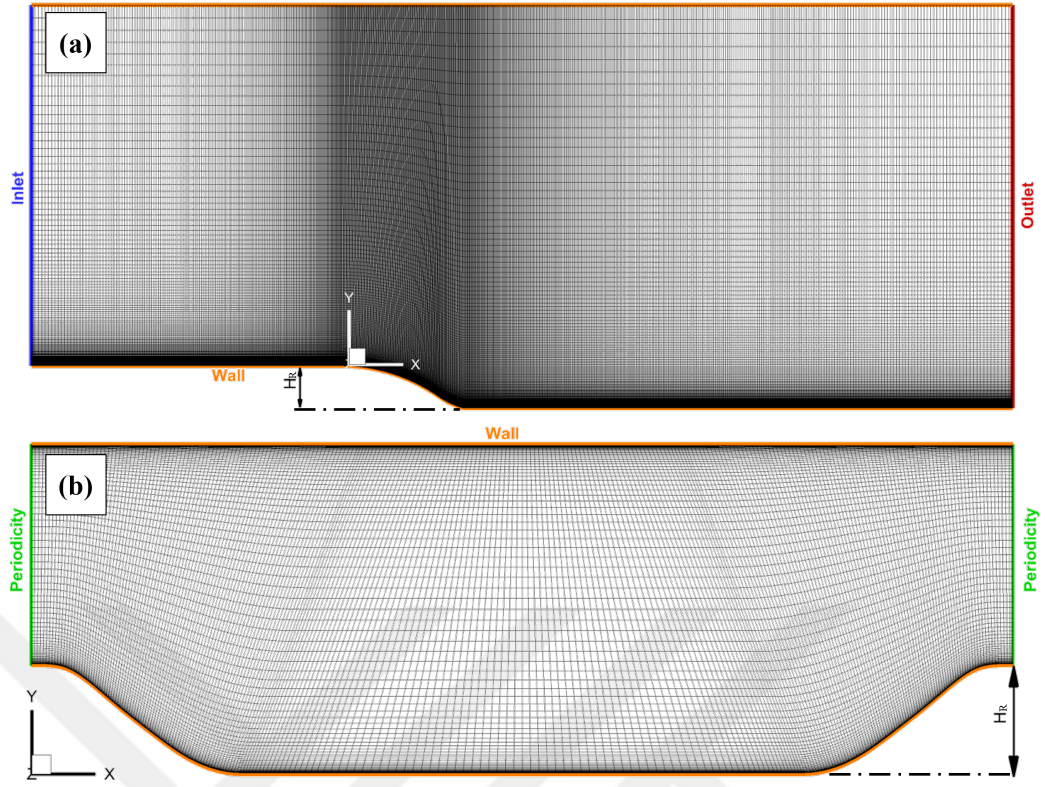


Figure 5.1 : Computational grids used in this study for (a) CBFS and (b) periodic hills.

thin, tapering sharply toward the separation line, which highlights the dominant influence of near-wall dynamics on the separation behavior. For the assessment of reattachment prediction, the BFS case is used as an additional benchmark, a case also used by Yakhot et al. [109] for the same purpose. The experimental campaign performed by Ruck and Makiola [187] is considered, which examines the confined flow over a BFS for a variety of step angles and Reynolds numbers. From the numerous configurations in the ERCOFTAC Classic Collection Database, four distinct cases are selected for investigation. Experimentally measured mean flow quantities and turbulence kinetic energy profiles of Ruck and Makiola's [187] experiment are imposed as the inflow boundary conditions. The ω value is calculated and applied using the relation $5U_\infty/L$, as proposed by Spalart and Rumsey [179], where the characteristic length (L) is taken as the step height. The mesh for the BFS cases follow the guidelines established by Yildizeli and Cadirci [127] and visualized in Figure 5.2. For all benchmark cases investigated in this study, comprehensive mesh independency studies had been previously done and validated by researchers [127, 191, 193]. However, a detailed examination of the generated mesh used in CFD analyses for the experimentally investigated ramp model is documented with its results.

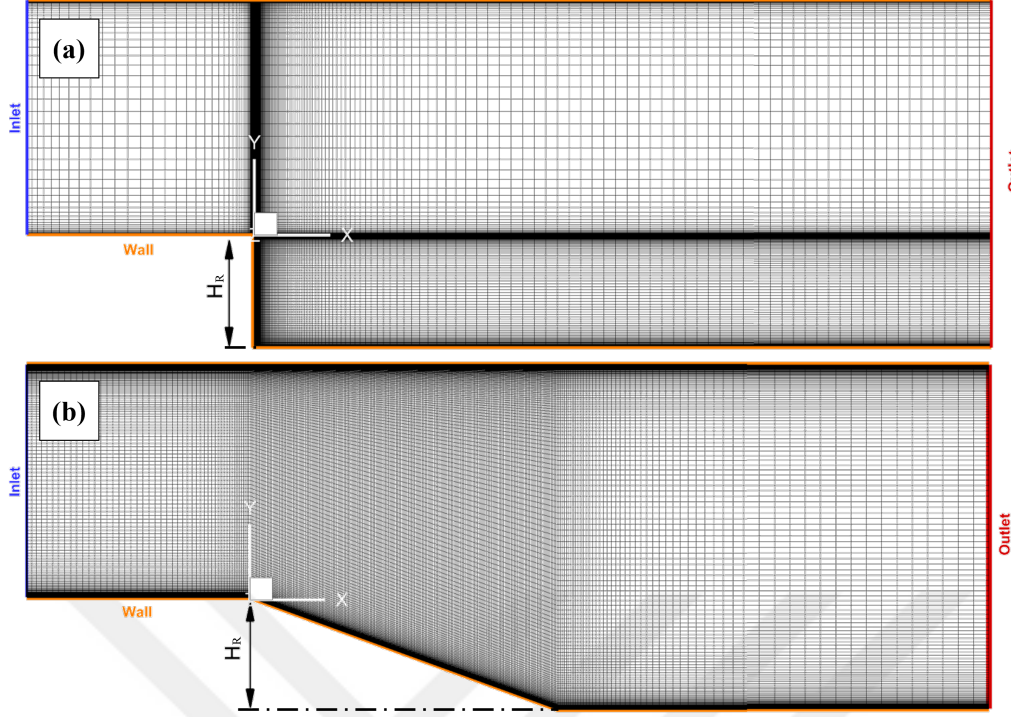


Figure 5.2 : Computational grids used for (a) 90° step angle BFS and for (b) 20° step angle BFS.

5.3.3 Experimental campaign

Experiments were performed in an open-circuit suction-type subsonic wind tunnel with an $80 \text{ cm} \times 80 \text{ cm}$ open test section located in the Trisonic Research Laboratory of Istanbul Technical University. The wind tunnel is driven by an axial fan, and the flow speed in the test section is regulated via a motor controller adjusting the fan's rotational speed. A two-dimensional turbulent boundary layer is developed over a curved backward-facing step positioned at the bottom of the test section. The ramp height is $H_R = 76.6 \text{ mm}$, and the curved surface of the ramp, valid up to the position $x/H_R = 3.33$, formed by a circular arc, has a radius of $6.3H_R$. Test section height is $10.44H_R$, and the model has a spanwise extent with the same length. The upstream edge of the ramp is located $7H_R$ downstream from the test section entrance, and a tripping wire is installed at the entrance to trigger transition and increase the boundary layer thickness. The boundary layer initially developed over a flat plate before encountering the backward-facing ramp.

Surface static pressure distributions are measured using a total of 54 pressure taps arranged in two parallel, closely spaced rows along the model centerline. An additional 27 taps are placed on the flat plate downstream of the ramp to investigate the

reattachment region. Pressure data is acquired using an ESP miniature 32-channel electronic pressure scanner with a measurement range of ± 2500 Pa and an accuracy of $\pm 0.1\%$ FS. The scanner is installed inside the model to minimize tubing length. Measurements are taken at a sampling frequency of 64 Hz over a period of 20 s, and the average of 10 data sets is used for analysis. Pressure data is normalized with the inlet dynamic pressure using equation 5.23, where P_0 denotes the static pressure at the inlet plane and ρ represents the fluid's density.

$$C_p = \frac{P - P_0}{\frac{1}{2}\rho U_{Ref}^2} \quad (5.23)$$

Velocity measurement is performed using a DANTEC 90C10 constant-temperature anemometer and boundary layer probes (55P15). The analog signal is digitized using a 16-bit A/D board with 16 input channels. Data is sampled at 8 kHz, and 16384 samples are recorded at each measurement point. A three-directional traverse mechanism with a linear motion accuracy of 0.016 mm positioned the probe at predefined locations. Boundary layer profiles were measured at multiple downstream stations along the ramp and the downstream flat plate to capture separation and reattachment behavior. The turbulent boundary layer is analyzed using both pressure and velocity measurements. The boundary layer thickness at the ramp upstream edge ($x/H_R = 0$) is $\delta_0 \approx 0.26H_R$. The shape factor at this location is $H \approx 1.24$, and the Reynolds number based on momentum thickness is $Re_\theta = 1430$. The flow accelerates when it approaches the ramp, then decelerates under the effect of an adverse pressure gradient (APG).

5.4 Results and Discussion

In this study, the turbulence model was developed by integrating Yakhot et al. [109] RNG formulation into Menter's [143] SST model to enhance the flow-field prediction capability of the SST for problems involving strongly separated flows. The results obtained with the proposed model are evaluated against high-fidelity benchmark data, such as LES and experimental measurements. The advantages and disadvantages of applying RNG-based enhancements to the SST model are then rigorously analyzed. Investigations are categorized and discussed under separate sections according to their flow physics characteristics. The investigated cases are shown in Table 5.1.

The curved backward-facing step configuration was selected as the primary case for detailed evaluation, as it best represents the type of separated flow from a continuous

surface. The obtained results were examined through both mean and averaged time-dependent flow properties, including velocity profiles, turbulence kinetic energy, and the budget of the k equation, as well as pressure and wall shear stress distributions. The influence of inflow boundary condition for this type of flows was also examined to accurately evaluate the model's performance.

This study further investigates multiple cases to evaluate how the flow type affects the model's performance. To determine whether results remain consistent under varying conditions, tests were performed on a curved backward-facing step with distinct geometric and flow characteristics. The agreement between the SST-2003e and presented model used in CFD simulations and experimental measurements was evaluated through qualitative and quantitative comparisons, allowing a comprehensive review.

5.4.1 Flow over curved backward-facing step

The ramp geometry is taken from Song and Eaton's [194] investigation, but an increased step height to generate a larger recirculation zone and developed collaboratively, with experimental investigations conducted by Zhang and Zhong [195] and large-eddy simulations performed by Bentaleb et al. [182]. For this specific case, previous studies conducted by Lardeau [191] using various RANS turbulence models are also considered for cross-validation. The results of these investigations are available in the ERCOFTAC KBwiki UFR database [191]. The measured separation length error associated with the SST model is found to be relatively higher compared to other common RANS models, making it a particularly valuable benchmark for model assessment. When defining the boundary conditions of the problem, mean flow and turbulence kinetic energy profiles are taken from the LES data of Bentaleb et al. [182]. However, as emphasized by Lardeau [191], a careful evaluation is required when specifying the length-scale boundary condition (or its corresponding variable for closure equation such as ω or ϵ). Therefore, a comparative study is conducted using the methods proposed in the literature to determine the most appropriate boundary condition. The results of this detailed investigation are presented in the following section.

5.4.1.1 Assessment of inflow boundary conditions

The application of appropriate boundary conditions constitutes a pivotal aspect of any CFD analysis, possessing equal significance for ensuring sufficient and high quality computation with adequate discretization schemes. For the CBFS problem investigated in this study, the aim was to minimize uncertainties originating from boundary conditions rather than from the modeling or mesh generation process. Since the geometry of the ramp model and computational grid are identical to those in the benchmark case, possessing the mesh independency is omitted.

No-slip wall boundary condition is implemented to the channel walls, the computational domain is assumed to be periodic along the spanwise extent, and the outlet boundary is placed far enough downstream to allow the use of a zero-gradient boundary condition. Therefore, uncertainties associated with these surfaces are effectively eliminated. However, as emphasized by Lardeau [191], potential uncertainties may still arise from the inflow boundary conditions.

At the inlet plane, the streamwise and wall-normal velocity components were imposed as velocity profile obtained from the benchmark LES study. For RANS simulations, additional boundary condition is required for the k and for its closure equation, i.e. ω . The TKE profile, derived using the fluctuating velocity components from the LES data, is calculated according to the definition given in equation 5.24, and applied as an inlet profile.

$$k = \frac{1}{2} (u_{RMS}'^2 + v_{RMS}'^2 + w_{RMS}'^2) \quad (5.24)$$

Because the RANS methodology is based on the eddy viscosity modeling approach, it models all turbulence scales. Consequently, it cannot directly resolve instantaneous, time-dependent turbulent structures across the full spectrum of motion. Thus, one more boundary condition is needed to implement turbulent quantities computed with LES at the inflow boundary. It is not possible to directly extract this condition from the results of a LES simulation. On the other hand, various researchers have proposed different approaches for defining the boundary condition required for the closure equation. Lardeau [191] suggested performing a preliminary computation to determine it. Although this approach is reasonable, conducting such an additional analysis for the entire model would be computationally expensive.

Davidson [180] argues that inlet boundary condition for closure equation should be

regarded as forcing conditions, primarily intended to trigger the turbulence equations rather than to represent physically accurate quantities. Accordingly, the exact physical correctness of the length scale is not critical; instead, the boundary condition should be chosen based on its effectiveness to initiate turbulence. Based on this suggestion, the author decided to apply different closure values at the inlet boundary and compare the resulting flow fields. To decide on the most suitable boundary condition, ten different CFD cases with six different inlet boundary conditions are configured for comparison and presented in Table 5.2.

Table 5.2 : Inflow boundary conditions for turbulence eddy frequency equation of SST model.

Case	Boundary Type	Method
BC 1	ω	$0.1U_{Ref.}/H_R$
BC 2	ω	$10U_{Ref.}/L$
BC 3	ω	$5U_{Ref.}/H_R$
BC 4	ω	$10U_{Ref.}/H_R$
BC 5	l	$0.1\delta_0$
BC 6	l	$0.2\delta_0$
BC 7	ϵ	<i>Profile</i>
BC 8	ϵ	<i>Profile(200%)</i>
BC 9	ϵ	<i>Profile(67%)</i>
BC 10	ϵ	<i>Profile(50%)</i>

The first approach, labeled BC 1 in Table 5.2, employs a constant ω value obtained using the expression $0.1U_{Ref.}/H_R$, which is applied by Lardeau in the investigation with the SST model [191]. The second configuration, BC 2, is derived from Menter's [110] proposed formulation. However, the original correlation was developed for external aerodynamic analyses. Therefore, instead of using the computational domain extent as the characteristic length (L) as suggested by Menter [110], the height of the inflow boundary (L_e) is used here, since the CBFS problem represents an internal flow. Similarly, BC 4 is generated using the same formulation but with the ramp height defined as the characteristic length. The third approach is based on the methodology suggested by Spalart and Rumsey [179], They compared several turbulence models and recommended that, for external aerodynamic flows, the characteristic length used to compute ω boundary condition in the SST model should correspond to the geometric property of the aerodynamic body of interest (e.g., the chord length of an airfoil). The fourth and final main approach follows the relation conducted by McConnell et al.

[196], in which the turbulent inlet length scale is defined as 10% of the boundary-layer thickness (δ_0) at the inlet surface. This case is referred to BC 5 in Table 5.2. In their comparative study of RANS and DES simulations for the same CBFS problem, McConnell et al. [196] concluded that this boundary condition provided the most suitable length scale definition. An additional configuration, BC 6, is generated using the same principle but by defining the constant length scale as 20% of δ_0 , accounting for boundary-layer development on both the upper and lower walls. For the rest of the cases, dissipation term values from the turbulence kinetic energy budget computed by Bentaleb et al. [182] and shared on the ERCOFTAC database are used to calculate turbulent length scale profile according to equation 5.25. Four additional configurations are created based on this methodology: the exact dissipation profile, twice its magnitude, 67% of the dissipation, and 50% of the dissipation.

$$l = \frac{k^2}{\varepsilon} \quad (5.25)$$

According to Bentaleb et al. [182], the dissipation rate reported in their turbulence budget balance is the sum of the other turbulence kinetic energy equation terms, while the explicitly resolved dissipation extracted from the LES accounts for approximately 50-70% of the total. Hence, the present study examined different levels of dissipation to capture this range. Note that in some references the length scale is derived using the C_μ coefficient but CFX does not use it for the calculation at the inlet boundary [52].

Figure 5.3 presents the distributions of the pressure coefficient (C_p) and the skin-friction coefficient (C_f) for all cases, compared with the LES results. Each boundary condition yields noticeably different flow field predictions. Table 5.3 summarizes the predicted separation and reattachment points and their relative errors with respect to the LES data, as well as the RMS errors of the C_p distribution computed at 40 streamwise positions from $x/H_R = -5$ to $x/H_R = 14$ ($x/H_R = -0.5$ to 5.5 in 0.25 increments to capture the main effects of the ramp) to compare C_p distribution quantitatively.

Among the tested conditions, BC 1 gives the smallest error in predicting the reattachment point (x_{Reatt}/H_R) and hence the separation length (L_{Sep}/H_R), whereas BC 3, BC 6, and BC 7 achieve the most accurate estimation for the separation point (x_{Sep}/H_R). Determining which boundary condition aligns better with the LES data is, therefore non-trivial. One could prioritize either the smallest overall error in C_p and

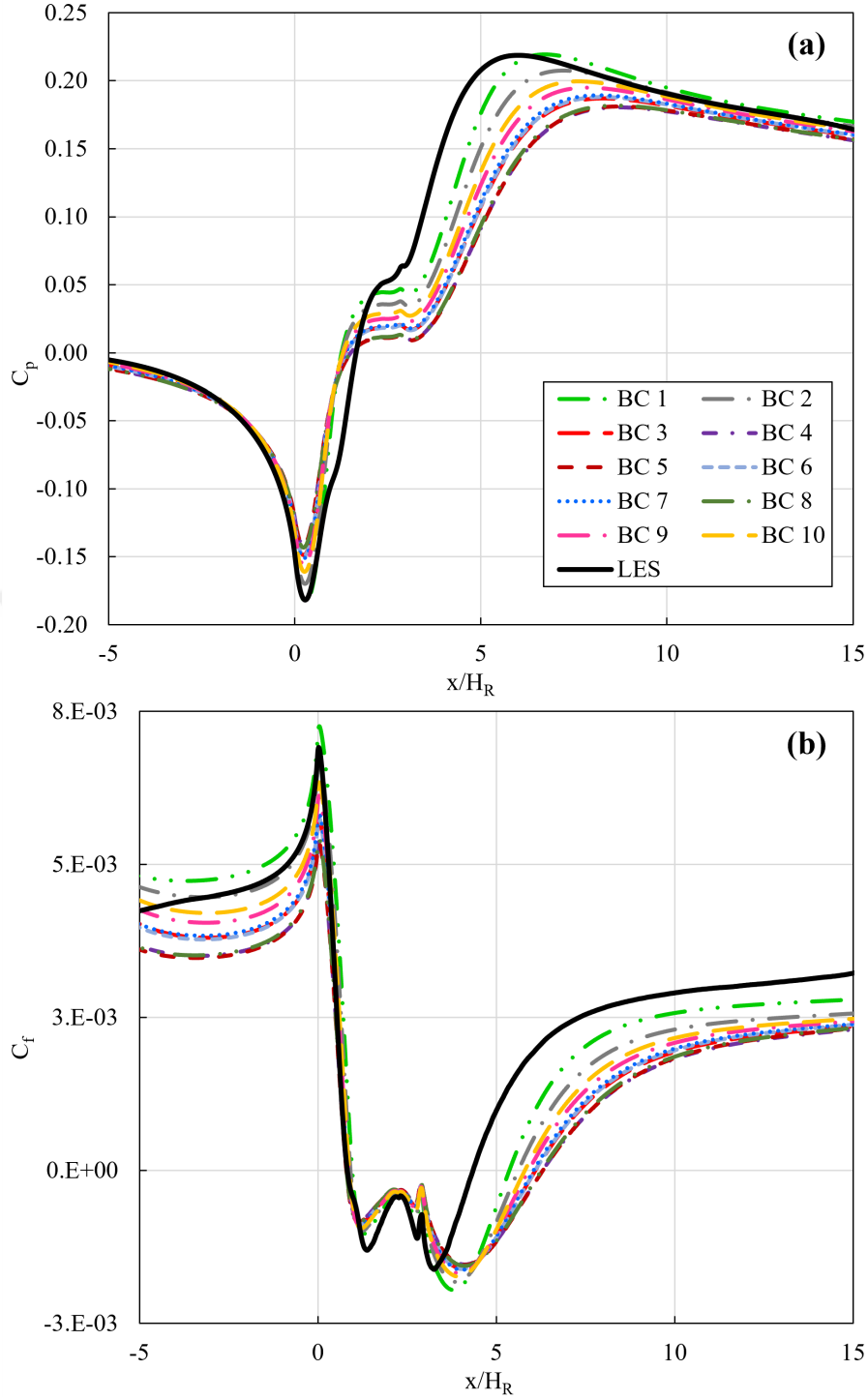


Figure 5.3 : Comparison of calculated (a) C_p and (b) C_f distributions along the wall with LES.

C_f distributions or the most accurate prediction of separation-reattachment locations. Both criteria are subject to interpretation.

Spalart and Rumsey [179] noted that a very low value of ω specification (such as $0.1U_{Ref.}/H_R$) is inadvisable due to specific vulnerability of the SST model, as it becomes susceptible to numerical instability and the limiter of the model given in

Table 5.3 : Comparison of estimated separation and reattachment points for different inlet boundary conditions in the CBFS case and relative errors with benchmark LES.

Case	$\frac{x_{Sep.}}{H_R}$	$\frac{x_{Reatt.}}{H_R}$	$\frac{L_{Sep.}}{H_R}$	$RMSE_{C_p}$	$Err_{Sep.}[\%]$	$Err_{Reatt.}[\%]$	$Err_{L_{Sep.}}[\%]$
BC 1	0.963	5.364	4.401	2.832	-16.0	-23.0	-24.7
BC 2	0.923	5.634	4.711	3.729	-11.2	-29.2	-33.5
BC 3	0.843	6.099	5.257	5.577	-1.5	-39.9	-48.9
BC 4	0.802	6.285	5.483	6.323	3.3	-44.9	-55.3
BC 5	0.802	6.317	5.514	6.345	3.3	-44.9	-56.2
BC 6	0.843	6.099	5.257	5.555	-1.5	-39.9	-48.9
BC 7	0.843	6.039	5.196	5.403	-1.5	-38.5	-47.2
BC 8	0.802	6.254	5.451	6.227	3.3	-43.4	-54.4
BC 9	0.869	5.920	5.050	4.863	-4.8	-35.8	-43.1
BC 10	0.883	5.803	4.920	4.437	-6.4	-33.1	-39.4

equation 5.9 tends to calculate the turbulent viscosity from $a_1 \omega$ which might result in an irrelevant turbulent viscosity value. Similarly, Menter [178] also recommends to define the lowest ω value within the computational domain as a lower limit and provides a calculation method. Based on his formulation, the minimum ω value for the CBFS problem is approximately $44s^{-1}$, whereas for BC 1 the inflow value is computed as $\omega_{inflow} \approx 20s^{-1}$. This discrepancy indicates that BC 1 does not represent a suitable boundary condition for this problem, possessing a "manipulated" result.

To provide a more reliable assessment, the turbulence kinetic energy budget at the inflow boundary and at the ramp onset ($x/H_R = 0$) is evaluated and compared with the benchmark study for a deeper physical validation and assessment of the boundary condition.

It is observed that BC 3, BC 6, and BC 7 cases exhibit similar results among themselves, as BC 4, BC 5, and BC 8 do. Thus, in the TKE budget comparisons, the results of BC 5 and BC 6 (where the inlet length scale value is defined using through δ_0) are not included.

To investigate the effect of inlet boundary conditions, the turbulence kinetic energy budget terms at the inlet boundary are presented in Figure 5.4, and compared with the LES results. The closest agreement with the LES convection term in the boundary layer is obtained with BC 3 and BC 7. Since the inlet boundary is the surface where the boundary conditions are directly applied, small changes to the closure equation boundary condition may lead to noticeable differences in the convection calculation

even though mean flow quantities are identical across all cases. It is also observed that the profiles do not align with the LES results in the vicinity of the wall due to the deficiency of RANS methodology.

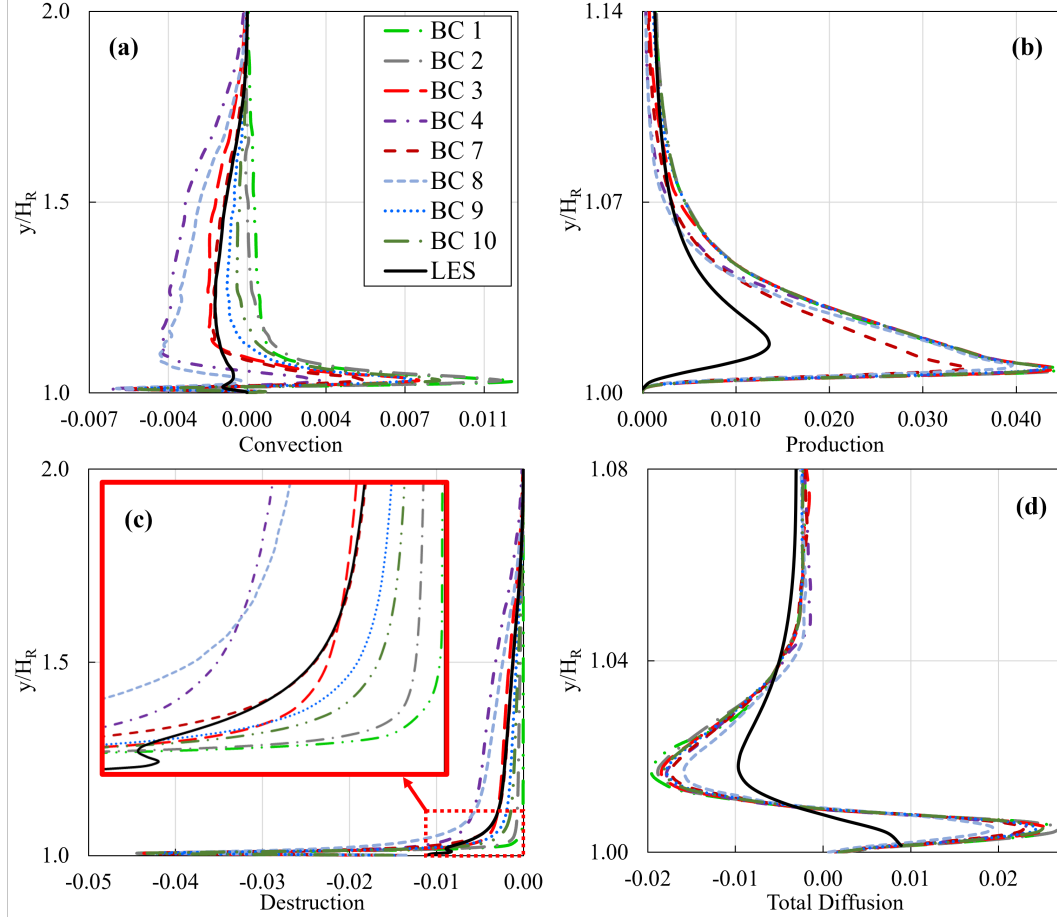


Figure 5.4 : TKE budget terms at the inflow boundary for boundary condition assessment cases (a) convection term (b) production term (c) destruction term (d) total diffusion term results comparison.

For the production term, all cases yield similar profiles, although BC 7 exhibits the lowest production near the wall. In the destruction term comparison, BC 7 again provides the closest alignment with LES results, as expected since this term is directly derived from the dissipation term of the LES based TKE budget in the case of BC 7. Interestingly, BC 3 produces nearly identical destruction term values along the inlet boundary.

The total diffusion term in the TKE budget is calculated by the sum of turbulent, viscous, and pressure diffusion terms of the LES results. In contrast, total diffusion is modeled as the sum of turbulent diffusion and viscous diffusion arising from molecular viscosity and the pressure diffusion is not explicitly resolved within the SST model's

TKE equation. The EVM's compute turbulent diffusion using the μ_t and the turbulent Prandtl number (i.e., σ_{k3} for the SST model given in equation 5.12).

A direct evaluation at the inlet boundary is insufficient due to the boundary's involvement in the governing equations. Thus, the evaluation is extended to include comparisons of TKE budget terms at the onset of the ramp, as illustrated in Figure 5.5. The TKE budget comparison along the wall-normal line at $x/H_R = 0$ shows that the BC 2 case provides the best agreement with the LES results for the convection term. However, regarding the magnitude of the production term BC 3, BC 4, BC 7, and BC 8 demonstrates superior agreement compared to the others, indicating a more accurate representation of turbulence generation mechanisms. Similarly, for the destruction term, results of the same cases are more consistent with the LES data, suggesting that their dissipation modeling aligns better with the physical characteristics of the flow. In contrast, the total diffusion term exhibits similar behavior across all boundary conditions and generally shows lower magnitudes than those predicted by the LES results. This underprediction implies that the modeled diffusion in the RANS formulation does not fully capture the combined effects of turbulent, viscous, and pressure diffusion resolved in the LES calculations.

Considering all, the most suitable inflow condition to predict the separation position is marked as the one imposed in BC 7, while BC 3 and BC 6 have nearly identical separation and reattachment predictions shown in Table 5.3. Consequently, BC 3, BC 6, and BC 7 are determined as the most suitable type of boundary condition cases. Recalling that;

- BC 3 uses the formulation $5U_{Ref.}/L$ to compute ω at the inlet boundary, where L is the characteristic length taken as the ramp height H_R for the CBFS problem, proposed by Spalart and Rumsey [179].
- BC 6 follows the approach suggested by McConnell et al. [196], where the length scale is set to 20% of the total boundary layer thickness at the inlet, considering the boundary layer thicknesses formed on both the upper and lower surfaces, instead of the suggested 10% of δ_0 .
- In BC 7 case, a length scale profile derived directly from the dissipation term computed through LES of the TKE budget equation is applied.

These three cases exhibit results that are highly consistent with one another. Similarly,

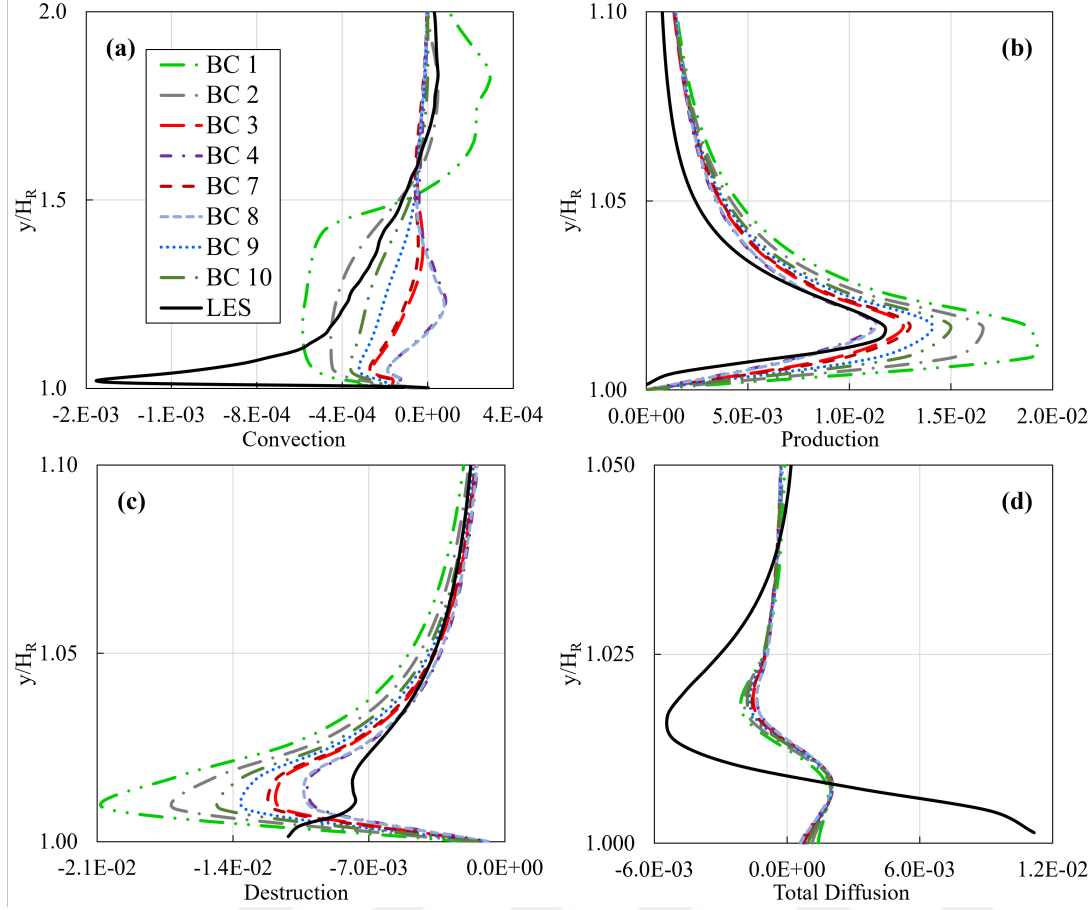


Figure 5.5 : TKE budget terms at the onset of the ramp for boundary condition assessment cases (a) convection term (b) production term (c) destruction term (d) total diffusion term results comparison.

the results from BC 4, BC 5, and BC 8 also show close agreement among themselves. Although both sets are based on solid physical reasoning, it is reasonable to expect that BC 7, which directly uses dissipation data from LES, would provide the best overall alignment.

In addition to its strong consistency with the Spalart and Rumsey formulation, the BC 7 boundary condition also satisfies the relation proposed by Davidson [180] i.e., $u_{in,RMS}/u_\tau = 1.5$. Since the ratio is calculated using the assumption $u'_{RMS} \approx \sqrt{2/3k}$, and the friction velocity (u_τ) computed from the LES data at the inflow gives ≈ 1.5 . It should be noted that $Re_\tau \approx 618$ obtained at the inlet for BC 7 show a close agreement with the LES. However, since dissipation profiles obtained directly from TKE budget equation of the LES are not always available, and given that BC 3 yields nearly identical results to BC 7, the BC 3 case is selected for subsequent analyses of the CBFS and other problems. Thus, the inflow condition for closure equation is set to $\omega_{inflow} = 5U_{Ref.}/H_R$ as suggested by Spalart and Rumsey [179]. It should also be

noted that this assessment is performed using CFX. As emphasized by Sarkar and Savory [197], the results may be different when using a different finite volume solver.

5.4.1.2 Model results

In this study, RANS simulation results of the curved backward-facing step problem with Menter's [143] SST 2003e model and the RNG-coupled SST model (hereafter referred to as SST and RNG-SST, respectively) is comparatively evaluated. Since the inlet boundary conditions were extracted from the LES of Bentaleb et al. [182], the inlet Reynolds number based on momentum thickness (Re_θ) was found to be 1192 in all simulations, consistent with the LES data. Table 5.4 presents the root mean square errors of the pressure coefficient distribution, along with the predicted separation and reattachment positions, separation length, and their relative errors with respect to LES results. For comparison, results obtained by other researchers using the standard SST model and its variants for transitional flow i.e. the $\gamma - Re_\theta$ model and intermittency SST, are also provided [158, 173]. To examine the three-dimensional effects, the computational domain is extended in the spanwise direction (z-axis) to match the spanwise extent provided in the ERCOFTAC KBwiki database. A total of 192 cells distributed along the z-direction resulted in approximately 23 million elements in the computational grid. For the considered case under the same boundary conditions, the comparison between two-dimensional and three-dimensional CFD analyses revealed no significant differences, indicating that the flow remains predominantly two-dimensional for this configuration.

All relative errors reported in this study are calculated using the relative error formulation given in equation 5.26.

$$\text{Approx. Rel. Err.} = \frac{\text{Exact Value} - \text{Computed Value}}{\text{Exact Value}} \times 100\% \quad (5.26)$$

The SST model demonstrates adequate accuracy in predicting the separation position. The discrepancies among different researchers in estimating the separation location primarily arise from variations in the boundary conditions determined for the inflow boundary. In contrast, the RNG-SST model delays the separation position, resulting in a higher relative error. It is noteworthy that the use of transition models also increases the error in predicting the separation point. For the evaluation of the reattachment position, SST model shows very high relative error. This error is reduced when $\gamma - Re_\theta$ transition correction is applied to SST. Nonetheless, applying intermittency

Table 5.4 : Predicted separation and reattachment points in the CBFS case and relative errors with benchmark LES.

Source	Model	$\frac{x_{Sep.}}{H_R}$	$\frac{x_{Reatt.}}{H_R}$	$\frac{L_{Sep.}}{H_R}$	$RMSE_{C_p}$	$Err_{Sep.}$ [%]	$Err_{Reatt.}$ [%]	$Err_{L_{Sep.}}$ [%]
Ref. [182]	LES	0.83	4.36	3.53	—	—	—	—
Ref. [191]	SST	0.87	5.62	4.75	—	-4.8	-28.9	-34.6
Ref. [196]	SST	0.80	6.33	5.53	—	3.6	-45.2	-56.7
	SST	0.84	6.10	5.26	5.577	-1.5	-39.9	-48.9
	RNG-SST	0.92	5.13	4.21	3.015	-11.2	-17.7	-19.3
	$\gamma-Re_\theta$	0.88	5.47	4.59	3.119	-6.4	-25.5	-29.9
Present	SST							
Numerical	Intermittency	0.79	6.16	5.37	6.137	4.9	-41.3	-52.2
Study	SST							
	SST (3D)	0.83	6.16	5.33	5.605	0.1	-41.3	-51
	RNG-SST (3D)	0.92	5.13	4.21	3.034	-11.2	-17.7	-19.3

correction increases it. In parallel with $\gamma - Re_\theta$, the RNG-SST shows reduced error. In fact, the proposed model achieves better results than SST and its enhanced versions for transitional flows. However, comparing models based on total separation length provides a more meaningful inference. From this perspective, the RNG-SST model yields the lowest overall error among all models evaluated, and decreases the relative error of SST by approximately 60%.

In terms of pressure coefficient distribution, the RNG-SST model again exhibits the best performance, as can be seen from the distribution of C_p shown in Figure 5.6. Similarly, the skin friction coefficient distribution indicates that the RNG-SST model provides a more accurate wall shear prediction than the SST. This improvement is particularly evident at critical regions, such as the onset and end of the ramp, where the magnitudes of both C_p and C_f predictions show better agreement with the LES results.

To evaluate the model's performance not only through wall quantities such as wall shear and pressure but also in the off-wall regions, comparisons of velocity and turbulent kinetic energy profiles are presented in Figure 5.7. Note that, in the TKE profiles, all k values are shifted by a factor of 13 to make the differences between profiles more distinguishable.

When examining the velocity profiles, slight differences can be observed. A clear deviation appears between the LES and SST results in the flow separation region

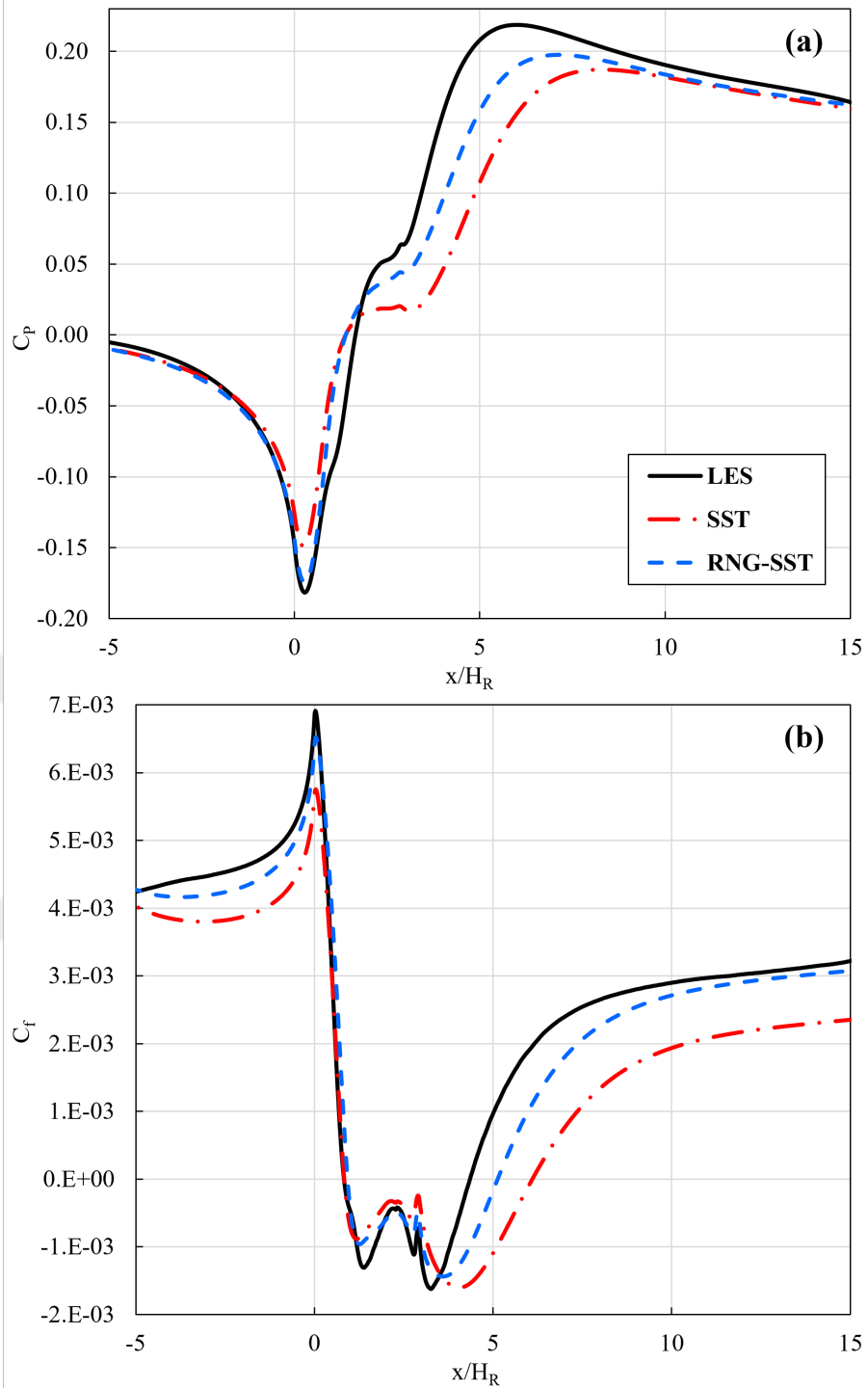


Figure 5.6 : RNG-SST model results comparison with SST and LES for (a) C_p and (b) C_f distributions along the wall.

($x/H_R = 0.83$ and $x/H_R = 4.36$) and during the post-reattachment recovery ($x/H_R = 5, 6, 7$, and 8). The RNG-SST model exhibits better consistency with the LES data in the pre-separation region ($x/H_R < 2$), though this agreement deteriorates downstream. This trend aligns with the RNG-SST model's prediction of a delayed reattachment compared to LES. Nevertheless, across all positions, RNG-SST provides a more

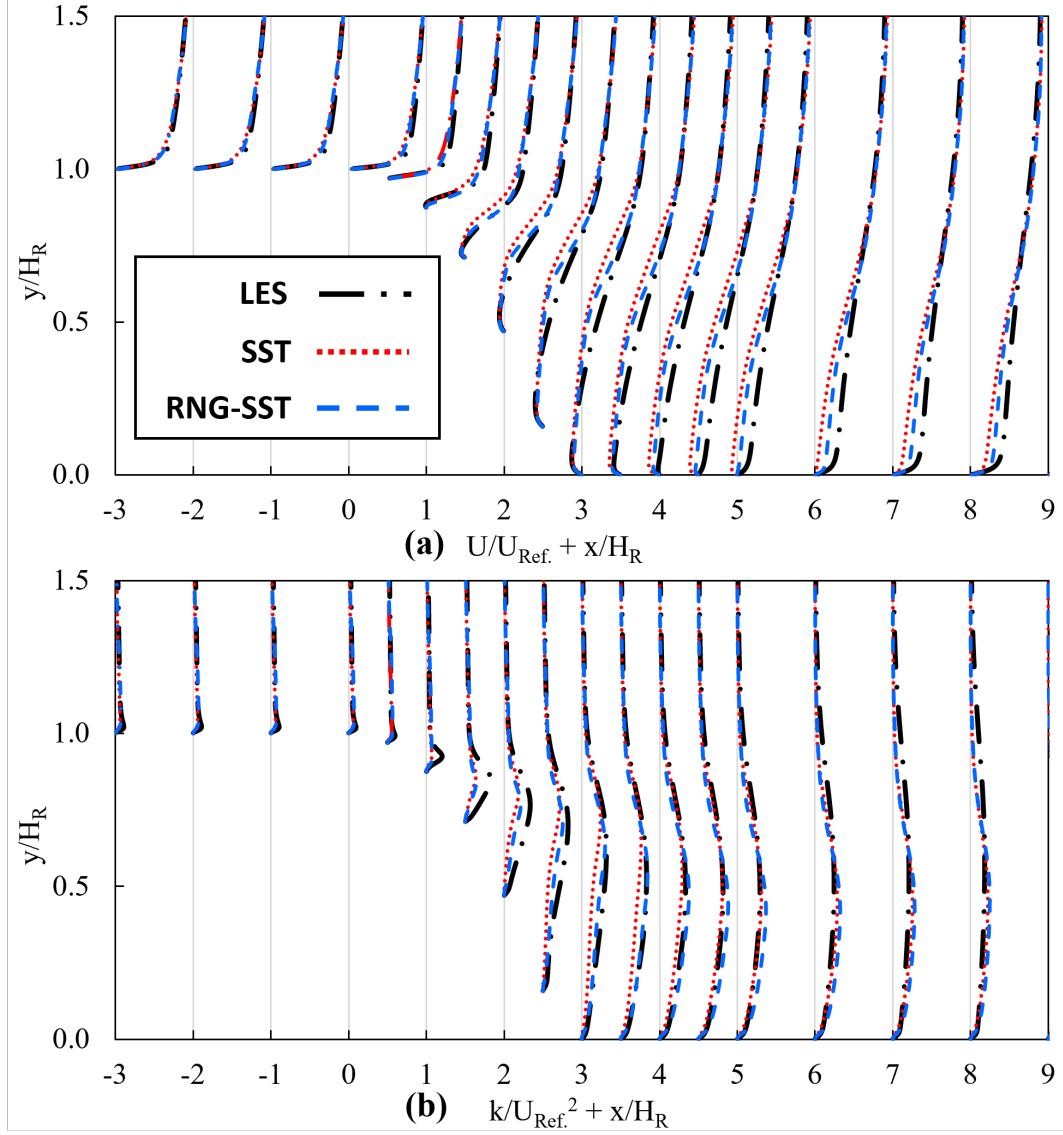


Figure 5.7 : RNG-SST model results comparison with SST and LES for (a) dimensionless mean velocity profiles and (b) dimensionless k profiles calculated in the flow field in CBFS case.

aligned velocity distribution than SST. A similar improvement is evident in the TKE profiles, where the RNG-SST shows higher accuracy compared to SST. Due to the inherent nature of RANS modeling, both models yield comparable results outside of the separated region, as they are strongly influenced by the imposed boundary conditions. Still, RNG-SST enhances the TKE prediction particularly within the separated region, which in turn leads to a more accurate estimation of the reattachment position. In order to discover the underlying physics of RNG enhancement, TKE budget terms were computed at multiple streamwise locations and compared with benchmark LES study as shown in Figure 5.8.

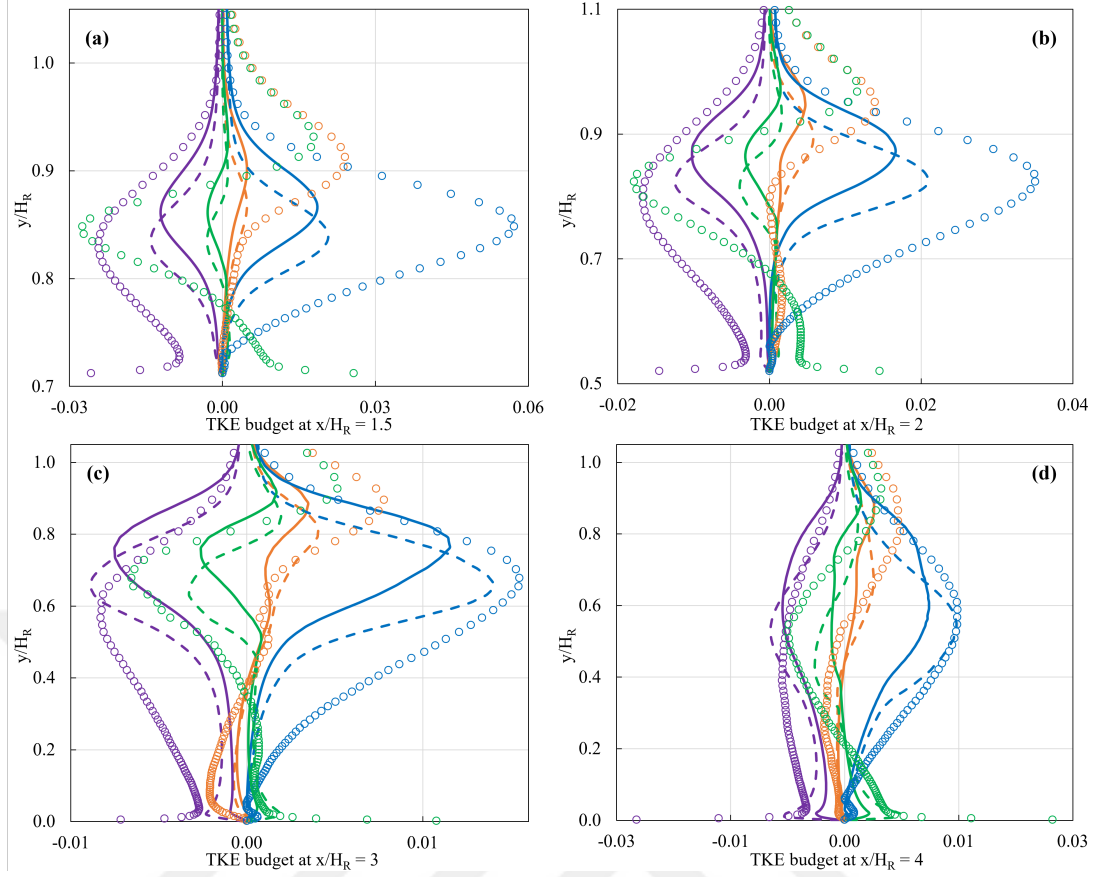


Figure 5.8 : TKE budget comparisons at (a) $x/H_R = 1.5$, (b) $x/H_R = 2$, (c) $x/H_R = 3$, (d) $x/H_R = 4$. Symbols: $\circ$, LES results; — (solid lines), present numerical study with SST model; - - - (dashed lines), present numerical study with RNG-SST model. Colors: orange, convection; blue, production; green, total diffusion; purple, dissipation.

As noted by Bentaleb et al. [182], the first two streamwise stations are located near the region where turbulent kinetic energy production reaches its maximum within the separated shear layer, which is also observed in the present RANS simulations. Before analyzing the cross-flow variations of the budget components at streamwise stations, it is essential to consider the thickness of the reverse flow layer as advised by Bentaleb et al. [182].

In the present study, the reverse flow layer thicknesses computed from the RANS simulations using SST are 0.074, 0.208, 0.426, and 0.329 y/H_R . With the RNG enhancement, these values were reduced to 0.056, 0.171, 0.373, and 0.236 y/H_R , respectively. Consequently, the absolute relative errors with respect to the LES results decreased from approximately 48% to 13%, from 60% to 31%, from 42% to 24%, and from 55% to 37% at each corresponding position. However, since the reattachment point predicted by RNG-SST does not precisely coincide with the LES reference value

at $x/H_R = 4.36$, the model still exhibits notably high relative error close to this point.

When the RNG-SST model is evaluated based on the trends of the budget term profiles, it generally predicts their positions relative to the wall more accurately than the SST. The locations of both the flat and peak regions in the LES data align more closely with those obtained from the RNG-SST model over and beyond the ramp. Similar to the comparisons of velocity and turbulent kinetic energy profiles, even though the magnitudes are not identical, the improvement in the profile trends indicates that the RNG-SST provides a better prediction of the flow field compared to the SST. As the adverse pressure gradient increases along the ramp, the production term peaks predicted by SST move further away from the wall, while RNG-SST captures this behavior more accurately. After the end of the ramp, the effect of the RNG modification becomes even more evident. At positions $x/H_R = 3$, and 4, the destruction term estimated by RNG-SST closely matches, and even exceeds, the LES results. The increasing adverse pressure gradient along the ramp alters the turbulence production mechanisms, shifting energy generation from the wall toward the shear layers in the separated region. With the RNG enhancement, the SST turbulence model gains the ability to capture these transitions more effectively. As observed in the $F_{Reattach}$ contour plot presented in Figure 5.9, the confinement of the RNG enhancement through $F_{Reattach}$ between the viscous sublayer and the outer region positively influences the flow field predictions within the shear layer.

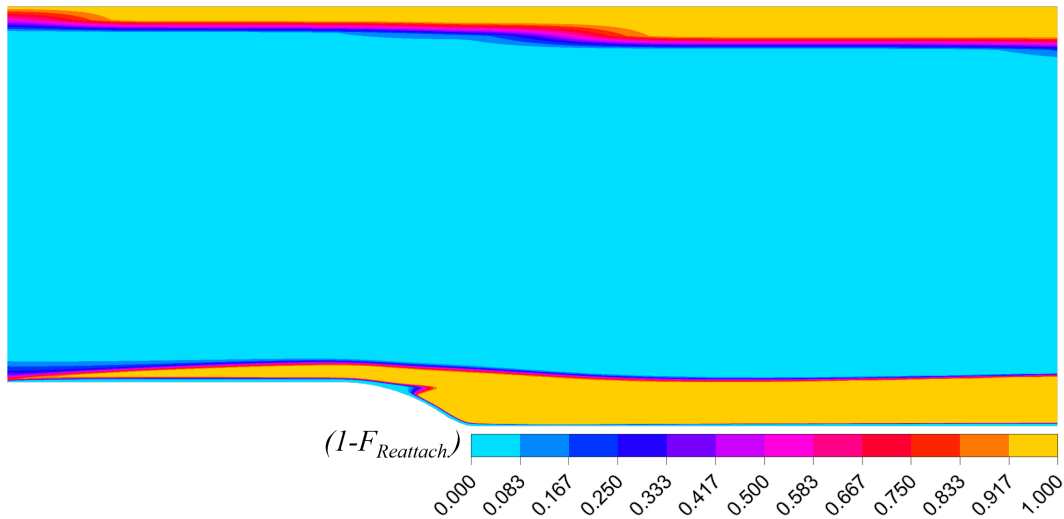


Figure 5.9 : $F_{Reattach}$ distribution computed in CBFS case.

Since the RNG-SST coupling is achieved by adding the RNG model's f_η term as a source term in the turbulence eddy frequency equation, its influence becomes

particularly apparent in the strongly separated flow region, where it enhances the destruction term, as illustrated in Figure 5.10. As can be seen in the figure, the f_η causes reduced the production of destruction, leading to a decrease in eddy frequency and consequently an increase in k . The rise in kinetic energy, combined with the decrease in eddy frequency, increases eddy viscosity, yielding to enhanced turbulent diffusion as can be understood from the total diffusion term. Hence, the diffusion process extracts energy from this region and transports it both toward the outer shear layer and back to the wall, aligning with the physical mechanisms governing turbulent mixing.

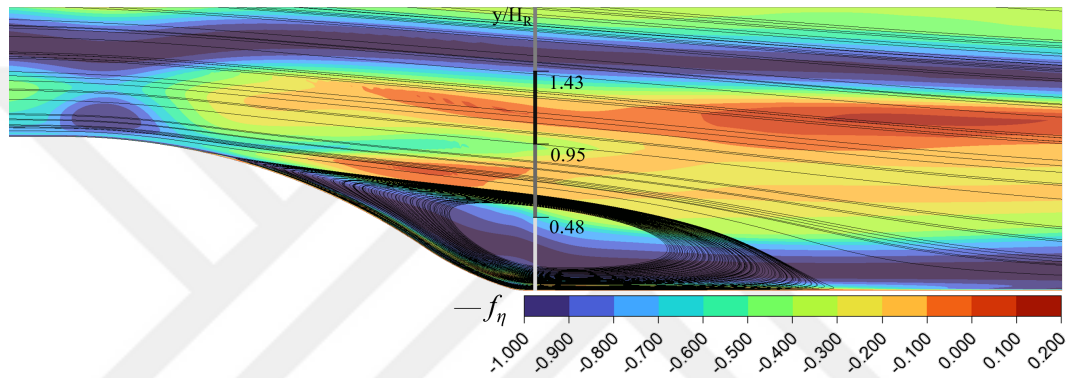


Figure 5.10 : $-f_\eta$ distribution computed in CBFS case.

5.4.2 Other test cases

To further examine the predictive capability of the proposed model, its performance is assessed using two benchmark configurations characterized by separated boundary layer flows subjected to adverse pressure gradients at various inlet Reynolds numbers. The selected test cases, namely the flow over backward-facing step and the flow over periodic hills, represent canonical flows that effectively capture distinct separation mechanisms, thereby providing a rigorous evaluation of the model's applicability to turbulent flows involving separation and reattachment.

5.4.2.1 Flow over backward-facing step

In contrast to separation from continuous surfaces (e.g., rounded steps), the recirculation zone in BFS flows is initiated exclusively by the geometric expansion at the step. Consequently, the separation point is fixed at the step corner, and the shear layer bounding the recirculation zone begins from this location. The prevalence of analogous flow structures in numerous engineering applications has

established the BFS as a fundamental benchmark case, extensively investigated through both experimental measurements and numerical simulations to reveal the underlying physics of turbulent separated flows [198, 199].

To systematically assess the influence of Reynolds number and step angle on separated flow characteristics, four BFS configurations were selected for comparative analysis within the experimental study conducted by Ruck and Makiola [187] and documented in the ERCOFTAC Classical Collection database. The corresponding CFD simulations have been performed under the same inlet and boundary conditions as given in the experiments, allowing a direct comparison between numerical and experimental results. The dynamics of the recirculation region, as well as the reattachment length and turbulent mixing behavior downstream of the step, are predominantly controlled by these parameters. In other respects, the inlet velocity profile imposed in this internal flow problem differs from that of the ramp cases and represents a fully developed channel flow with the step height $H_R = 0.012$ m, while the channel height corresponds to approximately $2.1 H_R$. The reference velocity for each BFS case is the maximum velocity at the inlet boundary, while the characteristic length is the step height. The ω boundary condition is applied as a constant value, calculated based on these reference parameters.

Figure 5.11 and 5.12 present the comparisons of the mean velocity and its fluctuating component obtained at various streamwise stations along the channel for the 90 step angle cases at 47000 and 64000 Reynolds numbers, respectively.

Since the fluctuating velocity is not directly computed in RANS simulations, it was derived using the approach given in equation 5.27 and then non-dimensionalized with the reference velocity. To make the differences more observable in the figures, all mean velocity profiles were multiplied by a factor of three, whereas all fluctuating velocity profiles were multiplied by nine.

$$u'_{RMS} = \sqrt{\frac{2}{3}k} \quad (5.27)$$

It is observed that within the recirculation region, the velocity profiles obtained using the SST align more closely with the experimental measurements compared to those predicted by the RNG-SST. The reattachment positions (x_R/H_R) calculated for the BFS cases and their relative errors with respect to the experimental data, as presented in Table 5.5, further support this observation. The flow reattachment occurs where the

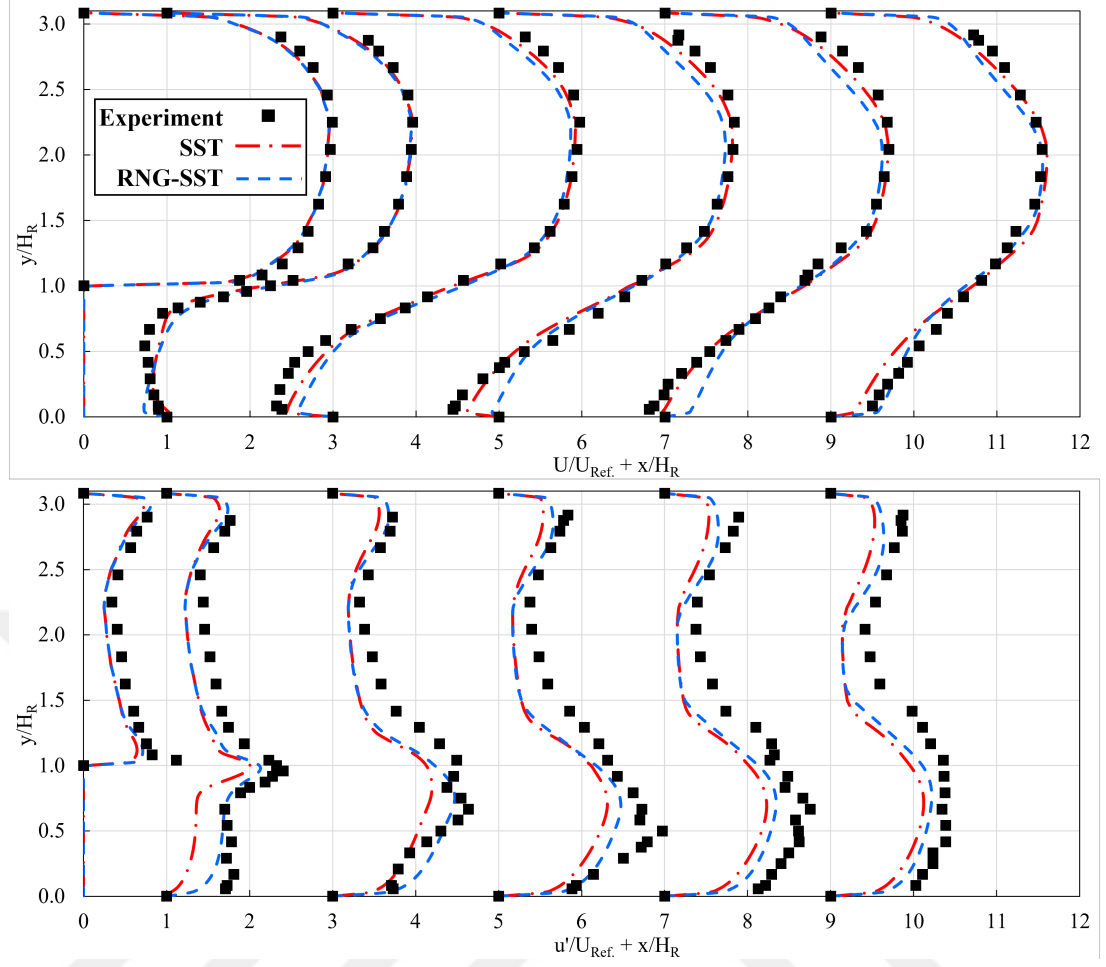


Figure 5.11 : Mean (top) and fluctuating (bottom) velocity profile comparisons of BFS-a case.

velocity gradient near the wall changes sign from negative to positive, resulting in a positive wall shear stress [187].

Table 5.5 : Reattachment positions predicted in the present study and their relative errors with the experimental measurements for the BFS cases.

Source	Re	Step Angle [$^{\circ}$]	$\frac{x_R}{H_R}$	$Err.R[\%]$
Ref. [187]	47000	90	7.60	—
Ref. [187]	64000	90	7.65	—
Ref. [187]	47000	20	5.30	—
Ref. [187]	64000	20	5.40	—
SST	47000	90	7.75	-2
SST	64000	90	7.85	-3
SST	47000	20	7.41	-40
SST	64000	20	7.54	-40
RNG-SST	47000	90	5.90	22
RNG-SST	64000	90	5.90	23
RNG-SST	47000	20	5.49	-4
RNG-SST	64000	20	5.49	-2

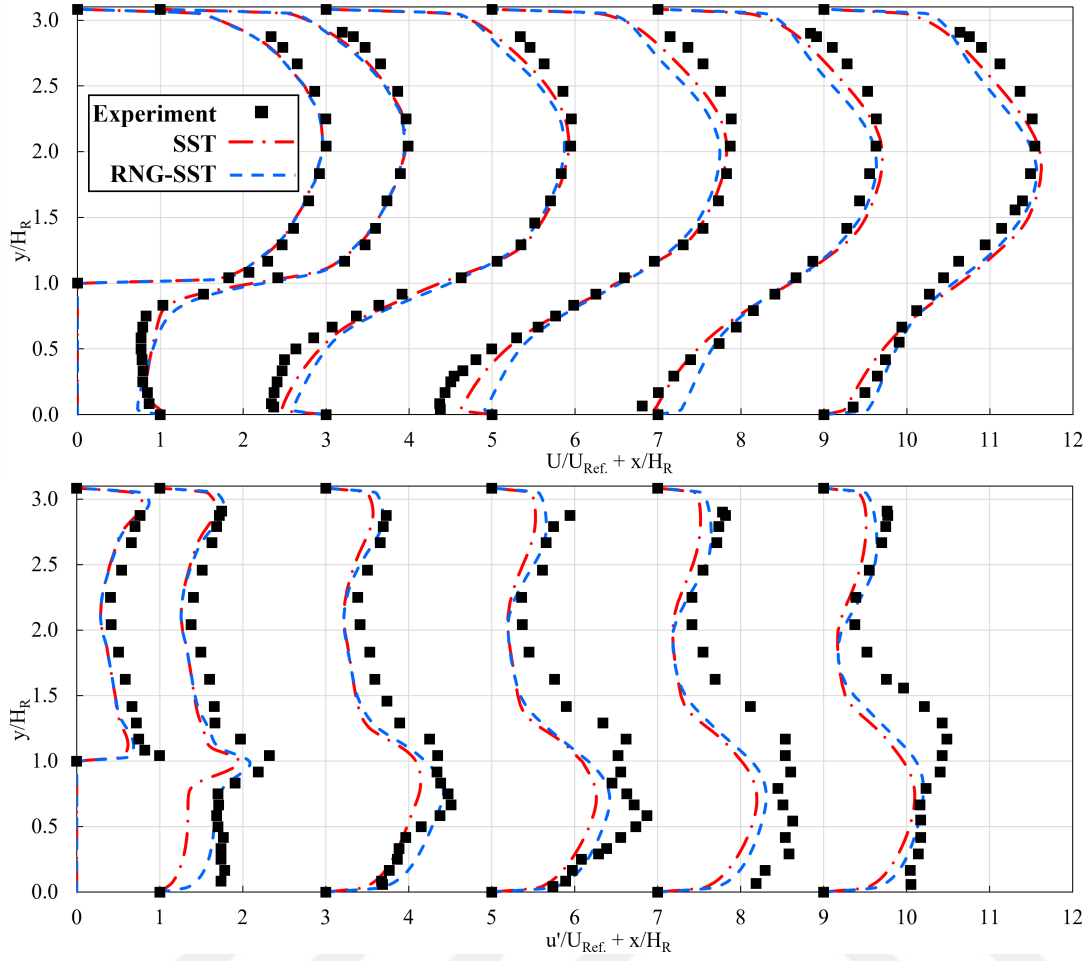


Figure 5.12 : Mean (top) and fluctuating (bottom) velocity profile comparisons of BFS-b case.

In contrary to mean velocity distribution, RNG-SST provides a more aligned fluctuating velocity profiles with the experimental measurements. This raises the question of why the RNG-SST model, despite its improved prediction of fluctuating velocity and consequently turbulent kinetic energy, tends to under-predict the reattachment position when compared to SST. The explanation lies in the change in flow physics and turbulence mechanics associated with the variation in the step angle. At the 90° step angle, the separation point is fixed, resulting in the mean flow becoming the dominant mechanism in the transport of momentum and turbulent kinetic energy. This fixed separation due to the sharp geometric discontinuity indicates the absence of a wandering adverse pressure gradient over the step. Consequently, the determination of the reattachment position relies primarily on the mean flow quantities, emphasizing that the convection term plays a leading role in predicting the reattachment position. According to the experimental results of Ruck and Makiola [187], for the same Reynolds number and expansion ratio (1.48), no flow separation

occurs when the step angle is 10° . However, when the step angle is increased to 20° , flow separation is observed. Figure 5.13 and 5.14 present the mean velocity and fluctuating velocity profiles for the 20° step angle cases at 47000 and 64000 Reynolds numbers, respectively.

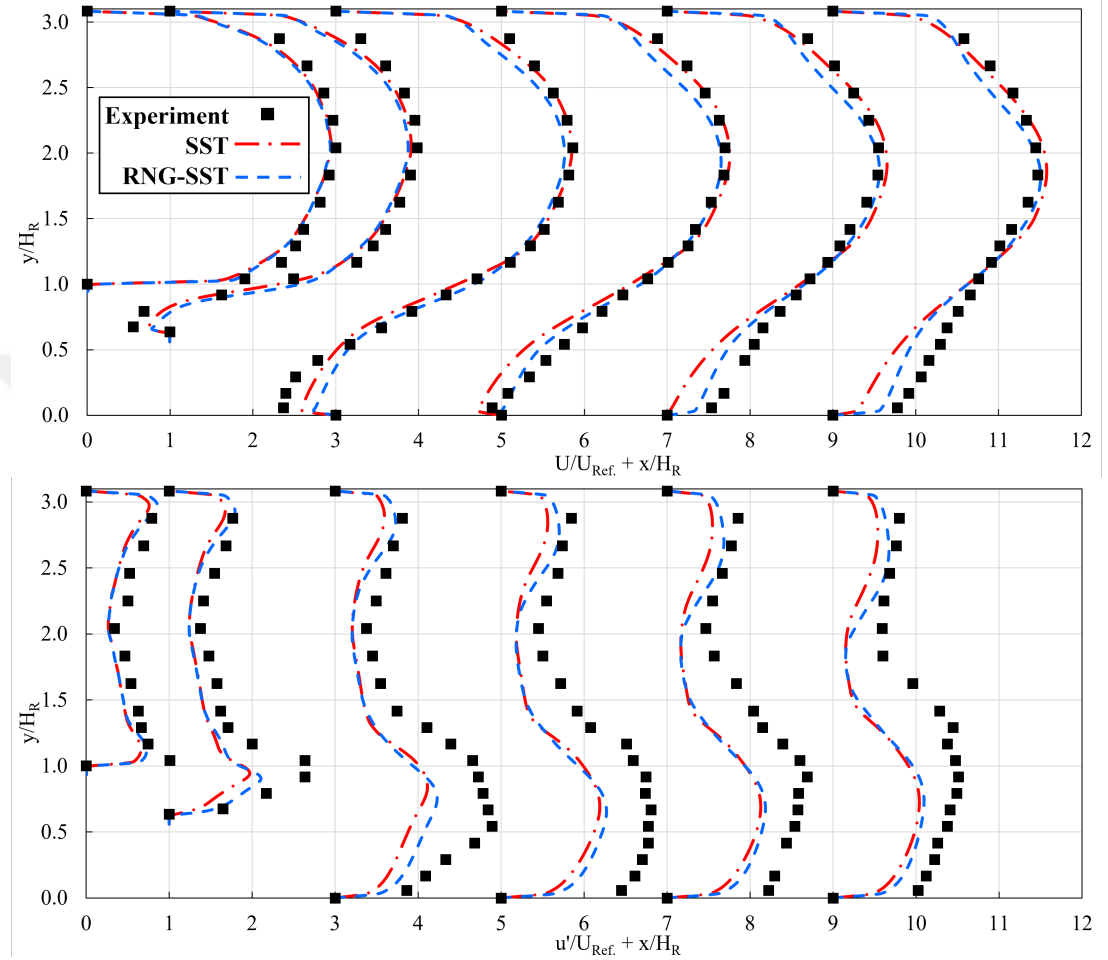


Figure 5.13 : Mean (top) and fluctuating (bottom) velocity profile comparisons of BFS-c case.

When the step angle is decreased to 20° , the flow separation no longer begins from a fixed location. Instead, it is triggered by the local flow properties, particularly by the pressure gradient evolving over the step surface. This transition in the separation mechanism changes the flow physics and increases the relative importance of the pressure gradient and the turbulence production terms over the convective transport. Similar conclusions were also reported by Lee and Nguyen [198], who conducted detailed investigations on the same BFS problems. They described the so-called “splat effect”, i.e., the mechanism in which large-scale coherent vortices generated by flow separation impinge on the wall, thereby increasing pressure and turbulent

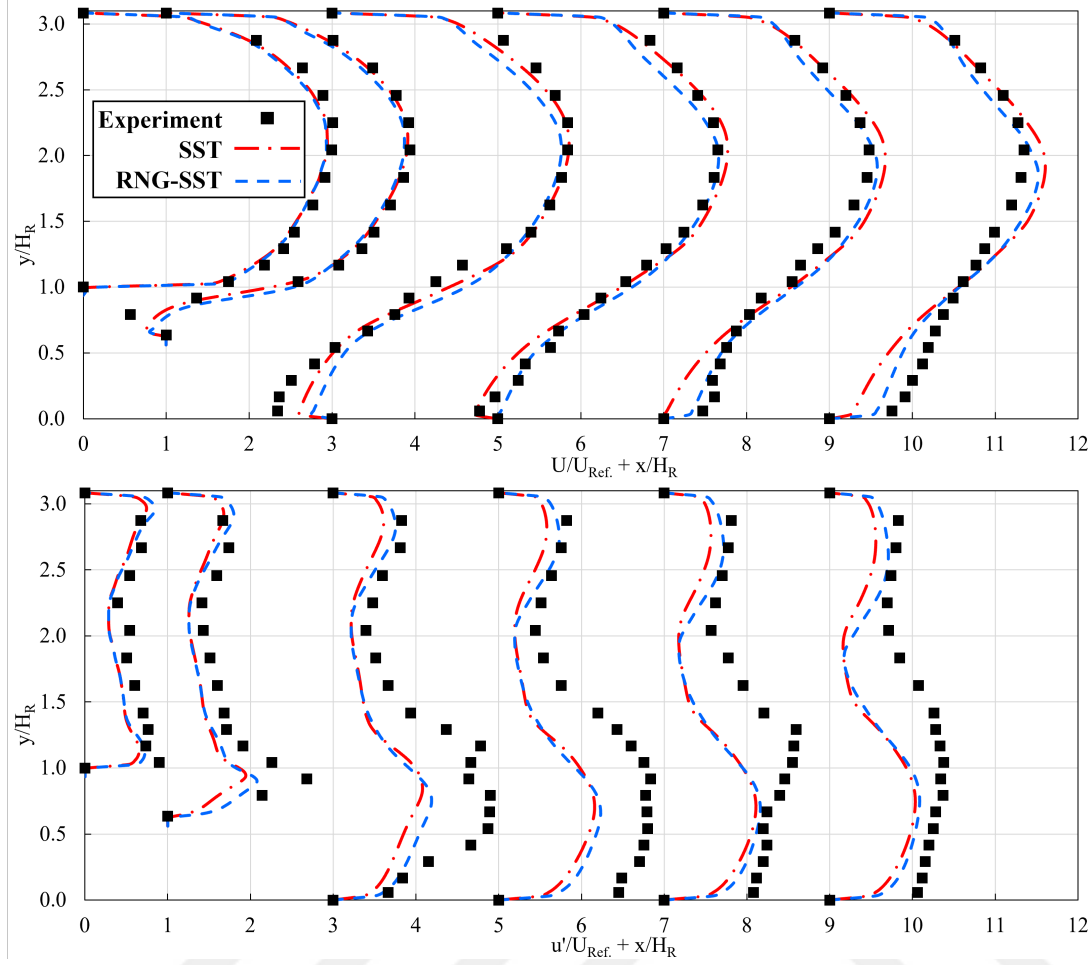


Figure 5.14 : Mean (top) and fluctuating (bottom) velocity profile comparisons of BFS-d case.

diffusion along the recirculation region and serving as the primary contributor to the production and turbulent diffusion terms in the TKE equation. It has been emphasized that such structures become dominant at low step angles, significantly influencing the turbulence mechanics. This observation aligns well with the argument presented in this study, i.e., when separation occurs over a continuous surface, the factors determining the reattachment position are primarily governed by diffusion and dissipation, rather than by convection. For the CBFS case, the agreement of the production, diffusion, and destruction peak locations in the k equation with the LES data was observed, achieved through the RNG modification of SST model (see Figure 5.8). The same improvement is not observed for the convection term, supporting this interpretation. Since the separation occurs on a continuous surface, the better alignment of other terms governing the reattachment mechanism with the LES results contributes to a reduction in the relative error in the prediction of the reattachment position. Considering all these findings, in the 90° BFS configuration, the flow separates sharply and forms a

distinct separation bubble adjacent to the step. The reattachment region immediately downstream of this bubble is characterized by a strong mean velocity gradient and high shear stress, indicating that mean flow properties play a primary role in determining the reattachment location. When the step angle is reduced to 20° , however, the separation point is no longer fixed, and the flow undergoes a more gradual separation along the step surface. Similar to the CBFS problem, this behavior alters the dynamics of the separation bubble. In such configurations, the unsteady nature of the flow enhances the formation of coherent vortical structures, making turbulent quantities such as k and Reynolds stresses more dominant. Consequently, the prediction of the reattachment position becomes more strongly influenced by these unsteady turbulent structures, rather than by the mean flow behavior.

5.4.2.2 Flow over periodic hills

Similar to the CBFS problem, the flow over periodic hills involves separation from a continuous curved surface induced by adverse pressure gradients. In addition to this similarity, it also includes other important flow characteristics such as streamline curvature, favorable pressure gradients, and streamwise periodicity, making it a valuable benchmark for a detailed evaluation of RANS model performance. By applying streamwise periodic boundary conditions at both the inflow and outflow boundaries, the influence of the ω boundary condition is effectively eliminated. Therefore, periodic hills case was investigated to assess the RNG improvement for the SST model's predictive capability at two different Reynolds numbers. The characteristic length is defined as the hill crest height ($H_R = 0.028$ m), corresponding to two Reynolds numbers of 10595 and 37000, respectively. The reference velocity (U_b) calculated at the crest of the first hill, and the mass-flow rate set at the inflow and outflow periodic surfaces ensure that the specified Reynolds number based on U_b is achieved.

Figure 5.15 presents the velocity profiles obtained within the flow field at $Re = 10595$ and $Re = 37000$, compared with the large-eddy simulation data provided by Temmerman et al. [193, 200]. Although no significant discrepancies are observed between the two Reynolds-Averaged Navier-Stokes models in terms of mean velocity distributions, Table 5.6, showing the reattachment positions (x_R/H_R) and their relative errors with respect to the large-eddy simulation and experimental measurements,

indicates that the reattachment mechanism in this case is primarily governed by turbulent quantities, as expected for separation from a continuous surface.

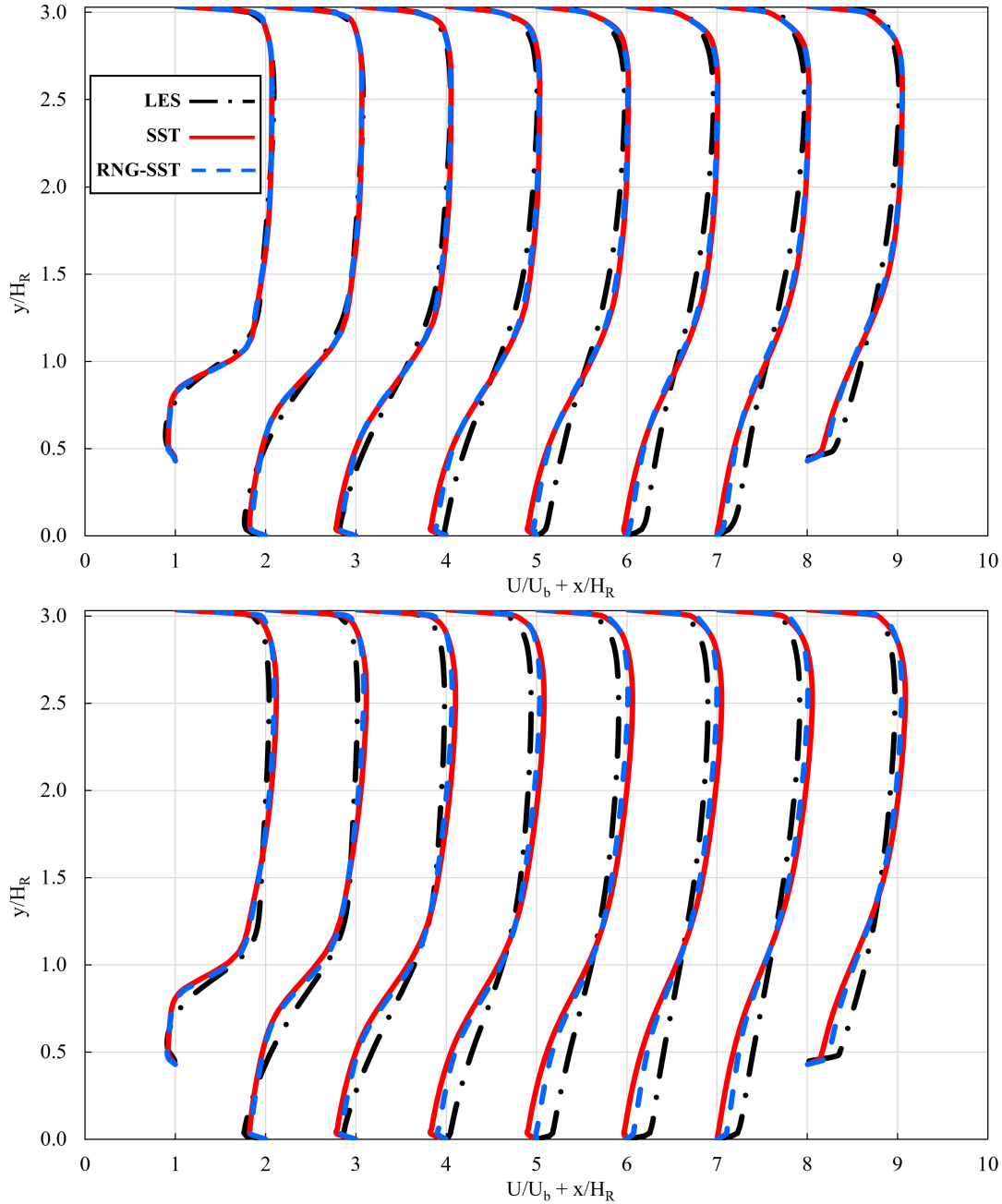


Figure 5.15 : Streamwise velocity profile comparison of Periodic Hills-a case (top) and Periodic Hills-b case (bottom).

As can be seen from the comparison of the skin friction coefficient distribution along the bottom surface for $Re=10595$ in Figure 5.16, the implementation of the RNG formulation enhances the SST model's capability to predict reattachment in separated flows over continuous surfaces. However, over the bottom wall downstream of the first hill, both models tend to under-predict the C_f . Near the end of the first hill, the RNG

Table 5.6 : Comparison of the estimated reattachment points in the present study with the benchmark studies of periodic hills case.

Source	Re	$\frac{x_R}{H_R}$	$Err.R[\%]$
LES [193]	10595	4.72	—
Exp. [189]	37000	3.76	—
SST	10595	6.99	-48.1
SST	37000	6.99	-85.9
RNG-SST	10595	6.13	-29.8
RNG-SST	37000	5.43	-44.5

enhancement reduces the relative error, but in the beginning of the bottom wall, the increased influence of the turbulent terms, i.e., production, destruction, and diffusion, intensifies the C_f in the positive direction due to the splat effect. Nevertheless, this mechanism contributes to a more accurate reattachment prediction, improving the overall model performance in comparison with the experiment and LES results. According to C_f distribution of LES, a secondary separation bubble forms at the beginning of the second hill. Although the RNG-SST model improves the reattachment prediction compared to SST, it fails to capture this secondary bubble. C_f values obtained with RNG-SST show a local decrease upstream of the second hill, yet the reduction is not strong enough to reach negative C_f levels, indicating reverse flow. This inability of the RNG-SST to predict this secondary separation bubble may be attributed to its enhancement of splat effects.

With increasing Reynolds number, the relative error in estimating the reattachment location increases for both models. However, coupling the SST with RNG reduces it by nearly half compared to the LES. At $Re=37000$, the streamwise velocity at the outer part of the shear layer tends to slow, which results in better alignment with the LES results, particularly close to the second hill. It can be attributed to the enhanced diffusion in the shear layer caused by the RNG modification, resulting in an energy transfer from the shear layer to its inner and outer regions.

The separation point, however, remains unchanged. Both models predict the onset of separation at $x/H_R = 0.315$, compared to $x/H_R = 0.22$ reported in the LES study conducted with the Reynolds number of 10595 [193]. This indicates that while the RNG-SST model effectively improves the reattachment prediction, it does not disrupt the separation mechanism, preserving the underlying physics of the flow.

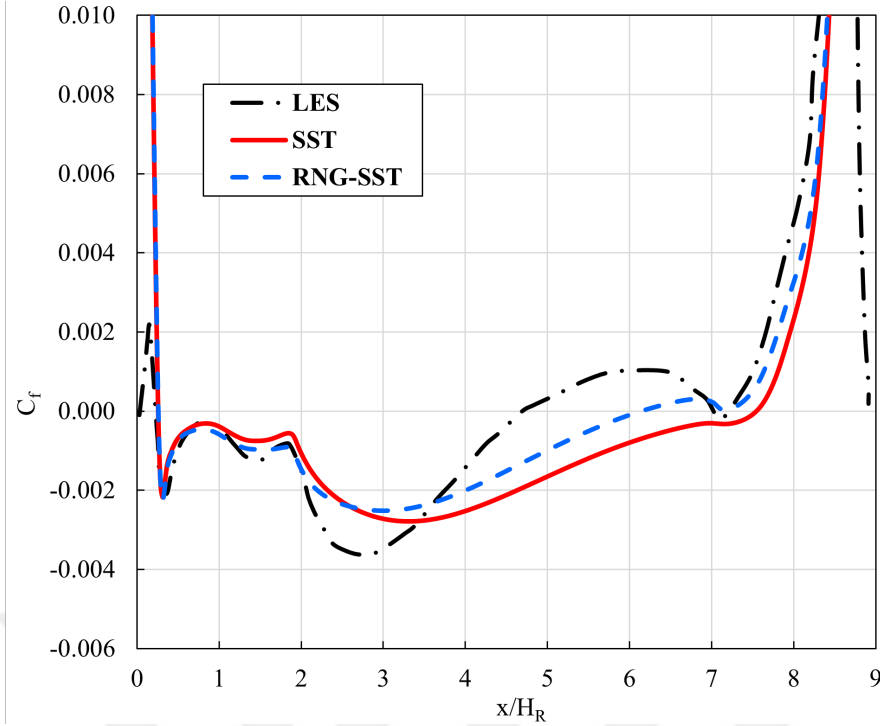


Figure 5.16 : C_f distribution on the hills' bottom wall at $Re=10595$.

5.4.3 Experimental investigation on backward-facing ramp

In the experimental campaign, the flow over the ramp placed within an open-channel wind tunnel setup is investigated in terms of velocity and pressure measurements. The ramp profile differs from that examined by Zhang and Zhong [195]. The Reynolds number, based on the reference velocity and the ramp height, is 48570. The momentum thickness Reynolds number (Re_θ) at the onset of the ramp is calculated as 1430 in the current study while it was reported as 490 in the study of Bentaleb et al. [182]. The freestream turbulence intensity of the wind tunnel at the inlet plane is measured to be approximately 9.6%. Velocity measurements are performed along 15 stations in the wall-normal direction. Boundary layer thickness (δ), momentum thickness (θ), and displacement thickness (δ^*) are calculated from measured velocity. The measurement locations and the corresponding boundary layer parameters, non-dimensionalized with the ramp height, are presented in Table 5.7.

To minimize the influence of computational grid on the numerical results of RANS simulations, eight computational grids with different cell numbers are generated by systematically increasing the number of nodes in the streamwise and wall-normal directions. Based on previous findings from the investigated CBFS of Zhang and Zhong's [195] ramp profile, CFD analyses were performed using a two-dimensional

Table 5.7 : Boundary layer parameters calculated with measured velocity along the wall-normal direction at each station.

$\frac{x}{H_R}$	$\frac{\delta}{H_R}$	$\frac{\delta^*}{H_R}$	$\frac{\theta}{H_R}$
-1.86	0.369	0.045	0.034
0	0.259	0.031	0.025
0.64	0.206	0.024	0.018
1.27	0.315	0.04	0.03
1.88	0.361	0.058	0.041
2.03	0.407	0.076	0.052
2.18	0.438	0.081	0.054
2.33	0.481	0.116	0.059
2.48	0.499	0.124	0.064
2.62	0.547	0.153	0.069
3.05	0.809	0.284	0.097
3.6	0.922	0.428	0.12
4.64	0.904	0.308	0.135
5.42	0.883	0.245	0.137
5.69	0.869	0.222	0.134
6.21	0.862	0.213	0.135

grid structure, since three-dimensional effects are found to be negligible in predicting separation and reattachment positions (see Table 5.4). CFD analyses were conducted under identical boundary conditions using the SST model for the mesh independence study. Figure 5.17 illustrates the variation of the relative error in the predicted separation point with respect to the estimation obtained from the finest mesh. Since the exact separation position was not measured during the experimental campaign, the highest element number mesh result was determined as the reference to calculate error metrics.

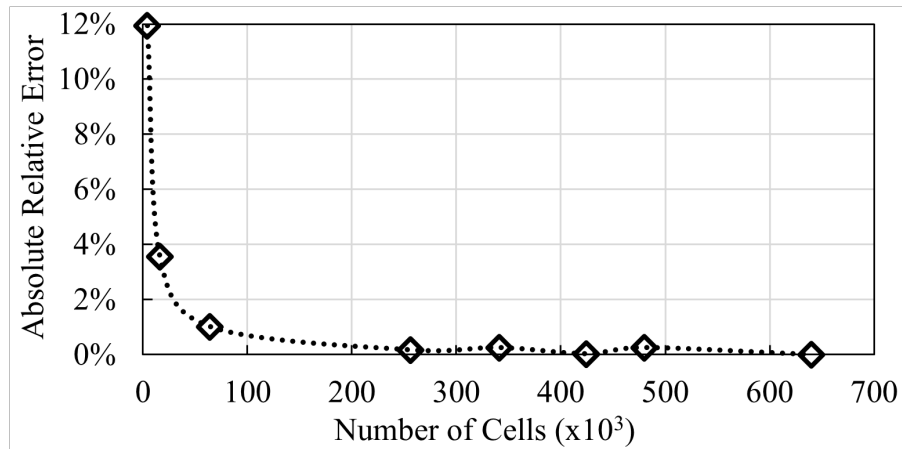


Figure 5.17 : Relative error of predicted separation position for each computational grid.

Computational grid containing 426,000 finite-volume cells was identified as the optimal configuration, providing sufficient element count and quality for the computations, presented in Figure 5.18. The maximum y^+ value on the wall is maintained below one to ensure consistency with the wall treatment approach. Menter et al.'s [158] recommendation of keeping 1.05 growth rate in the wall-normal direction of the computational grid for flows involving separation and reattachment is strictly enforced across the entire domain. Moreover, particular attention is devoted to preserving high mesh orthogonal quality, especially over the ramp, to avoid numerical diffusion that could adversely impact the numerical accuracy.

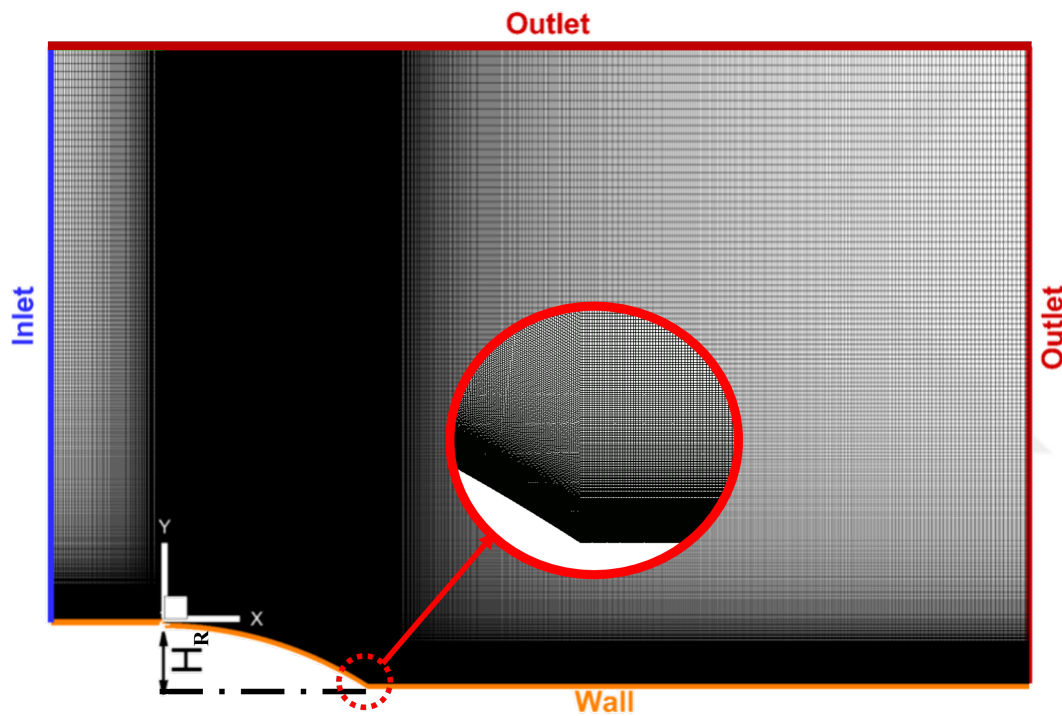


Figure 5.18 : Computational grid used in the numerical investigations of CBFS experiment.

Pressure data were measured along the wall, commencing at the ramp onset and extending to $1.5 H_R$. Additional measurements downstream of the ramp covering approximately $10 H_R$ were conducted. The flow accelerates as it approaches the ramp, attains a constant pressure coefficient, and then decelerates. In the deceleration region preceding separation, the pressure gradient remains approximately uniform. Figure 5.19 presents a comparison between the measured and computed C_p distribution along the wall.

Similar to observations in the previously investigated CBFS case, SST model underpredicts C_p values at the beginning and end of the ramp, while RNG-SST

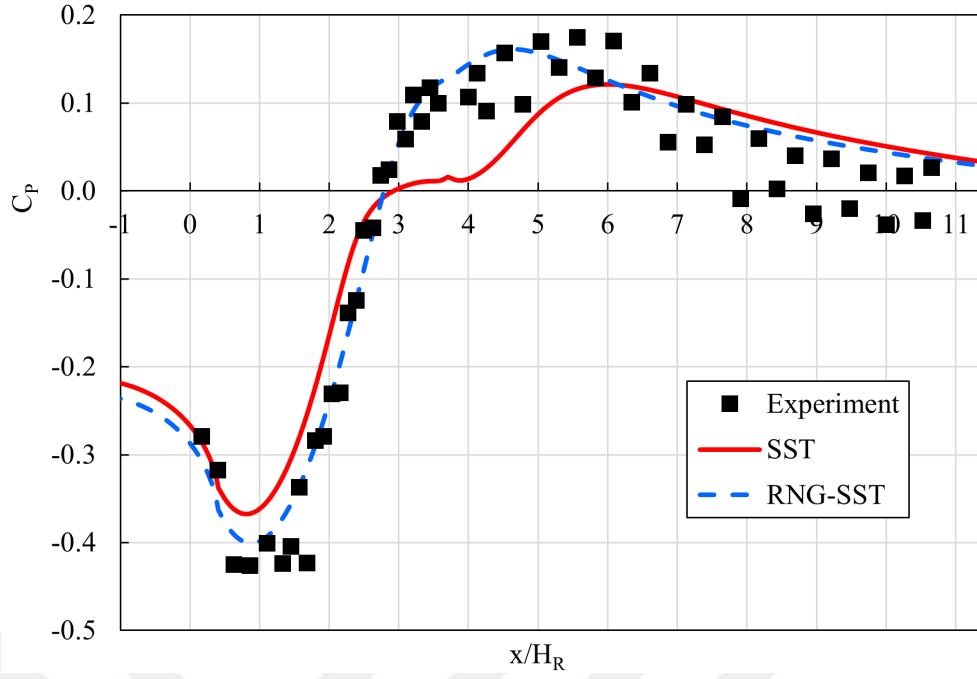


Figure 5.19 : C_p distribution along the wall.

shows higher consistency with experimental measurements. In the experiments, the maximum C_p was obtained on the plate downstream of the ramp ($x/H_R \approx 5.5$), but in the RNG-SST simulation earlier. In contrast, the SST model predicted this peak further downstream than the measured location.

Boundary layer wall-normal velocity profiles are acquired by a hot-wire anemometer at 15 streamwise stations and compared with the numerical results conducted with both SST and modified SST models in Figure 5.20.

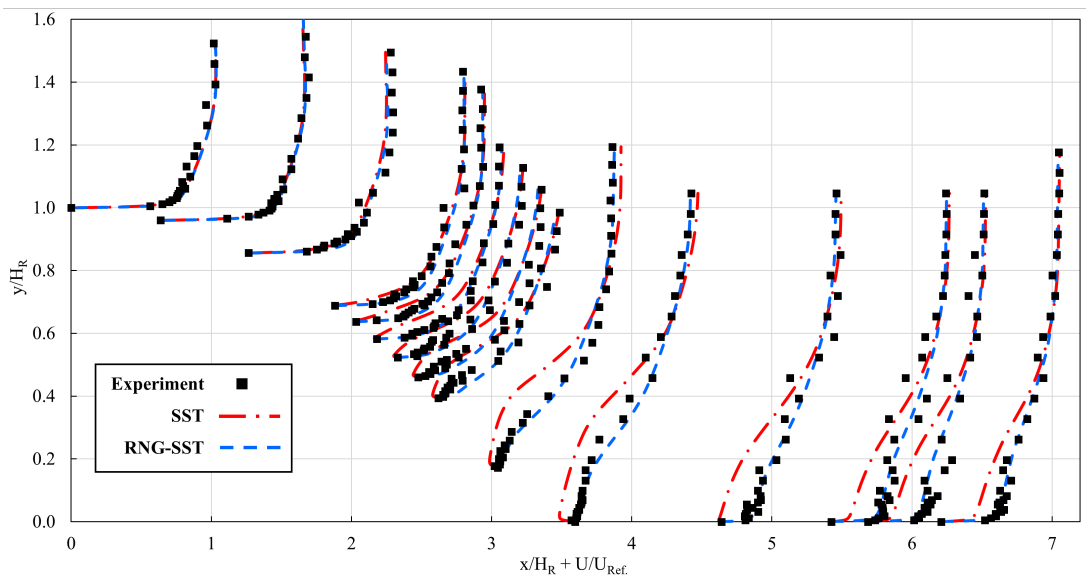


Figure 5.20 : Mean velocity distribution in the flow field.

The boundary layer velocity profile at the upstream edge of the ramp is affected by the favorable pressure gradient at the onset of the ramp. Compared to measurements, normalized mean velocity profiles by SST are under-predicted, particularly within the boundary layer region. This discrepancy becomes more evident inside the separated region. In general, SST estimates lower velocities closer to the wall. Although both models yield similar results before separation, the difference grows significantly around the separation point. Since the separation location predicted by SST lies farther upstream than that predicted by RNG-SST, the deviation between the two models appears earlier in the velocity profiles. In contrast, predictions obtained with RNG-SST show better agreement with experimental data. As also observed in the measurements, the separation bubble is relatively small, which explains why reverse flow velocities are not strongly observed in the profiles.

Displacement and momentum thickness values were computed using the relations given in equation 5.28 and 5.29, based on the von Kármán momentum integral theory [201]. For the SST and RNG-SST simulations, velocity distributions were evaluated at identical locations, and the corresponding displacement and momentum thickness values were calculated.

$$\delta^* = \int_0^\infty \left(1 - \frac{u}{U_{Ref.}}\right) dy \quad (5.28)$$

$$\theta = \int_0^\infty \frac{u}{U_{Ref.}} \left(1 - \frac{u}{U_{Ref.}}\right) dy \quad (5.29)$$

In Figure 5.21, predicted and non-dimensionalized boundary layer parameters using both models were compared with the experimental measurements. The evaluation of boundary layer parameters enables a more detailed interpretation of the differences observed in velocity profiles. For the SST model, displacement thickness values calculated at the same measurement stations show progressively increasing deviation from the experimental data beyond $x/H_R \approx 2$. In contrast, RNG-SST model yields predictions consistent with experiments up to $x/H_R = 3.6$. A similar trend is observed for the momentum thickness results. Unlike the displacement thickness, the RNG-SST model maintains good agreement with experimental momentum thickness values at $x/H_R = 3.6$, while noticeable discrepancies begin to appear downstream of $x/H_R = 4.64$.

Since wall shear stresses are not measured in the experimental campaign, the exact location of the separation point could not be determined. A more consistent assessment

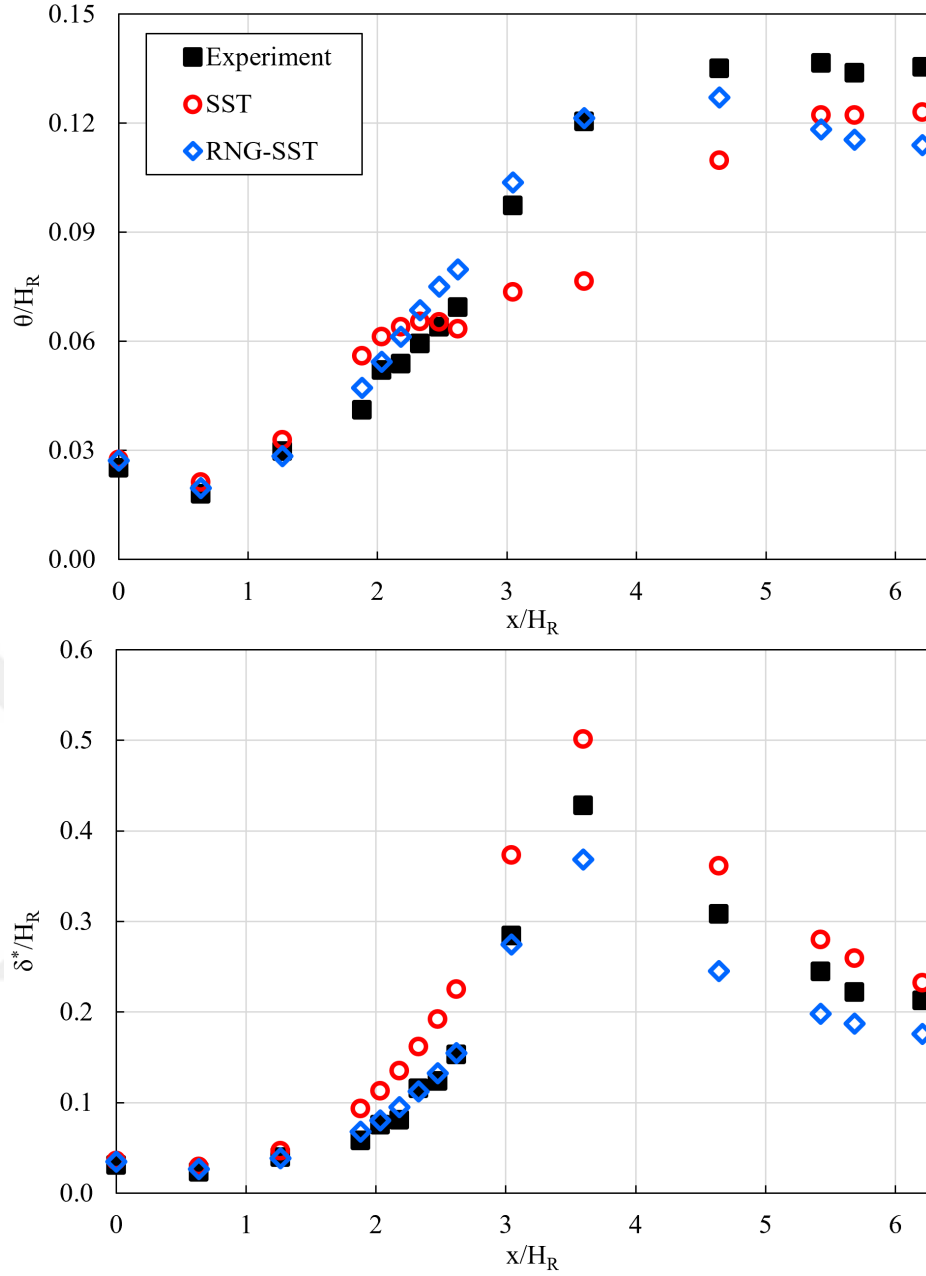


Figure 5.21 : Dimensionless momentum thickness (top) and displacement thickness (bottom) results.

was obtained by using the experimentally derived displacement and momentum thickness values in conjunction with the Sandborn [202] correlation to identify the flow separation stations. Figure 5.22 presents the calculated shape factor (H) values based on Sandborn correlation.

Flow separation occurs at $x/H_R = 2.62, 3.05,$ and 3.60 . Velocity measurements at the same stations also indicated reverse flow in the vicinity of the wall, consistent with calculated shape factors. The separation and reattachment locations predicted by the RANS models investigated in this study are summarized in Table 5.8.

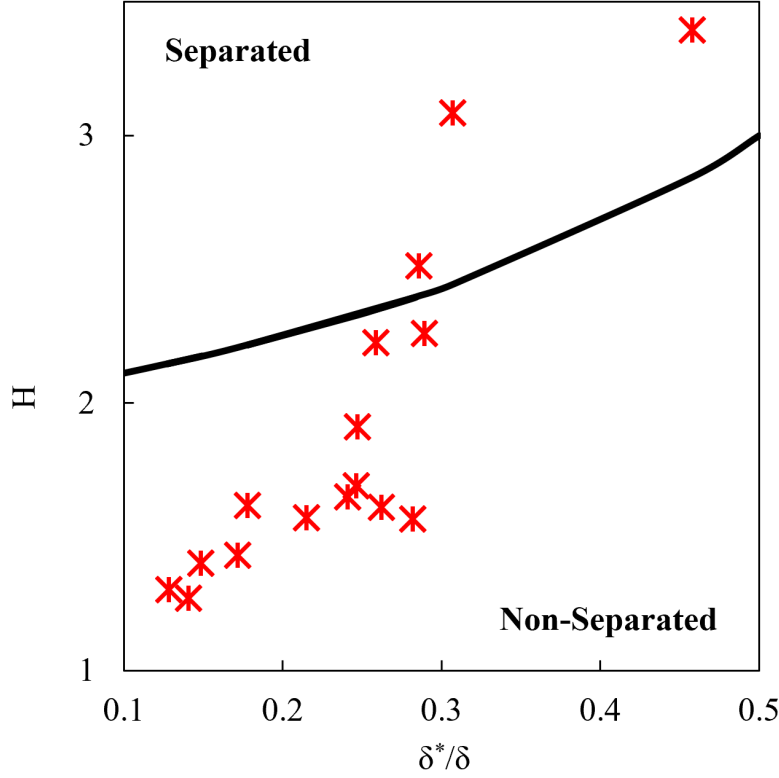


Figure 5.22 : Shape factor results calculated at measurement locations.

Table 5.8 : Separation and reattachment locations of experimental campaign on curved backward-facing step.

Result	$\frac{x_{Sep.}}{H_R}$	$\frac{x_{Reattach.}}{H_R}$
Experiment	$2.48 \leq \frac{x_{Sep.}}{H_R} \leq 2.62$	$3.60 \leq \frac{x_{Reattach.}}{H_R} \leq 4.64$
SST	1.92	4.85
RNG-SST	2.71	3.66

The separation point predicted with the SST falls upstream of the range determined using the shape factor, indicating an under-prediction of the separation position. In contrast, the RNG-SST predicts a later onset of separation and shows an over-predicted location. Considering that the RNG-SST model also over-predicted the separation point in the previously studied CBFS case, this behavior can be regarded to be consistent. While the SST is generally expected to perform better in predicting separation than reattachment, its earlier prediction in this case may be attributed to the open-channel configuration used in the experimental investigation. This reasoning also explains why SST estimates lower momentum thickness and higher displacement thickness compared to the experiments after $x/H_R = 2$. Similarly, the region where SST begins to calculate the magnitude of mean velocity profiles lower than measured in the experiment coincides with the location where flow separation occurs in the

numerical investigation. On the other hand, delayed separation prediction of RNG-SST relative to the experimental data can be explained by its enhanced ability to capture turbulence production under adverse pressure gradients along a continuous surface, allowing the flow to remain attached to the wall for a longer distance. It can be inferred that the proposed modification to the SST by implementing RNG theory of turbulence to enhance its reattachment prediction capability tends to delay the onset of flow separation. This outcome highlights the model's potential but also indicates the need for further refinement to achieve a balanced improvement in both separation and reattachment predictions.

As shown in Table 5.8, SST once again predicts the reattachment point farther downstream than observed in the experiments, confirming its tendency to over-predict reattachment locations for flows separating from continuous surfaces. However, the model achieves significantly lower relative error in reattachment prediction with the RNG modification. The reattachment position obtained with RNG-SST falls within the experimentally measured range. As illustrated in Figure 5.21, the reduction in displacement thickness predicted by RNG-SST around the reattachment region can be attributed to the transfer of energy from the shear layer into the inner region. Following flow separation, turbulence production intensifies across the flow field, and the RNG enhancement facilitates redistribution of energy from dissipation back into production and diffusion. This mechanism promotes the formation of the splat effect and enables reattachment to occur within the experimentally observed range. A similar influence is observed on momentum thickness, though with a delayed onset. Even after reattachment, the model continues to calculate slightly lower momentum thickness error. The strengthened interaction of high-energy eddies near the wall in the vicinity of reattachment leads to thinner boundary layer parameters compared to experimental measurements. Even though the exact location of the separation point in the experiment is not precisely known, the gradual recovery of the velocity profile and the decrease in magnitude of reverse flow around $x/H_R = 3.6$ suggest that separation occurs close to this position. Considering this observation, the SST model enhancement with RNG demonstrates a consistent prediction, capturing the reattachment behavior of the flow in good agreement with the experimental campaign.

5.5 Conclusion

In the present study, RNG theory of turbulence is integrated into the widely-used SST turbulence model, named as RNG-SST coupling, intended to enhance flow field predictions for separated flows originating from continuous surfaces. The RNG modification is introduced into the dissipation equation as an additional source term, confined by a blending function active near the reattachment region. This approach allows improved representation of energy redistribution within the shear layer while maintaining the computational efficiency and robustness of Menter [143] SST 2003e model. The new formulation was assessed using several canonical cases, such as curved backward-facing step, vertical and inclined BFS, and flow over periodic hills, at various Reynolds numbers. Prior to these evaluations, a systematic analysis for the influence of imposed inlet boundary conditions was performed, suggesting that the calculation methodology for ω by Spalart and Rumsey [179] is the most reliable approach.

In the CBFS case, the RNG modification yields substantial improvements in the wake region, particularly for C_p distribution, mean velocity profiles, and turbulence kinetic energy distribution. The modification enables a more accurate representation of energy redistribution mechanisms within the separated shear layer, leading to a significant reduction in the prediction error of reattachment length compared to the standard SST model. This enhancement stems from the RNG model's ability to dynamically adjust dissipation rates in response to local flow conditions, effectively bridging the energy transfer between inner and outer layer structures. In BFS configurations, CFD analyses reveal that the model is capable of distinguishing between fundamentally different separation mechanisms. For continuous surface separation, turbulence production and dissipation processes dominate the reattachment physics, while sudden expansions introduce convective transport as the primary governing mechanism. Further investigation for relatively more complex flows involves both considering adverse and favorable pressure gradients, as shown in the results of flow over periodic hills. Although improved reattachment predictions at various Reynolds numbers has been achieved, the separation bubble formed at the onset of the secondary hill can not be captured with the proposed methodology. The model's influence on production terms can amplify vortex structures in the wake region while partially suppressing

features at the incoming position. This behavior underscores the interconnected nature of turbulence closure modifications, where improvements in one aspect may introduce secondary effects requiring further refinement.

Turbulence kinetic energy budgets revealed that the modification strengthens the physical consistency of the production to dissipation ratio, improving the prediction of the wake region. However, the increased production associated with the reorganized dissipation occasionally leads to a slight over-prediction of the separation onset, suggesting that further refinement of the blending function $F_{Reattach}$ and coefficient calibration could extend the model's range of applicability. From an implementation perspective, the model offers practical advantages due to its computational cost-effectiveness while preserving numerical stability. The single additional source term enables straightforward integration into existing CFD frameworks while preserving the convergence characteristics of the standard SST model.

The proposed turbulence model shows the model's capability and highlights it as a robust, physics-based closure that improves reattachment prediction and wake region dynamics. The RNG-SST framework fills the gap between theoretical advances in turbulence modeling and requirements in practical engineering application. The model provides a promising alternative for simulating separated flows from continuous surfaces with further calibration.



6. CONCLUSIONS AND RECOMMENDATIONS

Reynolds-Averaged Navier-Stokes turbulence models remain widely employed in CFD due to their low computational cost, though their predictive accuracy is often compromised by errors highly dependent on specific flow configurations. This thesis systematically addresses this limitation through calibration and improvement of several RANS models, specifically the standard $k - \varepsilon$, Reynolds Stress Model, RNG $k - \varepsilon$, and SST $k - \omega$, applied to turbulent round jets, axial compressor cascades, and various problems involving flow separation from a continuous surface, including curved backward-facing step, backward-facing step, and periodic hills. The resulting enhancements, achieved through adjusted closure coefficients and presented model variants, stem from an extensive methodology combining measurement correlation, quantitative and qualitative engineering assessment, and rigorous mathematical formulation.

For turbulent round free jets, the closure coefficients of the standard $k - \varepsilon$ and the RSM were optimized using a multi-objective genetic algorithm. Jet flow indicators at various Reynolds numbers, including jet spreading rate, centerline velocity decay, and centerline velocity and turbulence intensity profiles, were used as evaluation criteria. Sensitivity analysis clarified how individual coefficients influence both the model equations and the governing turbulence mechanics. Tuning $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, and C_{μ} in the standard $k - \varepsilon$ yielded consistent improvements in centerline velocity and half-width across Reynolds numbers. An illustrative optimum set produced notably better half-width and velocity profiles at all axial stations. A half-width oriented optimization strategy reduced errors in velocity decay and spreading rate at $Re = 10000$ and $Re = 20000$. Turbulence budget indicated that the calibrated coefficients strengthened effective diffusion at the cost of shifted advection. Nonetheless, altered centerline velocity and turbulence intensity profiles show that optimizing a single target can disturb the balance governing other quantities and underscoring the need for multi-objective strategies. These results support viewing $k - \varepsilon$ coefficients as case-dependent calibration parameters rather than universal constants.

For the axial compressor cascade, a locally adaptive RNG variant, called RNG F_c , was developed by introducing a spatially varying function for η . Relative to default RNG and SST models, RNG F_c reduced errors in total pressure loss and flow characteristics in the wake region while maintaining robustness and computational cost efficiency across near-wall resolutions, wall treatment approaches, and different finite-volume solvers. Although transition onset was not captured perfectly, models capable of laminar-to-turbulent transition prediction remain preferable for regimes where intermittency dominates losses. The implementation was designed to be reasonable for different finite-volume solvers and cost-effective. Integration guidance for common finite-volume solvers was provided.

To overcome the SST model's known deficiency in predicting delayed reattachment for flows separating from a continuous surface, the RNG-SST model was developed through the coupling of the RNG theory of turbulence with the SST model equations. This hybrid approach demonstrated substantial improvement in separated flow predictions, reducing relative error in reattachment position by up to half across various Reynolds numbers for flows over curved backward-facing step, backward-facing step, and periodic hills.

This thesis establishes that RANS turbulence closures can be systematically improved for specific flow problems. For self-similar flows like jets, coefficient calibration reduces computational errors not only within the calibrated condition but also extends to broader flow regimes. When numerous closure coefficients make optimization computationally expensive, sensitivity analysis enables the identification of the most influential parameters for targeted adjustment. Successful overhauling requires meticulous attention to potential uncertainties beyond the turbulence model, including boundary condition specification, grid generation, and discretization schemes. Furthermore, validation must rely on numerical benchmark data from scale-resolving methods (e.g., LES, DNS) or reliable experiments to ensure meaningful outcomes.

At the same time, single-objective optimization may yield non-physical or numerically unstable computations by distorting the coupled momentum and turbulence transport. Fine-tuned coefficients might also increase model induced error when directly imposed on different investigations. Broader generalizability requires simultaneous tuning against multiple flow quantities across diverse regimes. Holistic calibration

or reinforced holistic calibration approaches are therefore recommended. Local, physics-informed augmentations such as RNG F_c improves CFD predictions in turbomachinery without extra computational cost and with robustness to meshing and wall modeling approaches, but they still require broader validation.

The generalizability of any turbulence model increases with the number and diversity of validation cases. Consequently, the RNG F_c model should be evaluated on additional applications such as turbine blades and airfoils. Likewise, the RNG-SST model requires assessment on more complex geometries and challenging flow types. Although the RNG-SST improves reattachment prediction, occasional delays in separation indicate that refining the blending function $F_{Reattach}$ and further coefficient calibration may extend applicability.



REFERENCES

- [1] **Newton, I.** (1846). *Newton's Principia: The Mathematical Principles of Natural Philosophy*. New-York: Daniel Adee.
- [2] **Bernoulli, D.** (1968). *Hydrodynamics*. Dover Publications.
- [3] **Euler, L.** (1761). Principia motus fluidorum, *Novi Commentarii academiae scientiarum Petropolitanae*, 6, 271–311.
- [4] **Navier, C.L.M.H.** (1822). Mémoire sur les lois du mouvement des fluides, *Mémoires de l'Académie Royale des Sciences de l'Institut de France*, pp. 389–440.
- [5] **Stokes, G.G.** (1845). On the theories of the internal friction of fluids in motion and of the equilibrium and motion of elastic solids, *Transactions of the Cambridge Philosophical Society*, 8, 287.
- [6] **White, F.M.** (2006). *Viscous Fluid Flow*. No. 3rd. McGraw-Hill.
- [7] **Cengel, Y.A. and Cimbala, J.M.** (2017). *Fluid Mechanics: Fundamentals and applications*. No. 4th. McGraw-Hill Education.
- [8] **Boussinesq, J.V.** (1877). *Essai Sur La Théorie Des Eaux Courantes*. Impr. Nationale, Paris.
- [9] **Reynolds, O.** (1895). On the dynamical theory of incompressible viscous fluids and the determination of the criterion, *Philosophical Transactions of the Royal Society of London*, 186, 123–164, doi : 10.1098/rsta.1895.0004.
- [10] **Prandtl, L.** (1904). Über flüssigkeitsbewegung bei sehr kleiner reibung, *In Verhandlungen des III. Internationalen Mathematiker Kongresses, Heidelberg, 8-13 August 1904*, B. G. Teubner, Leipzig, pp. 485–491.
- [11] **Prandtl, L.** (1926). Bericht über untersuchungen zur ausgebildeten turbulenz, *Zeitschrift für Angewandte Mathematik und Mechanik*, 5(2), 136–139.
- [12] **Prandtl, L.** (1926). Über die ausgebildete turbulenz, *In Lecture presented at International Congress for Applied Mechanics (Zurich)*.
- [13] **Richardson, R.G.D.** (1917). A new method in boundary problems for differential equations, *Transactions of the American Mathematical Society*, 18(4), 498–518, doi : 10.1090/S0002-9947-1917-1501081-8.
- [14] **Kolmogorov, A.N., Levin, V., Hunt, J.C.R., Phillips, O.M., and Williams, D.** (1941). The local structure of turbulence in incompressible viscous fluid for very large reynolds numbers, *Proceedings of the Royal Society of*

London. Series A: Mathematical and Physical Sciences, 434(1890), 9–13,
doi : 10.1098/rspa.1991.0075.

- [15] **Davidson, L.** (2020). *Fluid mechanics, turbulent flow and turbulence modeling*. Chalmers University of Technology, Sweden.
- [16] **Duraisamy, K.** (2025). *Data Driven Analysis and Modeling of Turbulent Flows*. Academic Press, Elsevier.
- [17] **Wilcox, D.C.** (2006). *Turbulence Modeling for CFD*. DCW Industries, Inc., La Canada CA, (3.. ed.).
- [18] **Smagorinsky, J.** (1963). General circulation experiments with the primitive equation: I. the basic experiment, *Monthly Weather Review*, 91(3), 99–164.
- [19] **Orszag, S.A.** (1970). Analytical theories of turbulence, *Journal of Fluid Mechanics*, 41(2), 363–386, doi : 10.1017/S0022112070000642.
- [20] **Chou, P.Y.** (1945). On velocity correlations and the solutions of the equations of turbulent fluctuation, *Quarterly of Applied Mathematics*, 3, 38–54.
- [21] **Rotta, J.** (1951). Statistische theorie nichthomogener turbulenz, *Zeitschrift für Physik A*, 129(6), 547–572.
- [22] **Harlow, F.H. and Nakayama, P.I.** (1967). Turbulence transport equations, *Physics of Fluids*, 10(11), 2323–2332, doi : 10.1063/1.1762039.
- [23] **Harlow, F.H. and Nakayama, P.I.** (1968). Transport of turbulence energy decay rate, (Technical Report), Los Alamos Scientific Laboratory of the University Of California.
- [24] **Patankar, S.V. and Spalding, D.B.** (1972). A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows, *International Journal of Heat and Mass Transfer*, 15(10), 1787–1806, doi : 10.1016/0017-9310(72)90054-3.
- [25] **Jones, W.P. and Launder, B.E.** (1972). The prediction of laminarization with a two-equation model of turbulence, *International Journal of Heat and Mass Transfer*, 15(2), 301–314, doi : 10.1016/0017-9310(72)90076-2.
- [26] **Turutoglu, C.** (2025). *GitHub: RNG Fc Turbulence Model*. Retrieved July 10, 2025 from: https://github.com/CTurutoglu/RNG_Fc.
- [27] **Wyganski, I. and Fiedler, H.** (1969). Some measurements in the self-preserving jet, *Journal of Fluid Mechanics*, 38(3), 577–612, doi : 10.1017/S0022112069000358.
- [28] **Bradshaw, P.** (1966). The effect of initial conditions on the development of a free shear layer, *Journal of Fluid Mechanics*, 26(2), 225–236, doi : 10.1017/S0022112066001204.

- [29] **Bradshaw, P., Ferriss, D.H., and Johnson, R.F.** (1964). Turbulence in the noise-producing region of a circular jet, *Journal of Fluid Mechanics*, 19 (4), 591–624, doi : 10.1017/S0022112064000945.
- [30] **Launder, B.E. and Sharma, B.I.** (1974). Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc, *Letters in Heat and Mass Transfer*, 1(2).
- [31] **Kyle, D.M. and Sreenivasan, K.R.** (1993). The instability and breakdown of a round variable-density jet, *Journal of Fluid Mechanics*, 249, 619–664, doi : 10.1017/S0022112093001314.
- [32] **Launder, B.E. and Spalding, D.B.** (1974). The numerical computation of turbulent flows, *Computer Methods in Applied Mechanics and Engineering*, 3(2), 269–289, doi : 10.1016/0045-7825(74)90029-2.
- [33] **Ouyang, Y., Tang, K.L., Xiang, Y., Zou, H.K., Chu, G.W., Agarwal, R.K., and Chen, J.F.** (2019). Evaluation of various turbulence models for numerical simulation of a multiphase system in a rotating packed bed, *Computers & Fluids*, 194, 104296, doi : 10.1016/j.compfluid.2019.104296.
- [34] **Ramesh, S., Xu, Z., Rivero, I.V., and Cormier, D.R.** (2023). Computational fluid dynamics and experimental validation of aerosol jet printing with multi-stage flow focusing lenses, *Journal of Manufacturing Processes*, 95, 312–329, doi : 10.1016/j.jmapro.2023.03.035.
- [35] **Wang, D., Du, G., and Wu, J.** (2005). Numerical experimental study on the 3-d flow field around a van with a dome for energy saving, *Energy Conversion and Management*, 46(6), 833–846, doi : 10.1016/j.enconman.2004.06.014.
- [36] **Townsend, A.A.R.** (1956). *The Structure of Turbulent Shear Flow*. Cambridge University Press.
- [37] **Champagne, F.H., Harris, V.G., and Corrsin, S.** (1970). Experiments on nearly homogeneous turbulent shear flow, *Journal of Fluid Mechanics*, 41(1), 81–139, doi : 10.1017/S0022112070000538.
- [38] **Harris, V.G., Graham, J.A.H., and Corrsin, S.** (1977). Further experiments in nearly homogeneous turbulent shear flow, *Journal of Fluid Mechanics*, 81(4), 657–687, doi : 10.1017/S0022112077002286.
- [39] **Tavoularis, S. and Corrsin, S.** (1981). Experiments in nearly homogeneous turbulent shear flow with a uniform mean temperature gradient. part 2. the fine structure, *Journal of Fluid Mechanics*, 104, 349–367, doi : 10.1017/S0022112081002942.
- [40] **Komatsu, T. and Matsunaga, N.** (1986). Defect of k-turbulence model and its improvements, *In Proceedings of the 30th Japan Conference on Hydraulics*.
- [41] **Chen, C.J. and Jaw, S.Y.** (1997). *Fundamentals of Turbulence Modeling*. Taylor & Francis.

- [42] **Rohr, J.J., Itsweire, E.C., Helland, K.N., and Atta, C.W.V.** (1988). An investigation of the growth of turbulence in a uniform-mean-shear flow, *Journal of Fluid Mechanics*, 187, 1–33, doi : 10.1017/S002211208800031X.
- [43] **Platteeuw, P.D.A., Loeven, G.J.A., and Bijl, H.** (2008). Uncertainty quantification applied to the k-epsilon model of turbulence using the probabilistic collocation method, *In 49th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference, 16th AIAA/ASME/AHS Adaptive Structures Conference, 10th AIAA Non-Deterministic Approaches Conference, 9th AIAA Gossamer Spacecraft Forum, 4th AIAA Multidisciplinary Design Optimization Specialists Conference*, No. AIAA Paper No. AIAA 2008-2150, doi : 10.2514/6.2008-2150.
- [44] **Hossain Khan, A., Md. Shafiqul Islam, and Uddin Sazzad, I.** (2021). Calibration of $\kappa - \epsilon$ turbulence model for thermal-hydraulic analyses in rib-roughened narrow rectangular channels using genetic algorithm, *SN Applied Sciences*, 3(6), 678, doi : 10.1007/s42452-021-04645-x.
- [45] **Mok, M.C.H., Yeoh, C.V., Tan, M.K., and Foo, J.J.** (2023). Space-filling single square and square fractal grids induced turbulence: Reynolds stress model parameters-optimization, *Results in Engineering*, 17, 100806, doi : 10.1016/j.rineng.2022.100806.
- [46] **Zhao, Y., Akolekar, H.D., Weatheritt, J., Michelassi, V., and Sandberg, R.D.** (2020). Rans turbulence model development using cfd-driven machine learning, *Journal of Computational Physics*, 411, 109413, doi : 10.1016/j.jcp.2020.109413.
- [47] **Ray, J., Lefantzi, S., Arunajatesan, S., and Dechant, L.** (2016). Bayesian parameter estimation of a $k - \epsilon$ model for accurate jet-in-crossflow simulations, *AIAA Journal*, 54(8), 2432–2448, doi : 10.2514/1.J054758.
- [48] **Matsui, K., Perez, E., Kelly, T.R., Tani, N., and Jemcov, A.** (2022). Calibration of modified spalart-allmaras model parameters for linear compressor cascade corner flow, *Journal of Thermal Science*, 31(1), 163–172, doi : 10.1007/s11630-022-1566-0.
- [49] **Ray, J., Dechant, L., Lefantzi, S., Ling, J., and Arunajatesan, S.** (2018). Robust bayesian calibration of a $k - \epsilon$ model for compressible jet-in-crossflow simulations, *AIAA Journal*, 56(12), 4893–4909, doi : 10.2514/1.J057204.
- [50] **Chowdhary, K., Hoang, C., Lee, K., Ray, J., Weirs, V.G., and Carnes, B.** (2022). Calibrating hypersonic turbulence flow models with the hifire-1 experiment using data-driven machine-learned models, *Computer Methods in Applied Mechanics and Engineering*, 401, 115396, doi : 10.1016/j.cma.2022.115396.
- [51] **Wawrzak, K. and Tyliczszak, A.** (2022). Influence of the mesh topology on the accuracy of modelling turbulent natural and excited round jets at different initial turbulence intensities, *Applied Sciences*, 12(21), 11244.

- [52] **ANSYS.** (2023). *Ansys® Academic Research, Release 23.1, Help System, CFX Solver Theory Guide*. ANSYS, Inc.
- [53] **Yilmaz, S., Erdem, D., and Acar, H.** (2010). Dairesel jetlerde reynolds sayısının etkileri, *In Ulusal Havacılık ve Uzay Konferansı*.
- [54] **Corrsin, S. and Uberoi, M.S.** (1950). Further experiments on the flow and heat transfer in a heated turbulent air jet, (Technical Report), NACA.
- [55] **Hinze, J.O. and Van der Hegge Zijnen, B.G.** (1949). Transfer of heat and matter in the turbulent mixing zone of an axially symmetrical jet, *Flow Turbulence and Combustion*, 1(1), 435, doi : 10.1007/BF02120346.
- [56] **Bradshaw, P.M.D.** (1970). *Experimental Fluid Mechanics*. Pergamon Press.
- [57] **Hussein, H.J. and George, W.K.** (1989). Measurement of small scale turbulence in an axisymmetric jet using moving hot-wires, *In Proceedings of the 7th Symposium on Turbulent Shear Flows*, Vol. 2.
- [58] **Hussein, H.J., George, W.K., and Capp, S.P.** (1988). Comparison between hot-wire and burst-mode lda velocity measurements in a fully-developed turbulent jet, *In 26th AIAA Aerospace Sciences Meeting*, Vol. 26.
- [59] **Lemieux, G.P. and Oosthuizen, P.H.** (1985). Experimental study of the behavior of plane turbulent jets at low reynolds numbers, *AIAA Journal*, 23(12), 1845–1846, doi : 10.2514/3.9183.
- [60] **Fellouah, H., Ball, C.G., and Pollard, A.** (2009). Reynolds number effects within the development region of a turbulent round free jet, *International Journal of Heat and Mass Transfer*, 52(17), 3943–3954, doi : 10.1016/j.ijheatmasstransfer.2009.03.029.
- [61] **Todde, V., Spazzini, P., and Sandberg, M.** (2009). Experimental analysis of low-reynolds number free jets: Evolution along the jet centerline and reynolds number effects, *Experiments in Fluids*, 47, 179–194, doi : 10.1007/s00348-009-0655-0.
- [62] **Dimotakis, P.** (2000). The mixing transition in turbulent flows, *Journal of Fluid Mechanics*, 409, 69–98, doi : 10.1017/S0022112099007946.
- [63] **Suresh, P.R., Srinivasan, K., Sundararajan, T., and Das, S.** (2008). Reynolds number dependence of plane jet development in the transitional regime, *Physics of Fluids*, 20, 0444105, doi : 10.1063/1.2904994.
- [64] **Deo, R., Mi, J., and Nathan, G.** (2008). The influence of reynolds number on a plane jet, *Physics of Fluids*, 20, 075108, doi : 10.1063/1.2959171.
- [65] **Mi, J., Xu, M., and Zhou, T.** (2013). Reynolds number influence on statistical behaviors of turbulence in a circular free jet, *Physics of Fluids*, 25, 075101, doi : 10.1063/1.4811403.

- [66] **Celis, C., Barreto, C., Lopes, G., Carneiro, J., Abrantes, J., Barros, J., and Nieckele, A.** (2005). Evaluation of different turbulence models to predict a turbulent free jet, *In Proceedings of 18th International Congress of Mechanical Engineering, COMEB*.
- [67] **Kmecová, M., Sikula, O., and Krajčík, M.** (2019). Circular free jets: Cfd simulations with various turbulence models and their comparison with theoretical solutions, *IOP Conference Series Materials Science and Engineering*, 471, 062045, doi : 10.1088/1757-899X/471/6/062045.
- [68] **Weaver, D. and Miskovic, S.** (2021). A study of rans turbulence models in fully turbulent jets: A perspective for cfd-dem simulations, *Fluids*, 6, 271, doi : 10.3390/fluids6080271.
- [69] **Zidouni, F., Danciu, D.V., Lucas, D., Salah, A., and Mataoui, A.** (2011). Comparison of two turbulence models in simulating an axisymmetric jet evolving into a tank, *Journal of Physics: Conference Series*, 318(4), 042035, doi : 10.1088/1742-6596/318/4/042035.
- [70] **Speziale, C., Sarkar, S., and Gatski, T.** (1991). Modelling the pressure-strain correlation of turbulence: An invariant dynamical systems approach, *Journal of Fluid Mechanics*, 227, 245–272, doi : 10.1017/S0022112091000101.
- [71] **Meslem, A., Bode, F., Croitoru, C., and Nastase, I.** (2014). Comparison of turbulence models in simulating jet flow from a cross-shaped orifice, *European Journal of Mechanics, B: Fluids*, 44, 100–120, doi : 10.1016/j.euromechflu.2013.11.006.
- [72] **Miltner, M., Jordan, C., and Harasek, M.** (2015). Cfd simulation of straight and slightly swirling turbulent free jets using different rans-turbulence models, *Applied Thermal Engineering*, 89, 1117–1126, doi : 10.1016/j.applthermaleng.2015.05.048.
- [73] **Li, Z., He, C., and Liu, Y.** (2023). A data-driven reynolds-number-dependent model for turbulent mean flow prediction in circular jets, *Physics of Fluids*, 35, 085130, doi : 10.1063/5.0160410.
- [74] **Pope, S.B.** (2000). *Turbulent Flows*. Cambridge University Press.
- [75] **Birch, S., Lyubimov, D., Secundov, A., and Yakubovsky, K.** (2003). Numerical modeling requirements for coaxial and chevron nozzle flows, *In 9th Aeroacoustics Conference and Exhibit*, No. AIAA Paper No. AIAA 2003-3287, doi : 10.2514/6.2003-3287.
- [76] **Guillas, S., Glover, N., and Malki-Epshtein, L.** (2014). Bayesian calibration of the constants of the $k - \varepsilon$ turbulence model for a cfd model of street canyon flow, *Computer Methods in Applied Mechanics and Engineering*, 279, 536–553.

- [77] **Rocha, P., Rocha, H.H., Carneiro, F.O., Silva, M.E., and Bueno, A.** (2013). $k-\omega$ sst (shear stress transport) turbulence model calibration: A case study on a small scale horizontal axis wind turbine, *Energy*, 65, 412–418, doi : 10.1016/j.energy.2013.11.050.
- [78] **Rocha, P., Rocha, H., Carneiro, F., Silva, M., and Freitas, C.** (2016). A case study on the calibration of the $k-\omega$ sst (shear stress transport) turbulence model for small scale wind turbines designed with cambered and symmetrical airfoils, *Energy*, 97, 144–150, doi : 10.1016/j.energy.2015.12.081.
- [79] **Shirzadi, M., Mirzaei, P., and Naghashzadegan, M.** (2017). Improvement of k -epsilon turbulence model for cfd simulation of atmospheric boundary layer around a high-rise building using stochastic optimization and monte carlo sampling technique, *Journal of Wind Engineering and Industrial Aerodynamics*, 171, 366–379, doi : 10.1016/j.jweia.2017.10.005.
- [80] **Mauro, S., Lanzafame, R., Messina, M., and Pirrello, D.** (2017). Transition turbulence model calibration for wind turbine airfoil characterization through the use of a micro-genetic algorithm, *International Journal of Energy and Environmental Engineering*, 8, 359–374, doi : 10.1007/s40095-017-0248-2.
- [81] **Shirzadi, M., Mirzaei, P., and Tominaga, Y.** (2020). Rans model calibration using stochastic optimization for accuracy improvement of urban airflow cfd modeling, *Journal of Building Engineering*, 32, 101756, doi : 10.1016/j.jobbe.2020.101756.
- [82] **Da Ronch, A., Panzeri, M., Drofelnik, J., and d'Ippolito, R.** (2020). Sensitivity and calibration of turbulence model in presence of epistemic uncertainties, *CEAS Aeronautical Journal*, 11, 33–47, doi : 10.1007/s13272-019-00389-y.
- [83] **Yoder, D. and Orkwis, P.** (2021). On the use of optimization techniques for turbulence model calibration, *Computers & Fluids*, 214, 104752, doi : 10.1016/j.compfluid.2020.104752.
- [84] **Romanova, D., Ivanov, O., Trifonov, V., Ginzburg, N., Korovina, D., Ginzburg, B., Koltunov, N., Eglit, M., and Strijhak, S.** (2022). Calibration of the $k - \omega$ sst turbulence model for free surface flows on mountain slopes using an experiment, *Fluids*, 7, 111, doi : 10.3390/fluids7030111.
- [85] **Song, X., Yu, Z., Liu, C., and Cheng, G.** (2022). Calibration of rans model constant based on data assimilation and accurate simulation of separated flow, *AIP Advances*, 12, 095324, doi : 10.1063/5.0103253.
- [86] **Subbian, G., Souza, A., Radespiel, R., Zander, E., Friedman, N., Moshagen, T., Moioli, M., Breitsamter, C., and Sørensen, K.** (2020). Calibration of an extended eddy viscosity turbulence model using uncertainty quantification, *In AIAA SciTech Forum*, No. AIAA Paper No. AIAA 2020-1031, doi : 10.2514/6.2020-1031.

- [87] **Yildizeli, A. and Cadirci, S.** (2023). Calibration of the $k-\omega$ sst turbulence model for backward facing step problem using multi-objective optimization, *In International Mechanical Engineering Congress and Exposition*, Vol. 9: Fluids Engineering, p. V009T10A031. ASME, doi : 10.1115/IMECE2023-115019.
- [88] **He, X., Yuan, C., Gao, H., Chen, Y., and Zhao, R.** (2023). Calibration of turbulent model constants based on experimental data assimilation: Numerical prediction of subsonic jet flow characteristics, *Sustainability*, 15, 10219, doi : 10.3390/su151310219.
- [89] **Liang, S., Guo, M.M., Yi, R.M., Tian, Y., Song, W.Y., Le, L.J., and Zhang, H.** (2023). Turbulence model parameter calibration method based on the combination of deep neural network surrogate model and genetic algorithm in supersonic flow over cavity-ramp, *Research Square*, doi : 10.21203/rs.3.rs-2897884/v1.
- [90] **Usov, L., Troshin, A., Anisimov, K., and Sabelnikov, V.** (2023). Calibration of a near-wall differential reynolds stress model using the updated direct numerical simulation data and its assessment, *Energies*, 16, 6826, doi : 10.3390/en16196826.
- [91] **Younoussi, S. and Ettaouil, A.** (2024). Calibration method of the $k - \omega$ sst turbulence model for wind turbine performance prediction near stall condition, *Heliyon*, 10, e24048, doi : 10.1016/j.heliyon.2024.e24048.
- [92] **D'Auria, F.** (2017). *Thermal Hydraulics in Water-Cooled Nuclear Reactors*. Woodhead Publishing.
- [93] **Lovella, Y.G., Moya, I.H., Jayasuriya, J., and Blondeau, J.** (2024). Reynolds-average navier-stokes turbulence models assessment: A case study of $ch_4/h_2/n_2$ -air reacting jet, *Heliyon*, 10, e26956, doi : 10.1016/j.heliyon.2024.e26956.
- [94] **Launder, B.E., Reece, G., and Rodi, W.** (1975). Progress in the development of a reynolds stress turbulence closure, *Journal of Fluid Mechanics*, 68, 537–566, doi : 10.1017/S0022112075001814.
- [95] **Faghani, E., Saemi, S., Maddahian, R., and Farhanieh, B.** (2011). On the effect of inflow conditions in simulation of a turbulent round jet, *Archive of Applied Mechanics*, 81, 1439–1453, doi : 10.1007/s00419-010-0494-8.
- [96] **ANSYS.** (2023). *Ansys® Academic Research, Release 23.1, Help System, DesignXplorer User's Guide*. ANSYS, Inc.
- [97] **Panchapakesan, N.R. and Lumley, J.L.** (1993). Turbulence measurements in axisymmetric jets of air and helium. part 1. air jet, *Journal of Fluid Mechanics*, 246, 197–223, doi : 10.1017/S0022112093000096.
- [98] **Gutmark, E.J., Schadow, K.C., Parr, T.P., Hanson-Parr, D.M., and Wilson, K.J.** (1987). Noncircular jets in combustion systems, *Experiments in Fluids*, 7, 248–258.

- [99] **Bogey, C. and Bailly, C.** (2006). Large eddy simulations of transitional round jets: influence of the reynolds number on flow development and energy dissipation, *Physics of Fluids*, 18, 065101, doi : 10.1063/1.2204060.
- [100] **Balajee, G. and Panchapakesan, N.** (2022). Large eddy simulations of single and multiple turbulent round jets, *Journal of Turbulence*, 23, 173–213, doi : 10.1080/14685248.2022.2051531.
- [101] **Barth, T.J. and Jespersen, D.C.** (1989). The design and application of upwind schemes on unstructured meshes, *In 27th Aerospace Sciences Meeting*, No. AIAA Paper No. AIAA 1989-366.
- [102] **le Roux, J.** (1914). Sur le problème de dirichlet, *Journal de Mathématiques Pures et Appliquées*, 10(6), 189–230.
- [103] **Phillips, H.B. and Wiener, N.** (1923). Nets and the dirichlet problem, *Studies in Applied Mathematics*, 2, 105–124, doi : 10.1002/sapm192321105.
- [104] **Lusternik, L.** (1926). On an application of the direct method in variation calculus, *Recueil de la Société Mathém. de Moscou*.
- [105] **Courant, R., Friedrichs, K., and Lewy, H.** (1967). On the partial difference equations of mathematical physics, *IBM Journal of Research and Development*, 11(2), 215–234, doi : 10.1147/rd.112.0215.
- [106] **Jones, W.P. and Launder, B.E.** (1973). The calculation of low-reynolds-number phenomena with a two-equation model of turbulence, *International Journal of Heat and Mass Transfer*, 16(6), 1119–1130, doi : 10.1016/0017-9310(73)90125-7.
- [107] **Yakhot, V. and Orszag, S.A.** (1986). Renormalization group analysis of turbulence. i. basic theory, *Journal of Scientific Computing*, 1(1), 3–51, doi : 10.1007/BF01061452.
- [108] **Reynolds, W.C.** (1987). *Fundamentals of Turbulence for Turbulence Modeling and Simulation*, (Doctoral Dissertation), Department of Mechanical Engineering, Stanford University, California.
- [109] **Yakhot, V., Orszag, S.A., Thangam, S., Gatski, T.B., and Speziale, C.G.** (1992). Development of turbulence models for shear flows by a double expansion technique, *Physics of Fluids*, 4(7), 1510–1520, doi : 10.1063/1.858424.
- [110] **Menter, F.R.** (1994). Two-equation eddy-viscosity turbulence models for engineering applications, *AIAA Journal*, 32(8), 1598–1605, doi : 10.2514/3.12149.
- [111] **Hinze, J.O.** (1987). *Turbulence*. McGraw-Hill, New York.
- [112] **Wu, J., Wang, J., and Xiao, H.** (2016). A bayesian calibration-prediction method for reducing model-form uncertainties with application in rans simulations, *Flow, Turbulence and Combustion*, 97, 761–786, doi : 10.1007/s10494-016-9725-6.

- [113] **Schaefer, J., Hosder, S., West, T., Rumsey, C., Carlson, J.R., and Kleb, W.** (2017). Uncertainty quantification of turbulence model closure coefficients for transonic wall-bounded flows, *AIAA Journal*, 55(1), 195–213, doi : 10.2514/1.J054902.
- [114] **Zhang, J. and Fu, S.** (2018). An efficient bayesian uncertainty quantification approach with application to $k-\omega-\gamma$ transition modeling, *Computers & Fluids*, 161, 211–224, doi : 10.1016/j.compfluid.2017.11.007.
- [115] **Witte, I., Stephanopoulos, K., Wray, T., and Agarwal, R.K.** (2018). Uncertainty quantification of turbulence model coefficients in openfoam and fluent for mildly separated flows, *In AIAA 2018 Fluid Dynamics Conference*.
- [116] **Zhao, Y., Yan, C., Wang, X., Liu, H., and Zhang, W.** (2019). Uncertainty and sensitivity analysis of sst turbulence model on hypersonic flow heat transfer, *International Journal of Heat and Mass Transfer*, 136, 808–820, doi : 10.1016/j.ijheatmasstransfer.2019.03.012.
- [117] **Zeng, F., Li, J., Wang, Y., Sun, M., and Yan, C.** (2022). Parametric uncertainty quantification of sst turbulence model for a shock train and pseudo-shock phenomenon, *Acta Astronautica*, 196, doi : 10.1016/j.actaastro.2022.05.002.
- [118] **Liu, H., Kong, Z., Li, G., Chen, C., Zhao, Y., and Zhang, S.** (2024). Uncertainty quantification of the standard k-epsilon turbulence model closure coefficients in predicting aerodynamics of high-speed train, *Engineering Applications of Computational Fluid Mechanics*, 18(1), 2430658, doi : 10.1080/19942060.2024.2430658.
- [119] **Xiao, H. and Cinnella, P.** (2019). Quantification of model uncertainty in rans simulations: A review, *Progress in Aerospace Sciences*, 108, 19–31, doi : 10.1016/j.paerosci.2018.10.001.
- [120] **Georgiadis, N. and Yoder, D.** (2013). Recalibration of the shear stress transport model to improve calculation of shock separated flows, *In 51st Aerospace Sciences Meeting*, No. AIAA Paper No. 2013-685.
- [121] **Edeling, W.N., Cinnella, P., and Dwight, R.P.** (2014). Predictive rans simulations via bayesian model-scenario averaging, *Journal of Computational Physics*, 275, 65–91, doi : 10.1016/j.jcp.2014.06.052.
- [122] **Georgiadis, N., Mankbadi, M.R., and Vyas, M.A.** (2014). Turbulence model effects on rans simulations of the hifire flight 2 ground test configurations, *In 52nd Aerospace Sciences Meeting*, No. AIAA Paper No. 2014-0624.
- [123] **DeChant, L.J., Ray, J., Lefantzi, S., Ling, J., and Arunajatesan, S.** (2017). K- ϵ turbulence model parameter estimates using an approximate self-similar jet-in-crossflow solution, *In 8th AIAA theoretical fluid mechanics conference*, No. AIAA Paper No. 2017-4167.

- [124] **Nelly, S.M., Nieratschker, W., Nadler, M., Einsle, K., and Delgado, A.** (2018). Calibration of the reynolds stress model for the simulation of gas flows in corrugated tubes, *Heat Transfer Engineering*, 39(13-14), 1235–1242, doi : 10.1080/01457632.2017.1363632.
- [125] **Edeling, W.N., Schmelzer, M., Dwight, R.P., and Cinnella, P.** (2018). Bayesian predictions of reynolds-averaged navier-stokes uncertainties using maximum a posteriori estimates, *AIAA Journal*, 56(5), 2018–2029, doi : 10.2514/1.J056287.
- [126] **Klavaris, G., Xu, M., Hill, C., Menter, F.R., Patwardhan, S., and Verma, I.** (2024). Tuning of generalized k-omega turbulence model by using adjoint optimization and machine learning for gas turbine combustor applications, *ASME Journal of Engineering for Gas Turbines and Power*, 146(8), 81014, doi : 10.1115/1.4064367.
- [127] **Yildizeli, A. and Cadirci, S.** (2025). Calibration of the k- ω shear stress transport turbulence model for transitional boundary layer flows over flat plate and turbulent backward facing step flows, *Physics of Fluids*, 37(1), 015120, doi : 10.1063/5.0246991.
- [128] **Shih, T.H., Liou, W.W., Shabbir, A., Yang, Z., and Zhu, J.** (1995). A new k- ϵ eddy viscosity model for high reynolds number turbulent flows, *Computers & Fluids*, 24(3), 227–238, doi : 10.1016/0045-7930(94)00032-T.
- [129] **Thies, A.T. and Tam, C.K.W.** (1996). Computation of turbulent axisymmetric and nonaxisymmetric jet flows using the k-epsilon model, *AIAA Journal*, 34(2), 309–316, doi : 10.2514/3.13065.
- [130] **Raje, P. and Sinha, K.** (2021). Anisotropic sst turbulence model for shock-boundary layer interaction, *Computers & Fluids*, 228, 105072, doi : 10.1016/j.compfluid.2021.105072.
- [131] **Du, Y., Shu, B., Gao, Z., Nie, S., and Ma, R.** (2021). On the correction of k- ω sst turbulence model to three-dimensional shock separated flow, *In Proceedings of the 5th China Aeronautical Science and Technology Conference, Lecture Notes in Electrical Engineering*, Vol. 821.
- [132] **Moioli, M., Breitsamter, C., and Sørensen, K.A.** (2022). Turbulence modeling for leading-edge vortices: an enhancement based on experimental data, *AIAA Journal*, 60(3), 1382–1399, doi : 10.2514/1.J060062.
- [133] **Zhu, L., Sun, D., Guo, Q., and Yuan, X.** (2025). Structure parameter modification of stress limiter in k - ω shear stress turbulence model for high-speed flows with shock-wave/turbulent boundary layer interaction, *Physics of Fluids*, 37(1), 015212, doi : 10.1063/5.0250145.
- [134] **Banliat, M.Z., Merle, X., Dergham, G., and Cinnella, P.** (2020). Bayesian model-scenario averaged predictions of compressor cascade flows under uncertain turbulence models, *Computers & Fluids*, 201, 104473, doi : 10.1016/j.compfluid.2020.104473.

- [135] **Li, Z., Ju, Y., and Zhang, C.** (2023). Machine-learning data-driven modeling of laminar-turbulent transition in compressor cascade, *Physics of Fluids*, 35(8), 085133, doi : 10.1063/5.0164131.
- [136] **Tu, J., Yeoh, G.H., and Liu, C.** (2018). *Computational Fluid Dynamics: A Practical Approach*. Butterworth-Heinemann, Newton, MA, United States.
- [137] **Denton, J.D.** (1993). Loss mechanisms in turbomachines, *ASME Journal of Turbomachinery*, 115(4), 621, doi : 10.1115/1.2929299.
- [138] **Prasad, A.** (2005). Calculation of the mixed-out state in turbomachine flows, *ASME Journal of Turbomachinery*, 127(3), 564–572, doi : 10.1115/1.1928289.
- [139] **Leggett, J.** (2018). *Detailed investigation of loss prediction of an axial compressor cascade at off-design conditions in the presence of incident free-stream disturbances using large eddy simulations*, (Doctoral Dissertation), University of Southampton.
- [140] **Wei, W., Li, X., Ren, X., Gu, C., and Shi, P.** (2024). Power-loss methodology for a compressor cascade at various reynolds numbers and its validation, *Physics o Fluids*, 36(1), 016109, doi : 10.1063/5.0185156.
- [141] **Roberts, W.A.** (1975). The effect of reynolds number and laminar separation on axial cascade performance, *ASME Journal of Engineering for Gas Turbines and Power*, 97(2), 261–273, doi : 10.1115/1.3445978.
- [142] **Zaki, T.A., Wissink, J.G., Rodi, W., and Durbin, P.A.** (2010). Direct numerical simulations of transition in a compressor cascade: the influence of free-stream turbulence, *Journal of Fluid Mechanics*, 665, 57–98, doi : 10.1017/S0022112010003873.
- [143] **Menter, F., Kuntz, M., and Langtry, R.** (2003). Ten years of industrial experience with the sst turbulence model, *Heat and Mass Transfer*, 4, 625–6632.
- [144] **Fottner, L.** (1990). Test cases for computation of internal flows in aero engine components, (Technical Report), Propulsion and Energetics Panel Working Group 18, AGARD Advisory report No. 275.
- [145] **Dickens, M. and Read, A.** (2008). *The ERCOFTAC Knowledge Base Wiki: Application Area 6 - Turbomachinery Internal Flows, Application Challenge AC6-10*. Retrieved March 8, 2024 from:https://kbwiki.ercoftac.org/w/index.php/AC_6-10.
- [146] **Orszag, S.A., Yakhot, V., Flannery, W.S., and Boysan, H.F.** (1993). Renormalization group modeling and turbulence simulations, *In Near Wall Turbulent Flows*. Elsevier.
- [147] **Wilson, K.G.** (1993). The renormalization group and critical phenomena, *Reviews of Modern Physics*, 55, 583, doi : 10.1103/RevModPhys.55.583.

- [148] **Calvert, W.J.** (1991). Application of s1byl2 to the agard wg18 compressor test cases, (Technical Report), AGARD CP 510, CFD techniques for propulsion applications.
- [149] **Calvert, W.J.** (1982). An inviscid-viscous interaction treatment to predict the blade-to-blade performance of axial compressors with leading edge normal shock waves, *In Proceedings of the ASME 1982 International Gas Turbine Conference and Exhibit*, Vol. 1.
- [150] **ANSYS.** (2023). *Ansys® Academic Research, Release 23.1, Help System, Fluent Theory Guide*. ANSYS, Inc.
- [151] **Launder, B.E. and B., S.D.** (1972). *Lectures in Mathematical Models of Turbulence*. Academic Press.
- [152] **Wolfshtein, M.** (1969). The velocity and temperature distribution in one-dimensional flow with turbulence augmentation and pressure gradient, *International Journal of Heat and Mass Transfer*, 12(3), doi : 10.1016/0017-9310(69)90012-X.
- [153] **Kader, B.** (1981). Temperature and concentration profiles in fully turbulent boundary layers, *International Journal of Heat and Mass Transfer*, 24(9), doi : 10.1016/0017-9310(81)90220-9.
- [154] **Rhie, C.M. and Chow, W.L.** (1983). Numerical study of the turbulent flow past an airfoil with trailing edge separation, *AIAA Journal*, 21(11), doi : 10.2514/3.8284.
- [155] **Tyacke, J., Vadlamani, N.R., Trojak, W., Watson, R., Ma, Y., and G., T.P.** (2019). Turbomachinery simulation challenges and the future, *Progress in Aerospace Sciences*, 110, doi : 10.1016/j.paerosci.2019.100554.
- [156] **Thangam, S. and Speziale, C.G.** (1992). Turbulent flow past a backward-facing step - a critical evaluation of two-equation models, *AIAA Journal*, 30(5), doi : 10.2514/3.48951.
- [157] **Zaki, T., Wissink, J.G., Durbin, P.A., and Rodi, W.** (2009). Direct computations of boundary layers distorted by migrating wakes in a linear compressor cascade, *Flow, Turbulence and Combustion*, 83(3), doi : 10.1007/s10494-009-9216-0.
- [158] **Menter, F.R., Smirnov, P.E., Liu, T., and Avancha, R.** (2015). A one-equation local correlation-based transition model, *Flow, Turbulence and Combustion*, 95, doi : 10.1007/s10494-015-9622-4.
- [159] **Zhang, Z., Wu, Y., Guo, Z., and Dong, J.** (2024). Extraction of geometric features and analysis of flow mechanism of high loaded compressor airfoils at low reynolds number, *Physics of Fluids*, 36(3), doi : 10.1063/5.0195387.
- [160] **Bell, R.M. and Fottner, L.** (1995). Investigations of shock/boundary-layer interaction in a highly loaded compressor cascade, *In Proceedings of the*

- [161] **Leipold, R., Boese, M., and Fottner, L.** (2000). The influence of technical surface roughness caused by precision forging on the flow around a highly loaded compressor cascade, *ASME Journal of Turbomachinery*, 122(3), doi : 10.1115/1.1302286.
- [162] **Boese, M. and Fottner, L.** (2002). Effects of riblets on the loss behavior of a highly loaded compressor cascade, *In Proceedings of the ASME Turbo Expo 2002*, Vol. 5.
- [163] **Li, Z., Ju, Y., and Zhang, C.** (2019). Parallel large-eddy simulation of subsonic and transonic flows with transition in compressor cascade, *Journal of Propulsion and Power*, 35(6), doi : 10.2514/1.B37371.
- [164] **Baldwin, B.S. and Lomax, H.** (1978). Thin layer approximation and algebraic model for separated turbulent flows, *In 16th Aerospace Sciences Meeting*, No. AIAA Paper 78-257.
- [165] **Spalart, P.R. and Allmaras, S.R.** (1992). A one-equation turbulence model for aerodynamic flows, *In 30th Aerospace Sciences Meeting and Exhibit*, No. AIAA Paper 92-0439.
- [166] **Gatski, T.B., Hussaini, Y.M., and Lumley, J.L.** (1996). *Simulation and Modeling of Turbulent Flows*. Oxford University Press, Inc.
- [167] **Durbin, P.A. and Petterson Reif, B.A.** (2010). *Statistical Theory and Modeling for Turbulent Flows*. John Wiley & Sons, Ltd., (2.. ed.).
- [168] **Hanjalić, K. and Launder, B.E.** (2021). Reassessment of modeling turbulence via reynolds averaging: A review of second-moment transport strategy, *Physics of Fluids*, 33(9), 091302, doi : 10.1063/5.0065211.
- [169] **Hanjalić, K. and Launder, B.E.** (2023). *Modelling turbulence in engineering and the environment: Rational Alternative Routes to Closure*. Cambridge University Press, (2.. ed.).
- [170] **Slotnick, J., Khodadoust, A., Alonso, J., Darmofal, D., Gropp, W., Lurie, E., and Mavriplis, D.** (2014). Cfd vision 2030 study: A path to revolutionary computational aerosciences, (Technical Report NASA/CR-2014-218178), National Aeronautics and Space Administration.
- [171] **Jakirlic, S.** (2012). Extended excerpt related to the test case: Flow over a periodical arrangement of 2d hills, (Technical Report ACP8-GA-2009-233710-ATAAC), TU Darmstadt.
- [172] **Manceau, R.** (2003). Report on 10th ERCOFTAC (sig-15)/IAHR/qnet-cfd workshop on refined turbulence modelling, (Technical Report ERCOFTAC Bulletin, No. 57), European Research Community On Flow, Turbulence and Combustion.

- [173] **Langtry, R.B. and Menter, F.R.** (2009). Correlation-based transition modeling for unstructured parallelized computational fluid dynamics codes, *AIAA Journal*, 47, doi : 10.2514/1.42362.
- [174] **Li, Y., Xu, J., Qiao, L., Zhang, Y., and Bai, J.** (2021). A novel local-variable-based reynolds-averaged navier–stokes closure model for bypass and laminar separation induced transition, *Physics of Fluids*, 33 (10), 104103, doi : 10.1063/5.0066007.
- [175] **Menter, F.R., Matyushenko, A., Lechner, R., Stabnikov, A., and Garbaruk, A.** (2022). An algebraic lctm model for laminar–turbulent transition prediction, *Flow, Turbulence and Combustion*, 109, 841–869, doi : 10.1007/s10494-022-00336-8.
- [176] **Hu, Z., Zhang, P., Tang, F., Yuan, X., Chen, J., and Xiang, X.** (2025). Investigation and improvement of $k_t - k_l - \omega$ transition model for pressure gradient and inflow effects, *Physics of Fluids*, 37(1), 014101, doi : 10.1063/5.0243059.
- [177] **Okumus, B., Baran, O.U., and Bas, O.** (2026). A local correlation-based algebraic transition model in $k - \omega$ sst formulation, *Aerospace Science and Technology*, 168, 110786, doi : 10.1016/j.ast.2025.110786.
- [178] **Menter, F.R., Smirnov, P.E., Liu, T., and Avancha, R.** (1992). Influence of freestream values on $k-\omega$ turbulence model predictions, *AIAA Journal*, 30 (6).
- [179] **Spalart, P. and Rumsey, C.L.** (2007). Effective inflow conditions for turbulence models in aerodynamic calculations, *AIAA Journal*, 45(10), doi : 10.2514/1.29373.
- [180] **Davidson, L.** (2007). Using isotropic synthetic fluctuations as inlet boundary conditions for unsteady simulations, *Advances and Applications in Fluid Mechanics*, 1(1).
- [181] **Lardeau, S. and Leschziner, M.A.** (2011). The interaction of round synthetic jets with a turbulent boundary layer separating from a rounded ramp, *Journal of Fluid Mechanics*, 683(4).
- [182] **Bentaleb, Y., Lardeau, S., and Leschziner, M.A.** (2012). Large-eddy simulation of turbulent boundary-layer separation from a rounded step, *Journal of Turbulence*, 13(4).
- [183] **Vogel, E. and Pederson, C.** (2024). *NASA Langley Research Center: The Menter Shear Stress Transport Turbulence Model*. Retrieved August 2, 2025 from: <https://turbmodels.larc.nasa.gov/sst.html>.
- [184] **Johnson, D., Menter, F.R., and Rumsey, C.L.** (1994). The status of turbulence modeling for external aerodynamics, *In Fluid Dynamics Conference*.
- [185] **Madumjar, S.** (1988). Role of underrelaxation in momentum interpolation for calculation of flow with nonstaggered grids, *Numerical Heat Transfer*, 13.

- [186] **Carreño Ruiz, M. and D'Ambrosio, D.** (2022). Validation of the $\gamma - re_\theta$ transition model for airfoils operating in the very low reynolds number regime, *Flow, Turbulence and Combustion*, 109, 279–308, doi : 10.1007/s10494-022-00331-z.
- [187] **Ruck, B. and Makiola, B.** (1993). Flow separation over the step with inclined walls, *In Near Wall Turbulent Flows*, Vol. 999. Elsevier.
- [188] **Temmerman, L. and Leschziner, M.A.** (2001). Large eddy simulation of separated flow in a streamwise periodic channel constriction, *In Proceedings of International Symposium on Turbulence and Shear Flow Phenomena, Stockholm*.
- [189] **Rapp, C. and Manhart, M.** (2011). Flow over periodic hills: an experimental study, *Experiments in Fluids*, 51, 247–269.
- [190] **Lardeau, S.** (2021). *NASA Langley Research Center: 2-D Curved Backward-Facing Step*. Retrieved March 14, 2025 from: https://turbmodels.larc.nasa.gov/Other_LES_Data/curvedstep.html.
- [191] **Lardeau, S.** (2012). *The ERCOFTAC Knowledge Base Wiki - Underlying Flow Regime 3-31: Flow over curved backward-facing step*. Retrieved March 16, 2025 from: https://kbwiki.ercoftac.org/w/index.php/UFR_3-31.
- [192] **Tan, W., Zhang, H., Wang, L., Nie, S., Jiao, J., and Zuo, Y.** (2024). Effects of the uncertainty of wall distance on the simulation of turbulence/transition phenomena, *Aerospace*, 11(11), 898, doi : 10.3390/aerospace11110898.
- [193] **Temmerman, L., Bentaleb, Y., and Leschziner, M.A.** (2021). *NASA Langley Research Center: 2-D Periodic Hill*. Retrieved August 14, 2025 from: https://turbmodels.larc.nasa.gov/Other_LES_Data/2dhill_periodic.html.
- [194] **Song, S. and Eaton, J.K.** (2004). Reynolds number effects on a turbulent boundary layer with separation, reattachment, and recovery, *Experiments in Fluids*, 36(2), 246–258.
- [195] **Zhang, S. and Zhong, S.** (2010). An experimental investigation of turbulent flow separation control by an array of synthetic jets, *In 5th Flow Control Conference*, No. AIAA paper no. AIAA 2010-4582.
- [196] **McConnell, M.R., Knight, J., and Buick, J.M.** (2025). Boundary layer separation from a curved backward-facing step using improved delayed detached-eddy simulation, *Fluids*, 10(6), 145, doi : 10.3390/fluids10060145.
- [197] **Sarkar, D. and Savory, E.** (2021). Numerical modeling of freestream turbulence decay using different commercial computational fluid dynamics codes, *Journal of Fluids Engineering*, 143(4), 041503, doi : 10.1115/1.4049679.

- [198] **Lee, J. and Nguyen, V.T.** (2023). Effects of various backward-facing steps on flow characteristics and sediment behaviors, *Physics of Fluids*, 35(6), 065124, doi : 10.1063/5.0152852.
- [199] **Choi, S., Lee, J., Kim, Y., and Lee, S.** (2025). Machine learning-based prediction of turbulent flows over backward-facing steps with varying step angles using large eddy simulation data, *Physics of Fluids*, 37(6), 065172, doi : 10.1063/5.0269372.
- [200] **Temmerman, L., Leschziner, M.A., Mellen, C.P., and Fröhlich, J.** (2003). Investigation of wall-function approximations and subgrid-scale models in large eddy simulation of separated flow in a channel with streamwise periodic constrictions, *International Journal of Heat Fluid Flow*, 24, 157–180.
- [201] **von Kármán, T.** (1921). Über laminare und turbulente reibung, *Journal of Applied Mathematics and Mechanics*, 1, 233–252, doi : 10.1002/zamm.19210010401.
- [202] **Sandborn, V.A. and Kline, S.J.** (1961). Flow models in boundary-layer stall inception, *Journal of Basic Engineering*, 83(3), 317–327, doi : 10.1115/1.3658953.



CURRICULUM VITAE

Name SURNAME: Cem TURUTOĞLU

EDUCATION:

- **B.Sc.:** 2015, Sakarya University, Faculty of Mechanical Engineering, Department of Mechanical Engineering
- **M.Sc.:** 2018, Istanbul Technical University, Heat and Fluid Programme, Department of Mechanical Engineering

PROFESSIONAL EXPERIENCE AND REWARDS:

- 2016-2017 Cvsair Ventilation Systems
- 2017-2018 SimuTek Engineering Technologies
- 2018-2022 Baykar Technologies
- 2022-Present: Delta V Space Technologies
- 2023 ASME FMTC Best Paper Award for the paper presented at IMECE2023

PUBLICATIONS, PRESENTATIONS AND PATENTS ON THE THESIS:

- **Turutoglu, C.** and Cadirci, S. (2023). Improvement of Standard K-Epsilon Turbulence Model for Round Free Jets by Adjusting Closure Coefficients. *Proceedings of the ASME 2023 International Mechanical Engineering Congress and Exposition. Volume 9: Fluids Engineering*, October 29-November 2, 2023 New Orleans, Louisiana, USA, doi : 10.1115/IMECE2023-115131.
- **Turutoglu, C.,** Cadirci, S., Yilmaz, S. and Erdem, D. (2024). Calibration of the Reynolds stress model for turbulent round free jets based on jet half-width, *Physics of Fluids*, 36(11), 115133, doi : 10.1063/5.0238900.
- **Turutoglu, C.** and Cadirci, S. (2025). A new variant of renormalization group turbulence model for flows in axial compressor cascade, *Physics of Fluids*, 37(7), 075123, doi : 10.1063/5.0274264.
- **Turutoglu, C.,** Yildizeli, A., Cadirci, S., Kuzuguden, C. and Erdem, D. (2026). Enhancing the $k-\omega$ shear stress transport turbulence model through renormalization group theory for flows separated from a continuous surface, *Physics of Fluids*, 38(1), 015165, doi : 10.1063/5.0307492.

OTHER PUBLICATIONS, PRESENTATIONS AND PATENTS:

- **Turutoglu, C.** and Cadirci, S. (2025). Investigation of Airflow in Human Nasal Cavity and Virtual Septoplasty Using CFD. *Proceedings of the 12th International Conference on Heat Transfer and Fluid Flow (HTFF 2025)*, August, 2025 Paris, France, doi : 10.11159/htff25.169.