Numerical study of trapped vortex combustors characteristics in small ramjets

Chen, Song

2016

Chen, S. (2016). Numerical study of trapped vortex combustors characteristics in small ramjets. Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/67062>

<https://doi.org/10.32657/10356/67062>

NANYANG
TECHNOLOGICAL
UNIVERSITY

**NUMERICAL STUDY OF TRAPPED VORTEX
COMBUSTORS CHARACTERISTICS
IN SMALL RAMJETS**

SONG CHEN

SCHOOL OF MECHANICAL AND AEROSPACE ENGINEERING

2016

**NUMERICAL STUDY OF TRAPPED VORTEX
COMBUSTORS CHARACTERISTICS
IN SMALL RAMJETS**

SONG CHEN

School of Mechanical and Aerospace Engineering

A thesis submitted to the Nanyang Technological University
in partial fulfillment of the requirements for the degree of
Doctor of Philosophy

2016

Acknowledgements

I am fortunate to receive guidance from various supervisors during my Ph.D. study at NTU. I want to express my sincere gratitude to Dr. Randy Chue, my former supervisor, not only for offering the opportunity to learn and work under him, but also for his continuous support and invaluable guidance throughout the research even after his retirement. He has been a source of inspiration of this work, and his insights, overwhelming enthusiasm in research and education as well as warm personality has won my highest respect. I also would like to express my sincere thanks to Dr. Jörg Schlüter, another former supervisor, who took me in his group without any hesitation after Dr. Randy Chue retired. I am very grateful to Dr. Dan Zhao, my current supervisor, who continuously provides me valuable suggestions and discussions during these years in many ways. I am also grateful to Dr. Simon Ching-Man Yu of Singapore Institute of Technology, my co-supervisor, for continuously providing me encouragement, guidance, discussions and caring in both research and personal life. Without their guidance, the research would not have been so enriching and fulfilling.

Thanks are due to the thesis examiners, Dr. Teng Yong Ng, Dr. Jian Jun Shu and Dr. Daniel New, for their time, comments, discussion, and valuable suggestions.

Special thanks are due to Dr. Tue Nguyen, who has worked closely with me during his stay at NTU. I have gained much from his enthusiasm, valuable discussions, and friendship.

I would also like to thank Mr. Youxiang Chew, Dr. Junting Xiang, Dr. John Chung-Hung Wang, and all the members in Dr. Zhao's group for friendship and moral support.

Finally, I want to express my deepest gratitude to my wife and my parents, who have always been with me for every tough moment to give me encouragement, understanding and support.

Abstract

Small-scale ramjets are promising air-breathing engines for high-speed propulsion in military, due to their high specific impulse, low aerodynamic drag and good compatibility with operating systems. The challenges of designing a miniature ramjet combustor mainly stem from insufficient space and short flow residence time for good mixing and burning, and difficulties in flame stabilization. Impossible utilization of relatively large air-blast fuel injectors in a small combustor provides more difficulties for the design. The trapped vortex combustor (TVC) concept, locking a pilot flame inside a cavity instead of exposing it to the main stream, provides a simple but effective solution for flame stabilization. In this thesis, a TVC designed for a small ramjet operating at Mach 4.0 is numerically investigated by using ANSYS FLUENT v14.5 and a large eddy simulation (LES) code from Stanford. The results show that the optimal cavity size that traps a vortex inside can be determined by calculating the drag coefficient. Because the designed small combustor does not have bypass air directed into the cavity, injecting all the fuel directly in the optimal cavity always leads to an overly fuel-rich condition. However, when the fuel is injected in the upstream with a proper fuel/air momentum flux ratio, part of fuel goes into the cavity and the rest directly flows over it. Hence an approximately stoichiometric cavity and a hot cavity flame can be obtained.

Under some circumstances, the TVC may operate in a high-spinning motion. For example when it is installed in a spin-stabilized ramjet projectile, in which the spinning rate can be as high as 30,000 rpm. Numerical methods were used to investigate the effects of spinning motion on the TVC. The results show that the Coriolis effects dominate the flow in the cavity when it is subject to a high spinning motion. Multiple secondary vortices and strong three-dimensional flows are observed in the cavity, so that the mixing is improved and a hotter cavity flame is generated. The centrifugal force effects, however, generate a short recirculation zone in the main combustor and concentrate the fuel stream on the combustor axis, which leads to deteriorate fuel-air mixing.

Contents

Chapter 1 Introduction	1
1.1 Background	1
1.2 Literature Review	2
1.2.1 The Ramjet	2
1.2.2 Flame Stabilization and Flame Holders in Ramjets	5
1.2.3 Trapped Vortex Combustor	9
1.3 Objectives and Outlines of the Present Work	18
Chapter 2 Numerical Modeling	20
2.1 Governing Equations of Gaseous Combustion	20
2.2 Turbulence Modelling	21
2.2.1 RANS Turbulence Models	21
2.2.2 LES Turbulence Model	28
2.3 Combustion Modelling	29
2.3.1 Reaction Rate Method	29
2.3.2 Primitive Variable Method	32
2.4 Numerical Solver and Boundary Conditions	35
2.4.1 RANS - FLUENT v14.5	36
2.4.2 LES - JETCODE	37
2.5 Verification and Validation	39
2.5.1 Flow Over Backward Facing Step	39
2.5.2 Locked Vortex Base Flow	43
2.5.3 Sandia H ₂ Diffusion Jet Flame	45
2.5.4 Hsu's TVC Combustor	50
2.5.5 Prediction of the Lifted Flame	58
2.5.6 Rotating or Swirling Flows	60
2.6 Summary	65

Chapter 3 Flow Field Characteristics of the TVC in a Mini-Ramjet	66
3.1 Inflow Conditions for the Combustor	66
3.1.1 Determination of the Flight Altitude	67
3.1.2 Stream Thrust Analysis	69
3.2 Classification of Flow Regimes in TVC	76
3.3 Grid Sensitivity Analysis	78
3.4 Flow Field Characteristics.....	80
3.5 Full Three-Dimensional Unsteady Results.....	81
3.6 Summary	83
Chapter 4 Gaseous Fuel Injection and Mixing	85
4.1 3-D Grid and Grid Sensitivity Study	86
4.2 Fuel injection Schemes Investigations	87
4.2.1 Injecting Locations	88
4.2.2 Upstream Injection: Effects of Fuel/Air Momentum Flux Ratio	93
4.2.3 Effects of Reducing the Fuel Mass Flow Rate in Case 3	98
4.3 Summary of the Mixing Schemes	100
Chapter 5 Combustion Performance	102
5.1 Definition of Axial Combustion Efficiency	102
5.1.1 Species-Based Efficiency	102
5.1.2 Temperature-Rise Based Efficiency.....	103
5.1.3 Energy-Based Efficiency.....	103
5.2 Effects of Injecting Locations and Momentum Flux Ratios	104
5.3 Effects of Reduced Fuel Mass Flow Rate in the Cavity.....	107
5.4 Angled Injection and Upstream Injection Locations.....	109
5.5 Effect of Combustor Length.....	111
5.6 Summary	112
Chapter 6 Effect of Spinning Motion.....	114
6.1 Introduction	114
6.2 Literature Review	114

6.3	Numerical Approach	117
6.3.1	Flow Solvers and Numerical Models	117
6.3.2	Computational Domain and Boundary Conditions	117
6.3.3	Discussion about the Combustor Inflow Conditions in Spinning Motion ...	118
6.3.4	Treatments of Numerical Boundary Conditions	120
6.4	Results and Discussion.....	121
6.4.1	Cold Flow Field Characteristics Predicted by LES.....	121
6.4.2	RANS Results	124
6.5	Summary	134
Chapter 7 Conclusions and Future Work		135
7.1	Conclusions.....	135
7.2	Suggestions for Future Work	136

List of Figures

Fig. 1-1: A schematic of a typical ramjet engine [7].	2
Fig. 1-2: The pre-prototype Ramgen engine [2, 3].	4
Fig. 1-3: Ramgen engine flow details [2, 3].	4
Fig. 1-4: Schematic of (a) a V-gutter flame holder; (b) a bluff body flame holder; (c) a sudden expansion flame holder [14].	5
Fig. 1-5: Schematic of flame stabilization boundaries [15].	6
Fig. 1-6: Schematic of barchan dune (BD) vortex flame holder: (a) side view and (b) top view [13].	7
Fig. 1-7: Schlieren images and schematics of the flow structure of a cavity exposed to a supersonic flow: (a) a rectangular cavity, and (b) a cavity with inclined back wall [17].	8
Fig. 1-8: Schematic of fluidic flame holders using: (a) an opposing jet; (b) a single transverse jet; (c) radially transverse jets [15].	9
Fig. 1-9: (a) Conventional swirl-stabilized combustor, (b) trapped vortex combustor [18].	10
Fig. 1-10: First generation TVC [5].	11
Fig. 1-11: Second generation TVC [23].	13
Fig. 1-12: Third generation TVC and illustration of its cavity [22, 23].	14
Fig. 1-13: Three optimum injecting schemes for the third generation TVC [22].	14
Fig. 1-14: GE 2-passage and 3-passage TVC test rig configurations [32].	15
Fig. 1-15: High pressure TVC test rig data [18].	15
Fig. 1-16: Schematic of the AVC [33, 34].	16
Fig. 1-17: Combustion efficiency for bluff body and TVC [33].	16
Fig. 1-18: Schematic of the tested ramjet combustor with a cavity flame holder [38].	18
Fig. 2-1: Time scales in chemically reacting flows [52].	35
Fig. 2-2: Staggered grid at the wall boundary [53].	38
Fig. 2-3: (a) Typical flow features of the backward facing step flow, (b) turbulent boundary layer upstream at $x = -10$ mm. U_{max} is the maximum velocity, and H is the step height.	40
Fig. 2-4: (a) Pressure coefficient and (b) skin friction coefficient distributions downstream of the backward facing step. X_r is the reattachment length.	40
Fig. 2-5: (a) General flow features of a turbulent base flow, (b) base pressure coefficient as a function of Mach number. D is the diameter of the base.	41
Fig. 2-6: Static pressure, total pressure and Mach number distributions along the centerline of the wake region. Each plot is computed at different upstream Mach number Ma .	41
Fig. 2-7: Turbulent base flow with after-bodies. L is the length of the after-body and D is the diameter of the main body.	42

Fig. 2-8: Wall and centerline pressure coefficient distribution of turbulent base flow with after-bodies.	42
Fig. 2-9: Schematic of a locked cavity after-base [20].	43
Fig. 2-10: Flow features of (a) $D_I/D_0=0.75$ and $H/D_I = 0.4$, (b) $D_I/D_0=0.75$ and $H/D_I = 0.55$, (c) $D_I/D_0=0.75$ and $H/D_I = 0.8$, and (d) drag reduction predictions and comparisons.	44
Fig. 2-11: Comparisons of locations of vortex cores at (a) $D_I/D_0=0.75$ and $H/D_I = 0.35$, (b) $D_I/D_0=0.75$ and $H/D_I = 0.6$. Upper half: from experiment [20], lower half: from CFD.	45
Fig. 2-12: Temperature contours of the H_2 jet flame predicted by the ED model.	46
Fig. 2-13: Axial velocity (upper) and temperature (lower) comparisons along the axis, at $x/L = 0.5$, and at $x/L = 1.0$. L is the experimentally observed flame length.	47
Fig. 2-14: Axial velocity (upper) and temperature (lower) comparisons along the axis, at $x/L = 0.5$, and at $x/L = 1.0$. L is the experimentally observed flame length.	48
Fig. 2-15: Axial velocity (upper) and temperature (lower) comparisons along the axis, at $x/L = 0.5$, and at $x/L = 1.0$. L is the experimentally observed flame length.	50
Fig. 2-16: (a) Schematic of the Hsu's TVC, (b) experimental temperature profiles [5].	51
Fig. 2-17: Temperature comparisons with different combustion models.	52
Fig. 2-18: Temperature comparisons with different turbulence models.	54
Fig. 2-19: Temperature distributions with different turbulent Schmidt numbers.	56
Fig. 2-20: Temperature predictions comparison: FLUENT v14.5 and CONVERGE v2.2. ..	57
Fig. 2-21: The comparisons of our numerical results to the published numerical data.	58
Fig. 2-22: Diffusion flame stabilized by a triple flame [73].	59
Fig. 2-23: Lift-off distance predictions against the experimental data from [74], the lift-off distance measured in the experiment is marked as the red dashed line.	59
Fig. 2-24: Lift-off distances predicted by EDC model at: (a) 300 K and (b) 900 K ambient air.	60
Fig. 2-25: Validation results with RSM at swirl number $S = 0$	63
Fig. 2-26: Validation results with RSM at swirl number $S = 0.6$: (a) axial velocity, (b) tangential velocity.	63
Fig. 2-27: Validation results of LES for a confined jet without and with swirl.	64
Fig. 3-1: Schematic of the proposed mini-ramjet engine.	67
Fig. 3-2: A typical schematic of the air-breathing flight corridor [77].	67
Fig. 3-3: Zero-lift drag coefficient [78].	68
Fig. 3-4: Lift/drag ratios for a circular cross section body and a lifting body flying at a small angle of attack [78].	68
Fig. 3-5: Schematic of the air-breathing engine temperature-entropy diagram [77].	70
Fig. 3-6: Schematic of the designed combustor.	76

Fig. 3-7: Four typical cavity flow regimes. a) Wake back flow, b) unsteady cavity vortex, c) steady cavity vortex, and d) compressed cavity vortex. Left column: taken from Ref. [70]. Right column: the current simulations.	77
Fig. 3-8: Schematic of the two dimensional grid.	78
Fig. 3-9: Static pressure comparison between three different 2-D axisymmetric grids: (a) along the combustor axis; (b) along the combustor casing.	79
Fig. 3-10: Static pressure comparison between 2-D axisymmetric and 3-D periodic simulations: (a) along the combustor axis; (b) along the combustor casing.	79
Fig. 3-11: (a) Drag reduction for different cavity sizes; (b) streamlines with optimum cavity size.....	81
Fig. 3-12: Flow vectors of unsteady RANS results.....	81
Fig. 3-13: Instantaneous vorticity contours from LES results.....	83
Fig. 4-1: Static pressure comparison between three different 3-D axisymmetric grids: (a) along the combustor axis; (b) along the combustor casing.	86
Fig. 4-2: Grid topology for the round fuel injector: (a) four corners approach and (b) center square approach.....	87
Fig. 4-3: Schematic of the 3-D periodic grid for the combustor and the grid for the fuel injector.....	87
Fig. 4-4: Streamlines and equivalence ratio contours in the cavity for different injecting locations: (a) case 1, (b) case 2, (c) case 3, and (d) case 4. The arrow indicates the fuel injection location and direction.	89
Fig. 4-5: Equivalence ratio contours in the main combustor: (a) case 1, (b) case 2, (c) case 3, and (d) case 4.	91
Fig. 4-6: Flow structures of a jet in crossflow.....	92
Fig. 4-7: Distributions of turbulent kinetic energy: (a) case 3, and (b) case 4.	92
Fig. 4-8: Equivalence ratio contours for different fuel-air momentum flux ratios: (a) case 5, (b) case 6, (c) case 7, and (d) case 8.....	95
Fig. 4-9: Equivalence ratio contours of case 5 at axial planes for $q = 0.52$	96
Fig. 4-10: Equivalence ratio contours of case 6 at axial planes for $q = 0.63$	97
Fig. 4-11: Equivalence ratio contours of case 7 at axial planes for $q = 1.09$	97
Fig. 4-12: Mixing performance in the main combustor: (a) case 5, (b) case 6, (c) case 7, and (d) case 8.	98
Fig. 4-13: Streamlines and equivalence ratio contours of: (a) case 9, (b) case 10, (c) case 11, and (d) case 12.....	99
Fig. 5-1: Temperature contours in the combustor: (a) case 3, (b) case 6, and (c) case 7.	105
Fig. 5-2: Distributions of turbulent kinetic energy: (a) case 3, and (b) case 7.	106

Fig. 5-3: Combustion efficiency.	106
Fig. 5-4: Temperature contours of the reduced cavity fuel schemes: (a) case 10, (b) case 11, and (c) case 12.....	108
Fig. 5-5: Combustion efficiency of injection schemes with (a) different injection angles, (b) different upstream injection locations.....	110
Fig. 5-6: Combustion efficiencies with different combustor lengths.....	112
Fig. 5-7: Streamlines for combustor with different length: (a) $z/D = 2.5$, (b) $z/D = 3.0$, and (c) $z/D = 3.5$	112
Fig. 6-1: The secondary flow in a teacup stirred to spin [94].	115
Fig. 6-2: The secondary flows in gas turbine blades [95].	116
Fig. 6-3: Grid topology of the $1/8^{\text{th}}$ segment combustor.	118
Fig. 6-4: Schematic of the struts in the design ramjet.	118
Fig. 6-5: Specified inlet profiles in FLUENT: (a) velocity profiles, and (b) pressure profiles.	120
Fig. 6-6: Vorticity contours of the combustor: (a) 0 rpm, (b) 30,000rpm.....	121
Fig. 6-7: Two-dimensional streamlines for the combustor: (a) 0 rpm, (b) 30,000rpm.	123
Fig. 6-8: Three-dimensional streamlines of the combustor viewed in the relative reference frame:	124
Fig. 6-9: 2-D streamlines with pressure contours in cold flows: (a) 0 rpm, and (b) 30,000 rpm.	125
Fig. 6-10: Two-dimensional streamlines in combusting flows: (a) 0 rpm, and (b) 30,000 rpm.	126
Fig. 6-11: 3-D streamlines in combusting flows: (a) 0 rpm, and (b) 30,000 rpm.	126
Fig. 6-12: Single 3-D streamline in combusting flow for the spinning TVC: (a) bird's eye view of the selected streamline, and (b) section view of the selected streamline (viewed in the relative reference frame).	127
Fig. 6-13: Full three dimensional fuel stream trajectories viewed in the relative reference frame:	131
Fig. 6-14: Turbulent kinetic energy in the combusting flows: (a) 0 rpm and (b) 30,000 rpm.	131
Fig. 6-15: Temperature contours of the combustor: (a) non-spinning and (b) spinning.	133
Fig. 6-16: Combustion efficiency for the non-spinning and spinning combustor.....	134

List of Tables

Table 2-1: Model constants in $k - \varepsilon$ standard model.....	23
Table 2-2: Model constants in $k - \varepsilon$ RNG model.....	24
Table 2-3: Model constants in $k - \varepsilon$ realizable model.....	24
Table 2-4: Model constants of $k - \omega$ standard model.....	25
Table 2-5: Model constants of $k - \omega$ SST model.....	27
Table 2-6: Selections of the flow solver and numerical models for TVC.....	65
Table 3-1: Standard atmosphere at 22.5 km.....	69
Table 3-2: Operating conditions of the designed combustor.....	75
Table 3-3: Combustor operating conditions.....	84
Table 4-1: Injection conditions at different locations.	88
Table 4-2: Upstream fuel injection with different momentum flux ratios.	94
Table 4-3: Bottom injection with reduced mass flow rate.	99
Table 5-1: Three injection schemes from mixing results for combustion simulations.	104
Table 5-2: Reduced cavity injection schemes for the combustion simulations.....	107
Table 6-1: Numerical flow solvers and models used in the spinning combustor.....	117

NOMENCLATURE

Roman Symbols

a	speed of sound, $a = \sqrt{\gamma RT}$
A	cross section area; pre-exponential factor in Arrhenius form
c	convection velocity
c_p	specific heat
C	molar concentration
C_d	drag coefficient
C_ω	cross diffusion term in $k - \omega$ Shear Stress Transport (SST) model
d, D	diameter
Da	Damkhöler number, $Da = \text{flowtime scale} / \text{chemical time scale}$
$D_{m,k}, D_{T,k}$	mass and thermal mass diffusion coefficient of the k^{th} species
E	activation energy
f	body force; mixture fraction
F	thrust
G_b	generation of turbulence kinetic energy due to buoyancy
G_k	generation of turbulence kinetic energy, $G_k = \mu_t S^2$
G_ω	production of turbulent specific rate
h	specific enthalpy of mixture(sensible + chemical) per unit of mass
h_k	specific enthalpy of the k^{th} species per unit mass
$\Delta h_{f,k}^\circ$	standard formation enthalpy of the k^{th} species per unit mass
H	cavity length
J_k	diffusion flux of the k^{th} species
k	species number index; turbulent kinetic energy; reaction rate constant
L	flame length

N	number of species
M_k	molecular weight of the k^{th} species
Ma	Mach number
$\dot{m}$	mass flow rate
p	static pressure
P	mole rate of production
$\dot{Q}$	heat source term
r	radius
R	gas constant, 8.3145 J/(mol·K)
s	stoichiometric oxidant/fuel ration by mass
S	modulus of the mean rate-of-strain tensor, $S = \sqrt{2S_{ij}S_{ij}}$; swirl number
Sc	Schmidt number, $Sc = \text{viscous diffusion}/\text{mass diffusion}$
t	time
T	static temperature
u	velocity
ν', ν''	stoichiometric coefficients for reactants and products
$\dot{\omega}_k$	source term of the k^{th} species associated with chemical reaction
$x, y, z(r, \theta, z)$	Cartesian coordinates (cylindrical coordinates)
Y_k	mass fraction of the k^{th} species
Y_M	turbulence term due to compressible effects, $Y_M = 2\rho\varepsilon\frac{k}{a^2}$

Greek Symbols

ρ	density
τ_{ij}	viscous stress
λ	thermal conductivity
η	rate exponent
σ	turbulent Prandtl number

δ_{ij}	Kronecker delta function
μ	laminar dynamic viscosity
μ_t	turbulent dynamic viscosity
ϕ	equivalence ratio
ε	turbulence dissipation rate
ω	turbulence specific dissipation rate
$\dot{\omega}$	species source term
ξ^*	length scale fraction in Eddy Dissipation Concept (EDC) model
χ	reaction fraction in Eddy Dissipation Concept (EDC) model
ν	kinematic viscosity
$\overline{\Omega}_{ij}$	mean rate-of-rotation tensor
Ω	angular velocity
Ψ	general variable

Chapter 1

Introduction

1.1 Background

A ramjet, the simplest form of the air-breathing engine, has been receiving much attention as an effective supersonic propulsion system in military applications or as a promising alternative for stationary power generation systems in industry over the past couple of decades [1-3]. The development of ramjet engines has trended towards smaller size, because of compactness, low aerodynamic drag as well as compatibility with operational systems such as unmanned aerial vehicles (UAVs) and missile launching platforms. The small size, however, also implies that it would not have the luxury of implementing complex design features for mixing and combustion in relatively large gas turbine combustors. Hence, the challenges in designing a small ramjet combustor mainly stem from difficulties in achieving effective fuel-air mixing, combustion and flame stabilization within a small confined space and a short flow residence time.

A combustor concept, trapped vortex combustor (TVC), was firstly proposed for gas turbines by Hsu in 1995 [4, 5]. It provides a novel and stable way for mixing and flame holding, as the pilot flame is anchored inside a carefully designed cavity, instead of exposing in the mainstream like that in normal bluff body or V-gutter flame holders. Because of the shielding of the cavity, the anchored flame is not very sensitive to the main flow speed. Therefore, the cavity pilot flame is expected to be still robust and stable, even when the flow speed in a ramjet combustor is high.

The intended interest of this thesis lies in the small-scale ramjets using the TVC concept. The feasibility of using the concept for ramjets is for potential applications to supersonic propulsion, for example, UAVs, missiles and ramjet-assisted projectiles.

1.2 Literature Review

“If I have seen further, it is by standing on the shoulders of giants”. Just as what Isaac Newton said, further advances in technology require a thoughtful study of the past.

1.2.1 The Ramjet

The ramjet, which was first invented by Rene Lorin of France in 1913 [6], has been through a development of a century until now. The ramjet gets its name from the way of air compression, which is realized solely by the forward motion of the engine. Nowadays the ramjet more commonly refers to the engine that slows the incoming air to a subsonic velocity in the combustion chamber, to distinguish from a scramjet (supersonic combustion ramjet) in which combustion occurs in supersonic flows. A schematic of a typical ramjet is shown in Fig. 1-1 below.

Fig. 1-1: A schematic of a typical ramjet engine [7].

As the simplest air-breathing engine, the ramjet has no moving or rotating parts such as compressor blades and turbines in gas turbine engines. It only consists of a ram-type inlet, a combustion chamber and a convergent-divergent exhaust nozzle. Incoming air enters the inlet where it is compressed before mixing with the fuel and burning in the combustor. Then, the hot mixtures after combustion are ejected out through the nozzle. During these processes from the inlet to the nozzle, the momentum change of the through flow generates a force to propel the engine forward. If the static pressure difference between the inlet and the exit of nozzle is considered, the net thrust of the engine can be expressed as follows:

$$F = \dot{m}_o u_o - \dot{m}_i u_i + p_o A_o - p_i A_i \quad (1.1)$$

where the subscripts o and i stand for the outlet and the inlet.

One major disadvantage of the ramjet is that it cannot provide thrust to propel a vehicle from a standing start. Even in subsonic flights, the air compression efficiency is still low. Hence, ramjets are only suitable for supersonic flights in which the efficiency of the inlets is increased. Assistant devices i.e. gas turbines or rockets are needed to boost the ramjet to supersonic speeds. In the supersonic regime, a system of shock waves is created by the rams of the inlet, and is used to compress the incoming air.

Compared to a rocket, another widely used engine for missiles, the ramjet has almost the same simplicity in construction, but a rather longer range for the same propellant weight, because it does not need to carry the oxidizer as the rocket does. It uses the air in the atmosphere as the oxidizer so that a larger specific impulse than the rocket can be obtained. Hence, the ramjet is quite attractive for military propulsion and there are many missiles propelled by ramjets, such as the Bloodhound and the Sea Dart (U. K.), the Talos, the Bomarc interceptor missile and the Navaho supersonic cruise missile (U.S.), and the Burya missile (Russia) [1]. In the application of airplanes, one example using the ramjet is the Lockheed SR-71 Blackbird, a supersonic strategic reconnaissance aircraft. The propulsion device of the Blackbird is a combination of the ramjet and the gas turbine (it is also called the turbo-ramjet). During the takeoff process or flight at low speeds, the engine works as a turbojet, while in high enough supersonic speed, it switches to the ramjet mode.

In the civil power generation industry, the ramjet is also a promising engine with potential to improve the power and to reduce the design complexity over conventional gas turbines. Ramgen Power Systems proposed the first rotary ramjet for ground power generation which is shown in Fig. 1-2 [2, 3]. The main concept of the Ramgen engine is to place one or more ramjets onto the rim of a rotor so that the thrust is generated tangentially, pushing the ramjet on the rim of the rotor to spin at supersonic speeds. Operation of the Ramgen engine requires the rim of the rotor spinning at supersonic speed. A starter motor is used to start up the rotor from rest to working conditions. In this way, the thrust from the ramjets is converted into shaft torque for power generations. As shown in Fig. 1-2, the air enters into the engine casing and mixes with fuel. Then, the fuel-air mixture enters the rotating ramjets where combustion takes place. The flow details in the moving ramjet are illustrated in Fig. 1-3. From the stations illustrated in Fig. 1-3, it can be seen that the working principle of the rotating ramjet is exactly the same with the typical ramjet as shown in Fig. 1-1. The thrust generated by the ramjets keeps the rotor spinning so that the shaft torque is exported continuously. A similar configuration named Rim-Rotor Rotary Ramjet Engine (R4E) was also proposed and built by Rancout et al. [8, 9]. In comparison with gas turbine power generations, rotary ramjets have small number of parts because of the absence of

turbine blades. The small number of parts directly means a reduction in design and manufacturing costs, which attracts a lot of attention from the industry.

Fig. 1-2: The pre-prototype Ramgen engine [2, 3].

Fig. 1-3: Ramgen engine flow details [2, 3].

1.2.2 Flame Stabilization and Flame Holders in Ramjets

In practical combustors, an igniter is usually used to initiate flames. After ignition, if the combustion process can be continuously sustained by the hot products from the reactions, flame stabilization is achieved. Different from the concept of combustion instability, which usually deals with pressure fluctuation problems, flame stabilization refers to the flame stationed at a certain location. However, the flame is not intrinsic stationary and it has different propagation speeds in laminar or turbulent flows. Therefore, a stabilized flame in fact is the result of a dynamic balance between flame propagation and fluid motion. In premixed mixtures with a small flow speed, the flame can propagate upstream if the flame speed is larger than the flow speed, which is called flashback. On the contrary, in a non-premixed flame, if the flow speed is too high, the flame may extinguish, i.e. blowout.

The function of flame holders is to create recirculation zones with low flow speeds so that combustion can occur and the products of chemical reaction can act as an ignition source for continuous combustion. The recirculation zones are widely used in practical combustors to avoid the flame blowout in high-speed flows. In general, flame holders can be classified as geometric flame holders and fluidic flame holders. The former employs mechanical structures with certain configuration such as bluff bodies and steps, whereas the latter uses aerodynamic means, usually in the form of jets, to generate recirculation zones.

1.2.2.1 Geometric Flame Holders

Different from traditional gas turbine combustors where a swirl stabilizer is used to generate circulation zones by utilizing swirling flows, flame holders in ramjets usually take the forms of bluff bodies or sudden expansions (see Fig. 1-4). Recirculation zones with relatively low flow speeds downstream of these geometric structures are responsible for fuel-air mixing and flame stabilization. These flame holders have been extensively studied over the last half century, and most of the researches had been focused on their stability performance [10-13].

Fig. 1-4: Schematic of (a) a V-gutter flame holder; (b) a bluff body flame holder; (c) a sudden expansion flame holder [14].

Flames are prone to be blown out in high speed flows. The relation of the blowout velocity and operational conditions is usually used to describe the stability performance of a flame holder, as illustrated in Fig. 1-5. It can be seen that flames can only be obtained within a certain range of flow speeds and equivalence ratios. The velocity at which the combustion ceases is called blowout velocity. It varies with the equivalence ratio, which is a ratio of the actual fuel-air ratio to the stoichiometric fuel-air ratio. When it has a value of 1.0, that means the mixture is stoichiometric, or both the fuel and oxidizer are completely consumed after combustion. If it is above 1.0, it means there is excess fuel (fuel rich) or the fuel cannot be consumed completely. If it is below 1.0, it means there is excess air (fuel lean) or the oxidizer cannot be consumed completely after combustion. The blowout velocity as shown in Fig. 1-5 defines the boundary of continuous combustion. When the velocity exceeds the blowout velocity, flame stabilization cannot be obtained.

Fig. 1-5: Schematic of flame stabilization boundaries [15].

Zukoski et al. [10] studied the flame stabilization performance of bluff bodies at low Reynolds numbers (in the order of $10^2 \sim 10^3$) and intermediate Reynolds numbers (in the order of 10^4), based on critical Reynolds numbers of the order of 10^4 . They identified that the flow transition from laminar to turbulent changes the blowout velocity significantly. Longwell et al. [11] investigated the flame stabilization performance of baffles in high-speed gas streams. The effects of the flow velocity, temperature, pressure, baffle diameters and shapes were studied. Their experiments found that pressure has little effect on the stability range, and a higher inlet air temperature helps to widen the limits of burning. Considering that the ramjet combustor has a relatively high inlet temperature (around 700 ~ 900 K), a wider burning range is expected. They also pointed out that, as small baffles generate small

recirculation zones and thus have a narrow operating range, they become inoperable at high subsonic velocity. In other words, large bluff bodies are more effective in flame stabilization at high-speed flows for ramjet combustors. However, larger bluff bodies have a larger blockage ratio, which means they provide more drag and total pressure loss than the small flame holders do. Penner and Williams [12] summarized the general effects of various parameters on the flame stability including the configuration of bluff bodies, temperature, pressure and the mixture composition. In 1980s, Gao and Ning [13] proposed a flame holder known as barchan dune (BD) vortex flame holder as shown in Fig. 1-6. The idea came from an inspiration of barchan dune in the desert which has a stable configuration even under strong winds. It essentially uses the aerodynamic characteristics of streamlined bodies. In comparison with the V-gutter flame holder, the BD-type flame holder can provide a wider flame stability limit and much smaller drag. However, the main disadvantage of BD-type flame holder comes from the difficulties of maintaining the complex curve configurations especially in the hot temperature environments, as the flame holder performance is highly dependent on the geometry.

Fig. 1-6: Schematic of barchan dune (BD) vortex flame holder: (a) side view and (b) top view [13].

The sudden expansion or backward facing step is another type of flame holder used in ramjets, especially in solid fuel ramjets, because of its mechanical simplicity and convenience for solid fuel grain assembly. The abrupt change of area allows the formation of recirculation zones downstream the step where the mixing and flame stabilization could take place.

Cavities can also be used for flame holding. The main idea of cavity flame holders is to generate a recirculation zone with hot chemical products inside the cavity, so that it could act as a continuous ignition source. The use of wall cavities to stabilize and enhance

combustion is well known in the scramjet community. Ben-Yakar et al. [16] reviewed the detailed research efforts on cavity flame holders. Different from the ramjet combustor where the flow is subsonic, scramjets need the combustion to occur in supersonic flows. The typical flow structure of a cavity exposed to a supersonic flow is shown in Fig. 1-7. Shock waves at the rear end of the cavity (the sharp dark lines) can be seen clearly. In the cavity with a large length/depth ratio as shown in Fig. 1-7(a), vortices are generated inside. Due to reflections of acoustic waves at both ends, the cavity vortices are under strong oscillations [17]. Although the vortex oscillation can improve fuel-air mixing, the unsteady oscillation induces fluctuating pressure, density and velocity resulting in drag penalties. If the unsteady vortex shed out the cavity, it would be also not desirable for the purpose of flame stabilization, because this type of the unsteady vortex may lead to flame extinguishment. Furthermore, if the acoustic wave resonates with the unsteady heat release rate, the thermal-acoustic interaction may induce strong structure vibrations and flame extinguishment. For these reasons, an inclined back wall is usually applied in scramjet to reduce cavity oscillations, as shown in Fig. 1-7. In this configuration, the inclined back wall reflects acoustic waves to the main flow instead of to the front surface of the rectangular cavity, so that oscillations can be reduced and cavity vortex stability can be enhanced.

Fig. 1-7: Schlieren images and schematics of the flow structure of a cavity exposed to a supersonic flow: (a) a rectangular cavity, and (b) a cavity with inclined back wall [17].

1.2.2.2 Fluidic Flame Holders

Fluidic flame holders usually use jets to generate low speed recirculation zones [15] (see Fig. 1-8). For example, a jet is injected in the opposite direction to the incoming flow to produce a stagnation point and a low-speed zone as shown in Fig. 1-8(a). Fig. 1-8(b) and (c) illustrate transverse jet flame holders using a single-round jet and radially transverse jets to generate low-speed zones. Once igniters provide enough energy, mixtures in these zones can be ignited. This kind of flame holder could almost provide the same performance as bluff bodies, and it can also reduce the combustor weight significantly as it has few physical components. Another advantage of the fluidic flame holder is that the performance can be insensitive to the incoming flow velocity because the injecting velocity of the jets can be adjusted. In comparison, performance of geometric flame holders is highly dependent on the flow condition because the geometry is fixed. However, the main disadvantage of fluidic flame holders is that additional energy source and control system for the injection is needed.

Fig. 1-8: Schematic of fluidic flame holders using: (a) an opposing jet; (b) a single transverse jet; (c) radially transverse jets [15].

1.2.3 Trapped Vortex Combustor

1.2.3.1 Flame Stabilization Mechanism

Conventional combustors can employ swirl vanes, bluff bodies and backward facing steps to establish recirculation zones. These zones provide continuous sources of ignition through mixing the hot combustion products with the incoming fuel and air. Fig. 1-9(a) illustrates such a conventional gas turbine combustor in which the swirlers and liner air jets are used. The swirler establishes a primary axial recirculation zone in the front part of the combustor. This primary zone transports some of the hot products of combustion back to the fuel-air mixing region to ignite the incoming fresh mixtures. The air from the liner jets further mixes with the hot mixtures from the primary zone and makes the fuel burn completely before exiting. Though this type of combustor has been used successfully for half a century, it requires a complicated mechanical design to generate the axial primary zone and

to distribute the air entering the combustor. Another disadvantage of the swirl stabilized combustor is that the primary zone is sensitive to the flow condition. It may not be stable especially when the incoming flow speed is high.

Fig. 1-9: (a) Conventional swirl-stabilized combustor, (b) trapped vortex combustor [18].

Different from the swirl-stabilized combustor, the trapped vortex combustor (TVC) relies on a cavity to stabilize the flame. Fig. 1-9(b) illustrates a schematic of the TVC. When the cavity size is well designed, a large vortex can be formed and filled inside the cavity, rotating smoothly and will not shed out the cavity. This vortex is therefore named a “locked” vortex [19, 20] or “stable” vortex [4, 5]. Fuel can be injected either from the diffuser upstream or directly into the cavity. When the fuel is injected from the diffuser, it can mix with the air before entering the combustor. If the fuel is injected into the cavity directly, it is beneficial to inject the fuel in a way that the cavity vortex is reinforced. Furthermore, the bypass air is directed into the cavity to promote the fuel-air mixing in the cavity. As the recirculation zone is anchored in the cavity instead of exposing to the main flow, stable combustion can be achieved even when the main flow speed is high.

1.2.3.2 The TVC Development History

The locked or trapped vortices concept is not new in the aerodynamics community. It has been known to reduce the base drag or to recover pressure for aerodynamic advantages. Mair [19] and Little and Whipkey [20] investigated the sizing rule for cavities to reduce drag for blunt-base bodies. Gharib and Roshko [21] studied the cavity drag from the view of flow oscillations. Using cavities as a flame-holding device has also been rather popular in the scramjet community during the past couple of decades [16]. However, the use of cavities for subsonic combustors did not receive much attention until the early 1990s.

Air Force Research Laboratory (AFRL) First Generation TVC

The TVC concept was firstly evaluated in Air Force Research Laboratory for the low emission and high performance gas turbine combustor [22, 23]. Hsu et al. [4, 5] were the first to report using the TVC to stabilize flame for the new generation gas turbine combustors.

Fig. 1-10: First generation TVC [5].

The combustor proposed by Hsu et al. as shown in Fig. 1-10 is the first generation TVC. It is an axisymmetric can-type configuration and the cavity is formed between two disks attached in tandem by a spindle. The main air flow comes from the annular cross-section between the combustor casing and the fore-body. Gaseous propane (C_3H_8) and air are injected directly into the cavity from the surface of the second disk named the after-body. In the experiment, the cavity size could be changed conveniently by sliding the spindle attached to the after-body in or out, and the diameter of the after-body could also be changed by using washers. The experiment was conducted under atmospheric conditions. It was demonstrated that a stable flame can be obtained when a main vortex is formed and trapped in the cavity. This condition is achieved when the cavity length is $0.59D_0$ and the after disk diameter is $0.73D_0$ where D_0 is the diameter of the fore-body.

Katta and Roquemore [24, 25] numerically studied the first generation TVC to understand the vortex dynamics in cavities with different sizes. The optimum cavity size which can trap a vortex in the cavity is studied by numerical simulations under cold and combusting flows. Their reacting flow results indicated that the stable vortex determined from cold flow conditions is still stable in the combusting environment, which means the heat release of combustion does not significantly affect the cavity vortex dynamics. They

also obtained a correlation of the optimum cavity size that can trap a vortex efficiently as follows:

$$H/D_0 = -1.62D_1/D_0 + 1.79 \quad (1.2)$$

where H is the cavity length or the distance between the fore-body and the after-body, D_1 and D_0 are the diameter of the after-body and the fore-body respectively. The effect of fuel and air injection in the cavity on vortex dynamics was also investigated by Katta and Roquemore [26, 27].

The locked cavity vortex is crucial to flame stabilization in a TVC, as it provides a stable cavity flow which is good for flame anchoring. Sturgess and Hsu [28] studied the entrainment of the main flow into the cavity in TVC. It was found that the optimum cavity size relatively restricts the entrainment of the main air into the cavity compared to the non-optimal cavities. This indicates that although the stable cavity vortex is the best choice for flame stabilization, but is not the best solution from the point of view of fuel-air mixing. They also found that if the air entrainment into the cavity is desirable over a wide range, the cavity length-to-depth ratio should be slightly greater than 0.65 for gas turbine flow conditions. Sturgess and Hsu [29] did additional lean-blow-out (LBO) studies and demonstrated that the TVC has superior flame stability. For the TVC with larger cavity, Hsu et al. [30] found the blowout limit is susceptible to disturbances from the shear layers and the flow impingement on the front surface of after-body. Stone and Menon [31] used large-eddy simulations (LES) to investigate the mixing and combustion process in the TVC. They found the fuel-air mixing rates are enhanced by a higher annular air velocity for both cold and combustor flow conditions.

In summary, one obvious advantage of the TVC lies in its simple design, which can greatly reduce the combustor weight and cost. Besides, several impressive performance characteristics of the first generation TVC have also been demonstrated through experiments [4, 5, 22], including: (1) extremely lower LBO limits compared to the bluff-body and swirl stabilized combustors, (2) very low pressure drop over the cavity, (3) a wider operating range with a combustion efficiency at or above 99%, (4) significant reduction in NO_x emissions. The aforementioned findings promoted further studies of TVC subsequently.

AFRL Second Generation TVC

Fig. 1-11: Second generation TVC [23].

The second generation TVC, as shown in Fig. 1-11, is also an axisymmetric can-type configuration. Different from the first generation TVC, the second generation has the cavity surrounding the casing instead of in the combustor center, and there are several struts or flame holders in front of the cavity as shown in Fig. 1-11. The size of the cavity was approximately the same as the first generation TVC but with a slightly larger cavity length [22]. In the main flow, gaseous propane fuel (C_3H_8) was premixed with air before entering the combustion chamber. Additional air and fuel were injected from the front face of the after-body. A system of splitters and struts was used as flame holders to aid the mixing process and to promote the transport of hot combustion products from the cavity to the main flow. The experiment was operated under an overall stoichiometric condition with an increased inlet temperature (558 K) and a pressure slightly above atmospheric.

A single stable cavity pilot flame was obtained in the experiments even when the inlet velocity was increased to 150 m/s. Moreover, the cavity flame seemed to be more intense when the main flow speed is high. Excellent LBO capabilities and efficient operation over a wide dynamic range without combustion instabilities were also observed in the experiments. The struts were thought to make a significant contribution to the impressive performance as they acted as additional flame holders.

AFRL Third Generation TVC

The third generation TVC is shown in Fig. 1-12. Unlike the previous generations which are axisymmetric configurations, a two-dimensional sector is designed to allow easy fabrication and optical viewing of the cavity. Similar to the second generation, the main air flow is split by a splitter and several vertical struts. Air and fuel are injected into the cavity from multiple injectors. The key feature of the third generation is the use of liquid fuel (ethanol and JP-8). Nineteen cavity designs with different injection patterns were tested to determine the optimized location of the fuel and air injectors in the cavity [22]. The

experiments were conducted at atmospheric conditions, but with an increased air temperature of 650 K.

Fig. 1-12: Third generation TVC and illustration of its cavity [22, 23].

It was found that the critical requirement for good performance of the third generation TVC is to inject the fuel deep in the cavity so that most of the fuel is burnt in the cavity before it is entrained into the main air flow. Three optimum configurations for liquid fuel injection on the front face of the cavity are shown in Fig. 1-13.

Fig. 1-13: Three optimum injecting schemes for the third generation TVC [22].

General Electric High Pressure TVC

General Electric (GE) Aircraft Engines and the AFRL have been jointly developing the TVC for next generation gas turbine engines since 1996 [32]. Unlike the three previous generation TVCs in the AFRL which were all operated at atmospheric pressure condition, a high-pressure test rig was used, as shown in Fig. 1-14. The testing was operated at an inlet pressure up to 20.5 atm and inlet temperature up to 950 K with JP-8 liquid fuel. The test conditions covered many aspects of performance and operability parameters for both commercial and military gas turbine engines, including NO_x emission, combustion efficiency, exit temperature profile, structural temperature, ground ignition, altitude relight, and LBO limits. Fig. 1-15 presents some testing results of combustion efficiency, emission level, LBO limits and combustor outlet temperature profiles in the high pressure TVC.

Fig. 1-14: GE 2-passage and 3-passage TVC test rig configurations [32].

Fig. 1-15: High pressure TVC test rig data [18].

The TVC test rig demonstrated improved ignition, LBO and altitude relight performance, reduced NO_x emission, and wider operating range than the conventional swirl stabilized combustors. The experiments also verified the feasibility of the TVC under realistic gas turbine combustor operating conditions.

Ramgen Power Systems TVC

Ramgen Power Systems [33, 34] developed its unique advanced vortex combustion (AVC) concept aiming at improving combustion efficiency, low emission, robust flame stability, fuel flexibility and reduced cost. A schematic of the AVC is shown in Fig. 1-16.

Similarly to the TVC concept, the AVC uses a cavity to trap a vortex to stabilize flames. As shown in Fig. 1-16, a cavity is formed between the bluff body and the TVC module. The Ramgen AVC was operated at the premixed combustion mode as the incoming main flow consisting of gaseous methane fuel and air. Additional fuel and air were also injected to the cavity from the TVC module. The experiments were conducted at an inlet pressure of 18.7 atm and an inlet temperature of 678 K.

Fig. 1-16: Schematic of the AVC [33, 34].

Bucher et al. [33] compared the AVC with the bluff body stabilized flame under the same flow conditions. The bluff body stabilized combustor can be easily realized through removing the TVC module. As shown in Fig. 1-17, the TVC has significant large combustion efficiency over the bluff body mode especially at the fuel-lean condition ($\phi_{fe} < 0.6$).

Fig. 1-17: Combustion efficiency for bluff body and TVC [33].

NASA Glenn High Speed Flow TVC

Impressed by the good performance of TVC, NASA Glenn Research Center [22, 35] showed great interest in evaluating the TVC at high combustor inlet velocities, aiming at applying it into ramjet or scramjet engines. A full-scale 12-inch-wide rectangular prototypical TVC sector rig was tested using a liquid fuel at elevated pressure and temperature conditions by Hendricks et al. [35]. The performance of different fuelling schemes including upstream and cavity fuelling, reduced upstream fuelling, increased cavity fuelling and cavity-only fuelling were investigated. The maximum operating Mach number in the experiments reached near 0.7. The experimental visualization showed that the vortices remain trapped inside the cavity even when the flow speed was high and the flames in the cavity became more intense but smaller.

Some Other Significant TVC Developments

The U. S. Department of Energy (DOE) National Energy Technology Laboratory (NETL) and the Italian National Agency for New Technology, Energy and the Environment (ENEA) both selected the TVC concept as a promising technology for future ultra-clean energy plants [23, 36]. They aimed to develop stationary power applications using natural gas or hydrogen-rich syngas with a high efficiency and extremely low emissions.

TVC for ramjet applications has also received many attentions from researchers around of the world. Li et al. [37] experimentally investigated an axisymmetric ramjet combustor with a cavity flame holder as illustrated in Fig. 1-18. The ignition characteristics of the tested combustor were studied with different fuel injection schemes. Two different igniters were tested in the experiments: a hydrogen pilot flame igniter and a high energy spark igniter both installed in the cavity. It was found that the injection schemes had a great impact on ignition characteristics. Injecting the fuel along the wall in the upstream can provide a steady combustion, while the injection in the center of the main stream and in the cavity can hardly achieve a successful ignition.

Fig. 1-18: Schematic of the tested ramjet combustor with a cavity flame holder [38].

He et al. [39, 40] experimentally investigated the LBO limits and inlet temperature effects of a ramjet with v-gutters and cavity flame holders. A LBO limit when the combustor operated at the atmospheric pressure condition was obtained. Fei et al. [41] conducted preliminary numerical studies of the cold flow field structures in a ramjet with the TVC concept. Chen et al. [42, 43] numerically studied the TVC application for an inflow Mach number of 0.7. Their results showed that the cavity with an aspect ratio around 1.0 is sufficient to obtain the stable trapped vortex.

1.3 Objectives and Outlines of the Present Work

The main interest of the thesis focuses on small-scale ramjets with the potential application to UAVs, missiles and military projectiles. The principal objective is to understand and elucidate the flow field, the mixing and combustion processes in a TVC-type mini-ramjet combustor, with the intention to investigate the capability of the TVC concept for ramjets, and also to provide guidance for the design. Another objective is to study the spinning motion effects on the TVC, as the combustor may operate under such a motion if it is install on a spin-stabilized ramjet projectile

In comparison with costly experiments, numerical simulations offer a practical and efficient way for the design especially at the preliminary stage where many parametric studies are needed. Besides, the computational analysis also provides more available information for better understanding of the working processes in the designed combustor. In Chapter 2, the numerical method is first introduced. Verification and validation studies of various experiments using ANSYS FLUENT and a LES code (JETCODE) are then performed to evaluate the accuracy of various turbulence and combustion models. Based on

the validation results, suggestions on the numerical method and the selection of turbulence and combustion models for the study are summarized at the end of Chapter 2.

Chapter 3 focuses on the cold flow field characteristics of the combustor with cavities. The goal is to have an understanding of the cavity flow field and determine the optimum cavity size which is able to trap a stable vortex for flame stabilization. However, the combustor is only one of the components in an entire engine. A realistic combustor configuration with appropriate dimensions and operating conditions is required before the numerical study. Hence, an preliminary overall design of a ramjet engine is first presented to determine the flight conditions of the ramjet in Chapter 3. The one-dimensional stream thrust analysis method is then used to derive the combustor operating conditions. After obtaining the practical combustor inflow conditions, the cold flow field characteristics of the TVC are investigated. Finally, the proper size of the cavity is determined and present at the end of this chapter.

The investigation of gaseous propane fuel injection and mixing are investigated in Chapter 4. As the constraints of the space in the designed mini-ramjet combustor, only simple round orifice fuel injectors are used. As no air is directly injected into the cavity like in the large TVC for gas turbines reviewed above, different fuel injection strategies are investigated, including the injecting locations, and the effect of injection momentum flux ratio. After obtaining the proper injecting schemes, the combustion process is investigated in Chapter 5. Combustion performance of different injecting schemes is further evaluated.

In Chapter 6, the effects of spinning motion on the mini-ramjet combustor are studied and discussed. The goal is to investigate the flow field characteristics, mixing and combustion performance under the effect of a high spinning rate.

Finally, the conclusions, and suggestions for future work of this research are summarized in Chapter 7.

Chapter 2

Numerical Modeling

This chapter provides details of the numerical method that will be used to study the TVC. The conservation equations governing the gaseous combustion process are given first. It is followed by the reviews of commonly used turbulence models, combustion models for generic combustors and the brief descriptions of flow solvers used in this thesis. These models are then examined by several verification and validation cases, with the intention to find the suitable ones for the study of the TVC. Finally, the final selection of the turbulence model, combustion model and some strategies are summarized at the end of this chapter.

2.1 Governing Equations of Gaseous Combustion

The governing equations for gaseous combustion include the conservation of mass, momentum, energy, species and constitutive relations, which are expressed in Cartesian coordinates as follows:

$$\text{Continuity equation: } \frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad (2.1)$$

$$\text{Momentum equations: } \frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ji}}{\partial x_j} + F_{Corolis} + F_{Centrifugal} \quad (2.2)$$

$$\text{Species continuity equations: } \frac{\partial(\rho Y_k)}{\partial t} + \frac{\partial(\rho u_i Y_k)}{\partial x_i} = -\frac{\partial}{\partial x_i} J_k + \dot{\omega}_k \quad (2.3)$$

$$\text{Energy equation: } \frac{D(\rho h)}{Dt} - \frac{Dp}{Dt} = \frac{\partial}{\partial x_i} \left(\lambda \frac{\partial T}{\partial x_i} \right) - \sum_{k=1}^N \frac{\partial}{\partial x_i} h_k J_k + \frac{\partial(u_i \tau_{ij})}{\partial x_i} + \dot{Q} \quad (2.4)$$

where the diffusion flux of the k^{th} species is: $J_k = -\rho D_{m,k} \frac{\partial Y_k}{\partial x_i} - \frac{D_{T,k}}{T} \frac{\partial T}{\partial x_i}$.

The enthalpy of the mixture is $h = \sum_{k=1}^N h_k Y_k$ where the enthalpy of the k^{th} species is

$h_k = \Delta h_{f,k}^\circ + \int_{T^0}^T c_{p,k} dT$. The preceding conservation equations can be closed by using the state equation:

$$p = \rho RT \sum_{k=1}^N \frac{Y_k}{M_k} \quad (2.5)$$

and the global species conservation equation:

$$\sum_{K=1}^N Y_k = 1 \quad (2.6)$$

The Coriolis and centrifugal forces in Eq. (2.2) only exist in the rotating reference frame in Chapter 6. For the cold flow problems without combustion, the governing equations can be simplified: the continuity and momentum equations remain the same, while the source term $\dot{\omega}_k$ in the species continuity equations and the heat source term due to chemical reaction in the energy equation vanish.

Thermodynamic properties of the mixture are species-dependent, which can be calculated by using the ideal gas mixing law from each individual species. The specific heat at constant pressure, c_p , of each species is calculated as a piecewise polynomial function of temperature. However, the thermal conductivity, viscosity and diffusion coefficients for each species are still estimated as constants. This is because even if more accurate temperature-dependent values are implemented, they would not contribute much to a better prediction as they are always overwhelmed by their turbulent counterparts (i.e., the turbulence thermal conductivity, turbulence viscosity and turbulent diffusion coefficients) in real combustor flows.

2.2 Turbulence Modelling

Many engineering apparatus encounter turbulent flows. Compared with laminar flows which have a smooth flow feature, turbulent flows are characterized by their chaotic nature or three-dimensional randomness. Reynolds number, a quantity measuring the importance of inertial force and the viscosity, gives a way to distinguish laminar and turbulent flows. Modelling turbulence is always a key topic in the computational fluid dynamics (CFD) community. Among the existing turbulence models, the Reynolds averaged Navier-Stokes (RANS) equations are most widely used in engineering. Nowadays also sees the rapid increasing interests in large eddy simulation (LES) with the development of high performance computers.

2.2.1 RANS Turbulence Models

Instead of resolving the whole range of scales of space and time as in the direct numerical simulations (DNS), RANS splits each flow property into a mean value and a fluctuation by using the Reynolds decomposition:

$$\phi = \bar{\phi} + \phi' \quad (2.7)$$

where $\bar{\phi}$ is the time-averaged value, ϕ' is the fluctuation, and the time average of the fluctuation $\bar{\phi}'$ equals to zero.

Applying this time-averaging method to Eq. (2.1-2.4), additional terms consisting of the fluctuations appear. If the flow has a variable density, the number of the additional terms becomes very large, and the number of unknowns is also larger than the number of equations. Turbulence models are thus introduced to bring additional equations to close the set of equations. The larger the number of the unknowns, the more difficult the turbulence modelling becomes. Fortunately, a density-weighted averaging method, Favre averaging, can be applied to significantly reduce the number of unknowns for variable density flows. The Favre averaging is defined as follows:

$$u = \tilde{u} + u'' \quad (2.8)$$

where $\tilde{u}$ is the density-averaged velocity ($\tilde{u} = \overline{\rho u} / \bar{\rho}$), and u'' is the fluctuation.

The turbulent Favre-averaged equations are expressed as:

$$\text{Continuity equation: } \frac{\partial \bar{\rho}}{\partial t} + \frac{\partial (\bar{\rho} \tilde{u}_i)}{\partial x_i} = 0 \quad (2.9)$$

$$\text{Momentum equations: } \frac{\partial (\bar{\rho} \tilde{u}_i)}{\partial t} + \frac{\partial (\bar{\rho} \tilde{u}_i \tilde{u}_j)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\bar{\tau}_{ji} - \overline{\rho u_i'' u_j''} \right) \quad (2.10)$$

$$\text{Species continuity equations: } \frac{\partial (\bar{\rho} \tilde{Y}_k)}{\partial t} + \frac{\partial (\bar{\rho} \tilde{u}_i \tilde{Y}_k)}{\partial x_i} = -\frac{\partial}{\partial x_i} \tilde{J}_k + \tilde{\omega}_k \quad (2.11)$$

$$\text{Energy equation: } \frac{D(\bar{\rho} \tilde{h})}{Dt} - \frac{D\bar{p}}{Dt} = \frac{\partial}{\partial x_i} \left[(\lambda + \lambda_t) \frac{\partial \tilde{T}}{\partial x_i} \right] - \sum_{k=1}^N \frac{\partial}{\partial x_i} \tilde{h}_k \tilde{J}_k + \frac{\partial (\tilde{u}_i \tilde{\tau}_{ij})}{\partial x_i} + \tilde{Q} \quad (2.12)$$

where the diffusion flux of the k -th species: $\tilde{J}_k = -\left(\bar{\rho} D_{m,k} + \frac{\mu_t}{Sc_t} \right) \frac{\partial \tilde{Y}_k}{\partial x_i} - \frac{D_{T,k}}{\tilde{T}} \frac{\partial \tilde{T}}{\partial x_i}$.

Even so, there are still additional unknowns, for example, the Reynolds stresses $\overline{\rho u_i'' u_j''}$ in Eq. (2.10). Turbulence models are hence developed to model these terms.

The Boussinesq hypothesis [44] is a widely used approach to relate the Reynolds stresses to the mean velocity gradients or the rates of deformation:

$$-\overline{\rho u_i'' u_j''} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} \rho k \delta_{ij} \quad (2.13)$$

where μ_t is the turbulent or eddy viscosity, k is the turbulent kinetic energy and δ_{ij} is the Kronecker delta function.

In the classical two-equation models, such as the $k-\varepsilon$ or $k-\omega$ models (ε is the turbulence dissipation rate and ω is the turbulence specific dissipation rate), the turbulent viscosity μ_t is computed by k and ε , or k and ω respectively. Several popular RANS turbulence models with different model constants are briefly discussed in the following.

2.2.1.1 $k-\varepsilon$ Standard Model

The transport equations of k and ε in this model are given as follows:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon - Y_M \quad (2.14)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (2.15)$$

where G_k and G_b are the production of turbulence kinetic energy due to mean velocity gradient and buoyancy, and Y_M is the term due to compressible effects. The turbulent viscosity is computed as:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (2.16)$$

The model constants are listed in Table 2-1.

Table 2-1: Model constants in $k-\varepsilon$ standard model.

Model constants	$C_{1\varepsilon}$	$C_{2\varepsilon}$	$C_{3\varepsilon}$	C_μ	σ_k	σ_ε
	1.44	1.92	-0.33	0.09	1.0	1.3

Although this model was developed in the 1970s, it is still widely used nowadays in engineering due to it being simple, stable and reasonable prediction for many flows. However, it also has poor predictions for many flows, i.e., the flow with strong separation, axisymmetric jets, rotating flows and so on.

2.2.1.2 $k-\varepsilon$ Re-Normalization Group Model

Many attempts have been made to improve the performance of the standard $k-\varepsilon$ model. The $k-\varepsilon$ re-normalization group (RNG) model is one of them. In this model, the transport equations of k and ε are expressed as:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k - \rho \varepsilon - Y_M \quad (2.17)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} G_k - C_{2\varepsilon}^* \rho \frac{\varepsilon^2}{k} \quad (2.18)$$

where $C_{2\varepsilon}^* = C_{2\varepsilon} + \frac{C_\mu \eta^3 (1 - \eta/\eta_0)}{1 + \beta \eta^3}$, and $\eta = S \frac{k}{\varepsilon}$.

In the high Reynolds number form, the turbulent viscosity is computed by using equation (2.16). The model constants are summarized in Table 2-2.

Table 2-2: Model constants in $k - \varepsilon$ RNG model.

Model constants	$C_{1\varepsilon}$	$C_{2\varepsilon}$	C_μ	$\sigma_k, \sigma_\varepsilon$	η_0	β
	1.42	1.68	0.0845	0.7194	4.38	0.012

Compared with the standard $k - \varepsilon$ model, the RNG model can provide better prediction for flows with high stream curvature, high strain rate and wall heat and mass transfer. But it still could not predict round jet flows well.

2.2.1.3 $k - \varepsilon$ Realizable Model

The $k - \varepsilon$ realizable model is another improved model based on the standard $k - \varepsilon$. The transport equations of k and ε in this model are expressed as:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon - Y_M \quad (2.19)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b \quad (2.20)$$

where $C_\mu = \frac{1}{A_0 + A_s \frac{k U^*}{\varepsilon}}$ $U^* = \sqrt{S_{ij} S_{ij} + \tilde{\Omega}_{ij} \tilde{\Omega}_{ij}}$ and $\tilde{\Omega}_{ij} = \bar{\Omega}_{ij} - \varepsilon_{ijk} \omega_k - 2 \varepsilon_{ijk} \omega_k$, $\bar{\Omega}_{ij}$ is the

mean rate-of-rotation tensor viewed in a rotating reference frame with the angular velocity ω_k . The turbulent viscosity is still computed by using equation (2.16), and the model constants are listed in Table 2-3.

Table 2-3: Model constants in $k - \varepsilon$ realizable model.

Model constants	$C_{1\varepsilon}$	C_2	σ_k	σ_ε
	1.44	1.9	1.0	1.2

Compared with the standard $k-\varepsilon$ model, it has a better performance for round jets, flows with strong pressure gradients, and rotating flows.

2.2.1.4 $k-\omega$ Standard Model

The $k-\omega$ model is another two-equation model, in which the turbulence kinetic energy k and the specific dissipation rate ω are obtained from the following two transport equations:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k - D_k \quad (2.21)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - D_\omega \quad (2.22)$$

where G_k and G_ω are the generation of k and ω ; D_k and D_ω are the dissipation of k and ω . The turbulent viscosity is computed by:

$$\mu_t = \alpha^* \frac{\rho k}{\omega} \quad (2.23)$$

The constants in this model are listed in Table 2-4.

Table 2-4: Model constants of $k-\omega$ standard model.

Model constants	α_∞^*	α_∞	α_0	β_∞^*	β_i	R_β	R_k	R_ω	ζ^*	M_{t0}	σ_k	σ_ε
	1	0.52	1/9	0.09	0.072	8	6	2.95	1.5	0.25	2.0	2.0

The standard $k-\omega$ model is suitable for wall bounded flows such as boundary layers, low Reynolds number flows, and flows with separation. However, a dense mesh is usually required in the near wall region to have a better prediction. The prediction for flow separation is excessive and early.

2.2.1.5 $k-\omega$ Shear Stress Transport Model

Menter's shear stress transport (SST) $k-\omega$ model [45] is one of the most widely used RANS models due to its suitability in both the near wall region and the far field. Moreover, it has good behavior and prediction in flows with adverse pressure gradients and separating flows. The transport equations for k and ω are given by the following:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + G_k - D_k \quad (2.24)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + G_\omega - D_\omega + C_\omega \quad (2.25)$$

where C_ω represents the cross diffusion term. The eddy viscosity μ_t at high Reynolds numbers is computed as:

$$\mu_t = \frac{a_1 \rho k}{\max(a_1 \omega, S F_2)} \quad (2.26)$$

where S is the strain rate magnitude, and the constant $a_1 = 0.31$.

F_1 and F_2 are the blending functions of the inner (1) and the outer (2) region respectively, which can be expressed as:

$$\left\{ \begin{array}{l} F_1 = \tanh \left(\Phi_1^4 \right) \\ F_2 = \tanh \left(\Phi_2^4 \right) \\ \Phi_1 = \min \left[\max \left(\frac{\sqrt{k}}{0.09 \omega y}, \frac{500 \mu}{\rho y^2 \omega} \right), \frac{4 \rho k}{\sigma_{\omega,2} D_\omega^+ y^2} \right] \\ D_\omega^+ = \max \left(2 \rho \frac{1}{\sigma_{\omega,2}} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-10} \right) \\ \Phi_2 = \max \left(2 \frac{\sqrt{k}}{0.09 \omega y}, \frac{500 \mu}{\rho y^2 \omega} \right) \end{array} \right. \quad (2.27)$$

The turbulent Prandtl numbers are computed as:

$$\sigma_k = \frac{1}{F_1 / \sigma_{k,1} + (1 - F_1) / \sigma_{k,2}} \quad \text{and} \quad \sigma_\omega = \frac{1}{F_1 / \sigma_{\omega,1} + (1 - F_1) / \sigma_{\omega,2}} \quad (2.28)$$

The turbulence production of k and ω can be expressed as:

$$G_k = \min \left(-\overline{\rho u_i' u_j'} \frac{\partial u_j}{\partial x_i}, 10 \rho \beta^* k \omega \right) \quad \text{and} \quad G_\omega = \frac{\alpha \rho}{\mu_t} G_k \quad (2.29)$$

The turbulence dissipation of k and ω are:

$$D_k = \rho \beta^* f_\beta k \omega, \quad D_\omega = \rho \beta \omega^2 \quad (2.30)$$

The cross-diffusion modification term is computed as:

$$C_\omega = 2(1 - F_1)\rho \frac{1}{\omega \sigma_{\omega,2}} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (2.31)$$

The model constants in $k-\omega$ SST model are listed in Table 2-5. The values of other constants (α_∞^* , α_∞ , α_0 , β_∞^* , R_β , R_k , R_ω , ζ^* and M_{t0}) are the same as those in $k-\omega$ standard model shown in Table 2-4.

Table 2-5: Model constants of $k-\omega$ SST model.

Model constants	a_1	$\beta_{i,1}$	$\beta_{i,2}$	M_{t0}	$\sigma_{k,1}$	$\sigma_{k,2}$	$\sigma_{\omega,1}$	$\sigma_{\omega,2}$
	0.31	0.075	0.0828	0.25	1.176	1.0	2.0	1.16

The $k-\omega$ SST model inherits the advantages of $k-\omega$ and $k-\varepsilon$ models, but its dependency on wall distance makes it not very suitable for the flow with free shear layers. Again, it requires a dense grid in the near wall region.

2.2.1.6 Reynolds Stress Model

For all of the two-equation turbulence models mentioned previously, the isotropic eddy-viscosity hypothesis is implemented. This assumption provides reasonable results in many flows, but it is not suitable for the flows with strong swirling, rotation and rapid changes in strain rate. The effects of spinning motion on TVC combustor, which involves a highly swirling flow, are of interest in the scope of this thesis. The Reynolds stress model (RSM) [46], which deals with anisotropic strain rates is expected to be more suitable for this type of flow.

In RSM, instead of using Boussinesq's eddy viscosity assumption, the Reynolds stresses $\overline{\rho u_i'' u_j''}$ are directly computed. The transport of the Reynolds stress equation can be illustrated as follows:

$$\begin{aligned} \text{Rate change of} & \quad \text{Transport by} & \quad \text{Rate of} & \quad \text{Transport by} & \quad \text{Rate of} \\ \text{the Reynolds stress} & + \text{convection} & = \text{production} & + \text{diffusion} & - \text{dissipation} \\ & & & & + \text{Transport due to turbulent} & + \text{Transport due} \\ & & & & \text{pressure - strain interaction} & \text{to rotation} \end{aligned}$$

On the right hand side, the diffusion, the dissipation, and the pressure-strain interaction terms are needed to be modeled. For the pressure-strain term, the linear pressure-strain model, quadratic pressure-strain model, and the stress omega model are widely used.

As this model considers the anisotropic nature of turbulence, it has six additional partial differential equations and one additional transport equation for the six independent Reynolds stresses, which turns it into a seven-equation model. However, the additional equations and the highly coupled equations also mean a significant increase in the computational cost.

2.2.2 LES Turbulence Model

RANS turbulence models use a single model to describe the whole range of turbulence. In many engineering problems, the predictions are acceptable. However, the results of RANS have also revealed that perfect predictions with a single model for various applications seem to be formidable. One reason contributing to that is the different characteristic of turbulence with different scales. For example, the behavior of large eddies whose energy comes from the main flow is highly dependent on macroscopic features, i.e. the flow domain geometries and boundary conditions. In other words, they are anisotropic. In contrast, the small-scale eddies have a universal behavior and hence are almost isotropic. When the large eddy and the small eddy are treated in the same way, unsatisfactory predictions are not surprising. This is the situation happening in RANS. LES, however, is an approach that tries to solve this problem by dealing with large and small eddies separately.

The variables in LES are decomposed by a spatial filter operation into two parts: the resolved and the modeled. Unlike RANS, they are not the mean value and the fluctuation. The resolved corresponds to the unsteady fields with large scales, whereas the modeled (or the sub-grid scale) relates to the universal small eddies. After filtering the governing equations, several new terms called sub-grid-scale (SGS) stresses arise. The SGS stresses are the results of the small eddies and their interactions with the large scales, which need to be modeled. This makes the computational cost of LES considerable large, compared with RANS.

The Smagorinsky model [47] computes the eddy viscosity and the sub-grid kinetic energy as:

$$\begin{cases} \mu_t = C_\mu \tilde{\rho} \Delta^2 |\tilde{S}| \\ k = C_k \Delta^2 |\tilde{S}|^2 \\ |\tilde{S}| = \sqrt{\tilde{S}_{ij} \tilde{S}_{ij}} \end{cases} \quad (2.32)$$

where $\tilde{S}_{ij}$ is the density-weighted strain rate tensor, C_μ and C_k are the model coefficients, which are computed by a dynamic procedure proposed by Germano et al. [48].

2.3 Combustion Modelling

In the governing equations, there are two additional unclosed terms due to chemical reactions which have not been discussed yet: the species source term (or the reaction rate) $\dot{\omega}$ and the heat source term $\dot{Q}$. The heat source term can be easily computed by using the formation enthalpy, once the amount of each species is known. Thus, the burden is only on calculating the species source term $\dot{\omega}$. There are many combustion models which try to directly model $\dot{\omega}$, and therefore are categorized as the reaction rate method. Several widely used models based on this method will be first described in Section 2.3.1.

The other way to solve the combustion problem is called the primitive variable method. It introduces a new variable named the “mixture fraction”. The species mass fraction, temperature, and mixture density can be expressed as functions of the mixture fraction. A transport equation of mixture fraction thus replaces the species continuity equation, and provides a mixture fraction field. Hence, the scalar quantities can be obtained from the mixture fraction field. This method will be illustrated in Section 2.3.2.

2.3.1 Reaction Rate Method

The reaction rate method calculates the reaction rate or the species source term directly. A general form of non-reversible chemical reaction can be expressed as:

where ν' and ν'' denote the stoichiometric coefficients for reactants and products, and $k_{f,r}$ is forward reaction rate constant.

2.3.1.1 Eddy Dissipation Model

In many combustors, turbulence plays a key role in mixing the fuel and air together to achieve the stoichiometric condition before combustion occurs. After combustion, turbulence also convects hot products to cold reactants, so that the flame or heat can propagate. In the aerospace industry, the widely used hydrocarbon fuels burn very fast, the overall rate of reaction in the combustor is thus usually dominated by the mixing, or the combustion is mixing-limited.

The eddy dissipation (ED) model [49] is a combustion model that calculates the reaction rate by neglecting the chemical kinetics and only considering the effects of turbulence. It is one of the most popular combustion models, due to its high computational speed and yet can produce satisfactory results for mixing-limited diffusion flames. In this model, the net rate of production of species i is provided by the smaller of the two expressions below:

$$\dot{\omega}_i = v'_i M_i A \rho \frac{\varepsilon}{k} \min \left(\frac{Y_R}{v'_R M_R} \right) \quad (2.34)$$

$$\dot{\omega}_i = v'_i M_i A B \rho \frac{\varepsilon}{k} \frac{\sum_P Y_P}{\sum_j v''_j M_j} \quad (2.35)$$

where ε/k represents the timescale of turbulence eddies, the subscript R and P denote the reactants and products. A and B are the empirical constants ($A = 4.0$ and $B = 0.5$). Essentially, equation (2.34) illustrates the reaction rate is constrained by the lean species (fuel or oxidizer) in the mixture, while equation (2.35) indicates the reaction rate is also controlled by the dissipation of hot product eddies or the propagation of the flame. The smaller reaction rate calculated by the two expressions is determined as the final reaction rate for the combustion. It should be noted that the ED model is an infinitely fast chemistry model, as the chemical reaction occurs immediately when the reactants meet with each other.

2.3.1.2 Laminar Finite-Rate Model

Similar to the ED model, but in an opposite direction, the laminar finite-rate (FR) model calculates the reaction rate by only considering the chemical kinetics and neglecting all turbulence effects. This situation happens mostly in laminar flames.

The net rate of production of species i due to the N_R reactions is provided by:

$$\dot{\omega}_i = M_i \sum_{r=1}^{N_R} P_{i,r} \quad (2.36)$$

$$P_{i,r} = (v''_{i,r} - v'_{i,r}) \left(k_{f,r} \prod_{j=1}^N [C_{j,r}]^{\eta'_{j,r} + \eta''_{j,r}} \right) \quad (2.37)$$

$$k_{f,r} = A_r T^{\beta_r} e^{-E_r / R_0 T} \quad (2.38)$$

where $P_{i,r}$ is the molar rate of production, $C_{j,r}$ is the molar concentration of reactant species, and $\eta''_{j,r}$, $\eta'_{j,r}$ are the rate exponent for product and reactant species j . The reaction rate constant $k_{f,r}$ is expressed in Arrhenius form, where A_r is the pre-exponential factor, β_r is the temperature exponential, and E_r is the activation energy for this reaction.

2.3.1.3 Eddy Dissipation/Laminar Finite-Rate Model

This eddy dissipation/laminar finite-rate (ED/FR) model combines the previous two models, and locally chooses the smaller reaction rate as the final reaction rate. This model tries to consider both the mixing and the chemical kinetics in the combustion process. However, this model deals with the two factors separately, which is not true as they are coupled in a real combustion process.

2.3.1.4 Eddy Dissipation Concept model

Combustion happens only when fuel and air are mixed at the molecular level, and the activation energy of these molecules is large enough. Based on this idea, an extension model of ED, called the eddy dissipation concept (EDC) model was proposed by Magnussen [50, 51]. Compared with the ED model in which mixing only controls reaction rates, the EDC model assumes that there exists a “fine structure” in the combusting flow where micro mixing (not necessary stoichiometric) is achieved and most combustion happens there. The fine structure has a length scale defined as follows:

$$\xi^* = 2.1377 \left(\frac{\nu \varepsilon}{k^2} \right)^{1/4} \quad (2.39)$$

where ν is kinematic viscosity. In the fine structure regions, combustion only occurs in those regions which are heated sufficiently. A parameter, called reaction fraction, is defined to quantify this process:

$$\chi = \frac{Y_{pr}/(1+s)}{Y_{min} + Y_{pr}/(1+s)} \quad (2.40)$$

where Y_{pr} is the mass fraction of products, s the stoichiometric oxidant/fuel ratio by mass and $Y_{min} = \min(Y_{fu}, Y_{ox}/s)$. Then the reaction rate is obtained as:

$$\dot{\omega} = -C\rho \frac{\varepsilon}{k} \min \left(Y_{fu}, \frac{Y_{ox}}{s} \right) \left(\frac{\chi}{1 - \xi^{*2} \chi} \right) \quad (2.41)$$

where C is a model constant with a recommended value of 11.2, Y_{fu} and Y_{ox} are the mass fraction of fuel and oxidant.

The reaction fraction χ plays a crucial role in determining the final reaction rate. Not like the fast chemistry assumption in the ED model in which combustion immediately occurs whenever the fuel and oxidant encounter with each other (even the activation energy for molecules is not large enough), the reaction fraction in EDC model, however, eliminates the chance of combustion for those in which micro-mixing has been achieved but no

sufficient ignition energy. It should be noted that unlike the ED/FR model, which computes two models separately and locally chooses the smallest one as the final rate, the EDC model essentially combines the effects of chemical kinetics and turbulence together in one model, although the effects of chemical kinetics are realized by the reaction fraction instead of calculating the chemical kinetics.

2.3.2 Primitive Variable Method

2.3.2.1 Probability Density Function – Chemical Equilibrium (CM) Model

Different from the reaction rate method, which computes the species source term directly, the primitive variable method solves the transport equation of a variable called mixture fraction. Obtaining the mixture fraction field is useful, as the scalars like species mass fraction, mixture density and temperature can be easily computed once the mixture fraction is known. In this way, the species continuity and energy equations (2.3) and (2.4) are not needed. However, obtaining the scalars needs the help of the probability density function (PDF).

The PDF uses a statistics approach to calculate mean quantities. The probability function $F_V(\psi)$ of a random variable V is defined as:

$$F_V(\psi) = \text{prob}\{V < \psi\} \quad (2.42)$$

where $\text{prob}\{V < \psi\}$ means the probability that the value of variable V is smaller than the given value ψ . The PDF of the variable V is obtained by taking the derivative of the above probability function and can be expressed as:

$$P_V(\psi) = \frac{dF_V(\psi)}{d\psi} \quad (2.43)$$

For any PDFs, their values are non-negative. If a PDF is integrated in the whole interval, the integrated value must equal to one (as the possibility in the whole range is one), whose mathematical expression is:

$$\int_{-\infty}^{\infty} P_V(\psi) d\psi = 1 \quad (2.44)$$

Another important property of the PDF is, if a function $V(\psi)$ is dependent on ψ , its mean value can be obtained from:

$$\bar{V}(\psi) = \int_{-\infty}^{\infty} V(\psi) P_V(\psi) d\psi \quad (2.45)$$

As discussed previously, any scalar quantity in combustion (temperature, species concentration, and density) are dependent on the mixture fraction f , which indicates the mean values of any scalar can be obtained through the above formula. If applying the Favre-averaged, equation (2.45) can be expressed as:

$$\tilde{V}(f) = \int_{-\infty}^{\infty} V(f) \tilde{P}_V(f) df \quad (2.46)$$

Therefore, once the mixture fraction and the PDF are known, the scalar-mean quantities can be determined. The PDF can be observed and obtained from experiments, but many analytical functions have been built to approximate the measured PDF. The beta PDF, the most widely used in the combustion community, is applied in the current study.

The new variable, mixture fraction, also needs more elaboration. Instead of directly computing the species source terms, the mixture fraction is constructed to represent the fuel-air mixing process. The definition of the mixture fraction f is:

$$f = \frac{\xi - \xi_0}{\xi_1 - \xi_0} \quad (2.47)$$

where $\xi = sY_{fu} - Y_{ox}$, ξ is an artificial variable indicating the local fuel lean/rich conditions; s is the stoichiometric oxidant/fuel ratio by mass. For example, the reaction of gaseous propane:

The subscripts 1 and 0 denote the fuel and oxidant streams respectively. Hence, it can be known that, for non-premixed combustion where the fuel and oxidant streams are separated:

$$[Y_{fu}]_1 = 1 \quad [Y_{ox}]_1 = 0 \quad [Y_{fu}]_0 = 0 \quad [Y_{ox}]_0 = 1$$

The mixture fraction for the non-premixed combustion system $[f]_1$ equals to 1 at the fuel stream, which is the maximum value, and $[f]_0$ equals to 0 at the oxidant stream which is the minimum value in the system. The Favre-averaged transport equations of mixture fraction and its fluctuation are expressed as:

$$\frac{\partial}{\partial t}(\bar{\rho} \bar{f}) + \frac{\partial}{\partial x_i}(\bar{\rho} \bar{u}_i \bar{f}) = \frac{\partial}{\partial x_i} \left(\frac{\mu_t}{\sigma_t} \frac{\partial \bar{f}}{\partial x_i} \right) \quad (2.48)$$

$$\frac{\partial}{\partial t} \left(\bar{\rho} \overline{f'^2} \right) + \frac{\partial}{\partial x_i} \left(\bar{\rho} \bar{u}_i \overline{f'^2} \right) = \frac{\partial}{\partial x_i} \left(\frac{\mu_t}{\sigma_t} \frac{\partial \overline{f'^2}}{\partial x_i} \right) + C_g \mu_t \left(\frac{\partial \bar{f}}{\partial x_i} \right)^2 - C_d \rho \frac{\varepsilon}{k} \overline{f'^2} \quad (2.49)$$

where the model constant σ_t , C_g and C_d are 0.85, 2.86 and 2.0 respectively.

2.3.2.2 Chemical Equilibrium

It is mentioned earlier that the scalar quantities are dependent on the mixture fraction. The relation between these scalar quantities and the mixture fraction can be determined from chemical equilibrium (CE). CE is a state in which both the reactants and products concentrations are in steady state, or they have no tendency to change. It means that both forward reaction (the combustion) and backward reaction (the dissociation) exist, and their reaction rates in the opposite direction are the same. The dissociation introduces intermediate species, which reduce the system temperature, as the reverse reaction of combustion is an endothermic process. Compared with the combustion models with the one- or two-step combustion mechanisms which usually over-predict the maximum temperature, CE can provide more realistic temperature field because more intermediate species are included.

In this thesis, the chemical equilibrium composition is calculated by using the second law of thermodynamics – minimization of Gibbs energy. A practical constant pressure for the combustor of interest needs to be estimated properly first. The average combustor pressure obtained from other combustion models (i.e. ED model) is a good start. When the pressure is fixed and the mixture fraction is known, the equilibrium composition could be obtained by performing the calculation of minimization of Gibbs energy. In this way, the mixture fraction, temperature, density in the computational domain can be updated respectively until a convergent solution is obtained.

However, an important precondition to use this approach is that the flow residence time should be long enough, because the CE state needs a certain time to achieve. A dimensionless Damkhöler number (Da) is a useful non-dimensional parameter, as it quantifies the time scales between flows and chemical reactions as follows:

$$Da = \frac{\text{flow time scale}}{\text{chemical time scale}} \quad (2.50)$$

A schematic to illustrate the flow and combustion time scales is shown in Fig. 2-1. If Da is very large (usually greater than 10), that means the flow time scale is significant larger than the chemical reaction time scale. For engineering problems in aero-combustors, a typical flow time scale ranges from 10^{-4} s to 10^{-2} s. The chemical time scale, however, may change from 10^{-10} s to 1 s, mainly depending on type of fuels, pressure, temperature, chemical reactions and so on. As for hydrocarbon fuels used in aerospace, they burn very fast (usually in an order of $10^{-5} \sim 10^{-6}$ s [52]). Thus, when proper flame stabilizations are established (the flow residence time is increased), Da number for aero-combustor is

sufficiently larger than 10. In this case, chemical equilibrium can be assumed safely. The large Damkhöler number in aero-combustors also indicates that infinitely fast chemistry assumption is usually proper, and the combustion is thus mixing-limited.

Fig. 2-1: Time scales in chemically reacting flows [52].

For the CE approach used in the current study, the mixture fraction transportation equations Eq. (2.48) and (2.49) are computed, instead of the species conservation equations Eq. (2.3) and energy equation Eq. (2.4). Then based on the mixture fraction distribution, the thermodynamic related properties (density, species, and temperature) can be obtained by the chemical equilibrium calculations at a constant pressure. For conventional aero-combustors, it is a safe assumption that combustion occurs at a constant pressure condition. Through these processes, more realistic temperature and species prediction can be obtained. However, the constant pressure assumption is challenged in the spinning combustor. Because when the combustor and flow are in a stable spinning motion, a radial pressure gradient is generated to balance the centrifugal force. Hence, the combustor is not operated at a constant pressure condition any more. Other infinitely fast chemistry combustion models (i.e. the ED model) may be more suitable for the spinning combustor.

2.4 Numerical Solver and Boundary Conditions

Two types of flow solvers are used: the commercial flow solver ANSYS FLUENT v14.5 and a LES code – JETCODE from Stanford University. Several verification and validation cases have been performed to evaluate both the solvers, and also to justify the proper selection of different numerical models (turbulence and combustion models) for the

TVC. The boundary conditions of both flow solvers used in the following validations are also briefly summarized here.

2.4.1 RANS - FLUENT v14.5

The RANS computations in the present study are all performed using the commercial flow solver ANSYS FLUENT, which has been validated for many years. Specific boundary conditions used in this flow solver are also discussed concisely here.

Wall boundaries

A non-slip boundary condition has been assumed for the walls in the present study. In this boundary, the relative velocity of fluids at the wall surface is zero. For a stationary wall, the mathematical form of the condition is expressed as:

$$u = 0 \quad (2.51)$$

Besides, if an adiabatic assumption is made for the wall, that means there is no heat flux through them.

Pressure inflow boundary

It is a common boundary condition in which the total pressure, total temperature and the flow direction are specified. For incompressible flows, total temperature is equal to the static temperature, and Bernoulli's equation is used to relate the inlet velocity to the total quantities assigned:

$$p_0 = p_s + \frac{1}{2} \rho u^2 \quad (2.52)$$

In compressible flows the isentropic relation for idea gas is used to obtain the static pressure, velocity and static temperature at the inlet boundary.

$$\frac{p_0}{p_s} = \left(1 + \frac{\gamma - 1}{2} Ma^2 \right)^{\frac{\gamma}{\gamma - 1}} \quad (2.53)$$

$$\frac{T_0}{T_s} = 1 + \frac{\gamma - 1}{2} Ma^2 \quad (2.54)$$

Mass flow inflow boundary

This boundary condition is used in situations where the mass flow matters. In this boundary condition, the mass flow rate (or mass flux), total temperature and the flow direction are specified. If a total mass flow rate is specified, a uniform mass flux is assumed at the boundary as:

$$\rho u = \frac{\dot{m}}{A} \quad (2.55)$$

For ideal gas, the state equation is used to compute the density:

$$p = \rho RT \quad (2.56)$$

The static pressure is extrapolated from the flow field and the static temperature is computed from the total enthalpy:

$$h_o(T_o) = h_s(T_s) + \frac{1}{2} u^2 \quad (2.57)$$

Pressure outflow boundary

A static pressure is specified at this boundary. If the flow at the outlet is subsonic, the value specified here is used for the calculation. When the flow is supersonic at the boundary, the specified value will be neglected and the value of static pressure is extrapolated from the calculated flowfield in the interior.

2.4.2 LES - JETCODE

The LES computations in the present study are all performed by using the JETCODE. The LES solver was originally developed by Pierce et al. [53, 54] at Stanford University. The code is based on a finite volume method using a staggered grid. The dynamic LES sub-grid scale model of Germano et al. [55] and Lilly [56] has been used, which does not require external constants. The code has so far been validated on a wide range of flow computations [57-64].

Wall boundary

Neumann conditions for the scalar and pressure, and non-slip Dirichlet conditions for the velocity have been applied to the wall. For Neumann conditions, the gradient of scalars are calculated by using a staggered grid, while for Dirichlet conditions, the value is directly located on the boundary. A schematic of the staggered grid in two dimensions is shown in Fig. 2-2. The symbols show where variables are located: the black dots are the locations of density, pressure and scalar properties, the crosses indicate the locations of velocity in the x direction, and the rectangles are the locations of velocity in the y direction.

Fig. 2-2: Staggered grid at the wall boundary [53].

Inflow boundary condition

The LES simulation needs a three-dimensional, unsteady turbulence inflow. In this study, a fully developed turbulent assumption is made. An “inflow database” method proposed by Schluter et al. [59] is used. The database not only contains the mean quantities but also the time dependent turbulence quantities. It is generated by a separate code, which simulates a parallel, periodic, and fully developed duct flow. A corrective technique is enforced to satisfy the specified mean statistics. The mathematical description of this inflow conditions is written as:

$$u_{i,LES}(t) = \bar{u}_{i,mean}(t) + [u_{i,DB}(t) - \bar{u}_{i,DB}] \cdot \frac{\sqrt{u_{i,mean}^{'2}(t)}}{\sqrt{u_{i,DB}^{'2}}} \quad (2.58)$$

where the subscript *mean* denotes the specified mean quantities and *DB* stands for the database.

Outflow boundary condition

The convective condition, which is expressed mathematically below, is used for the outflow boundary.

$$\frac{\partial \phi}{\partial t} + c \frac{\partial \phi}{\partial n} = 0 \quad (2.59)$$

where ϕ stands for scalar variables or velocity components, c is the convection velocity which equals to the maximum outflow velocity, n denotes the direction normal to the boundary.

2.5 Verification and Validation

The TVC is a complex combination of many types of flows, which include pipe flows, flow separations, vortical flows, jets in the cross-flow, and the chemically reacting flows. Several validation and verification cases have been performed to determine the most appropriate numerical models.

2.5.1 Flow Over Backward Facing Step

As one of the simplest separating flows, the backward facing step has been extensively studied in experiments and widely used for CFD validations. The main feature of this flow is characterized by a large recirculation region downstream of the step as shown in Fig. 2-3(a). Several subsonic backward facing steps have been considered in this section.

The first test case is an axisymmetric backward facing step experiment performed by Devenport et al. [65]. Numerical simulations were performed by ANSYS FLUENT v14.5 with the pressure-based, double precision, and axisymmetric flow solver. The grid was generated by GRIDGEN v15. Different versions of the $k-\varepsilon$, $k-\omega$ and RSM models with enhanced wall treatment had been tested. For the turbulent boundary layer upstream the backward facing step, the $k-\omega$ models (the result of $k-\omega$ is overlapped with that of the $k-\omega$ SST) and the Reynolds stress model provide better prediction (see Fig. 2-3(b)). Fig. 2-4 shows the prediction of pressure and skin friction distribution downstream of the backward facing step. Although none of the model provides perfect agreement with the experimental data, the prediction by the SST $k-\omega$ model is the closest one to the measurements in the whole comparison range. As the SST model is known to be one of the most reliable turbulence models for aerodynamic applications [66], it is selected for further investigations.

Fig. 2-3: (a) Typical flow features of the backward facing step flow, (b) turbulent boundary layer upstream at $x = -10$ mm. U_{max} is the maximum velocity, and H is the step height.

Fig. 2-4: (a) Pressure coefficient and (b) skin friction coefficient distributions downstream of the backward facing step. X_r is the reattachment length.

The second test case is an axisymmetric turbulent base flow experiment performed by Merz et al. [67]. Numerical simulations were performed using the density-based, double precision, and axisymmetric ANSYS FLUENT solver with the SST $k-\omega$ turbulence model. The general flow feature consists of a strong recirculation region, a mixing region, a wake region and two stagnation points, as illustrated in Fig. 2-5(a). The predicted base pressure coefficient, which is equivalent to the base drag coefficient, is shown in Fig. 2-5(b) over a range of upstream Mach numbers. The numerical results are in good agreement with the measurements. The total and static pressures as well as the Mach number distributions on the

wake centerline of the base are shown in Fig. 2-6 for two different inflow conditions. Numerical predictions of static and total pressures are in reasonable agreement with the experimental data. However, the maximum Mach number is slightly under-predicted, and the rear stagnation point is over-predicted. It can be seen that the SST $k-\omega$ model is able to provide moderate predictions of turbulent base flows.

Fig. 2-5: (a) General flow features of a turbulent base flow, (b) base pressure coefficient as a function of Mach number. D is the diameter of the base.

Fig. 2-6: Static pressure, total pressure and Mach number distributions along the centerline of the wake region. Each plot is computed at different upstream Mach number Ma .

The third test case is an axisymmetric base flow with after-body attached to the base from Depres et al. [68] and Meliga et al. [69]. This configuration was selected for

investigation because the interaction between the after-body and the wake flow causes significant changes to the vortex formation, which is similar to the typical flow phenomenon in a TVC. Numerical simulations were performed using the density-based, double precision, and axisymmetric solver with the SST $k - \omega$ turbulence model. The general flow features are shown in Fig. 2-7 where it can be seen that the recirculation region downstream of the base is either deformed ($L/D = 0.6$) or divided into two separation bubbles ($L/D = 1.2$). L is the length of the after-body, D is the diameter of the main body, and $L/D = 0$ means no after-body. As shown in Fig. 2-8, the predicted wall and centreline pressure distribution are in close agreement with the measurements, indicating that the SST $k - \omega$ model is suitable for this type of flow.

Fig. 2-7: Turbulent base flow with after-bodies. L is the length of the after-body and D is the diameter of the main body.

Fig. 2-8: Wall and centerline pressure coefficient distribution of turbulent base flow with after-bodies.

2.5.2 Locked Vortex Base Flow

Little and Whipkey [20] systematically studied the drag and flow characteristics of a cylinder with a cavity attached behind. The cavity is formed by attaching a thin disk and a spindle behind as shown in Fig. 2-9. As it has a similar cavity configuration to the TVC of interest, it is a suitable case to validate the numerical method in cold flow conditions. Two-dimensional axisymmetric simulations with structured mesh were performed by using a pressured-based steady solver with the $k - \omega$ SST turbulence model.

Fig. 2-9: Schematic of a locked cavity after-base [20].

Fig. 2-10 shows the general flow features of the flow over base cavities with different sizes. According to Little and Whipkey [20], the flow field characteristics are highly dependent on the cavity size, which is measured by the after-disk diameter D_1 and the cavity length H . For convenience, they are all normalized by the fore-body diameter D_0 . If the cavity size is small, there are two counter-rotating vortices generated within the cavity region, as shown in Fig. 2-10(a). When the cavity is at a specific size (see Fig. 2-10(b)), a large vortex is generated and “locked” inside the cavity [20]. If the cavity size is further increases, there is still only one large vortex inside, but it has been elongated to fill the whole cavity region as shown in Fig. 2-10(c). The vector plots and the locations of vortex cores predicted by CFD and measured in experiments are compared in Fig. 2-11, and it can be seen that good agreements are obtained.

Little and Whipkey also found that the overall base drag is minimized when the locked or stable vortex is generated inside the cavity. The comparison of drag reduction $\Delta C_{D,P}$ between the experimental results and the current simulation results with a varying cavity length and after-body diameter is shown in Fig. 2-10(d). The drag reduction $\Delta C_{D,P}$ is obtained by comparing the current geometry which has a cavity behind the base with the one without cavity (no thin disk and spindle). As the friction is small compared with the pressure

drag, in the simulation results only the pressure drag is counted in the drag coefficient. It can be seen that the numerical results are in the same trend and match with the experimental results reasonably well. Briones's RANS $k-\varepsilon$ results [70] are also shown in the plot for comparison. In Fig. 2-10(d), the cavity size which provides the largest pressure drag reduction (the lowest point in the drag reduction line) is almost the same as that in the experiment. This test case further shows that the SST $k-\omega$ turbulence model is suitable for cavity-type flows.

Fig. 2-10: Flow features of (a) $D_1/D_0 = 0.75$ and $H/D_1 = 0.4$, (b) $D_1/D_0 = 0.75$ and $H/D_1 = 0.55$, (c) $D_1/D_0 = 0.75$ and $H/D_1 = 0.8$, and (d) drag reduction predictions and comparisons.

Fig. 2-11: Comparisons of locations of vortex cores at (a) $D_1/D_0=0.75$ and $H/D_1 = 0.35$, (b) $D_1/D_0 = 0.75$ and $H/D_1 = 0.6$. Upper half: from experiment [20], lower half: from CFD.

2.5.3 Sandia H₂ Diffusion Jet Flame

Compared with flames in a practical combustor, a jet flame is simple as it excludes many factors, such as the complex combustor geometry, different flame stabilization mechanisms, and complicated swirling flow fields. Before conducting a validation on the combustor with similar configuration of our interest (the one with a cavity), the Sandia H₂ jet flame [71] was first investigated to understand the general features of jet flames, and also to obtain experience and guidance for the study of the TVC that follows. The experiment selected to study was a 100% H₂ diffusion flame with a Reynolds number of 10,000 (based on the fuel injector diameter). Numerical simulations were performed using a three-dimensional, density-based, double precision and steady solver. The SST $k-\omega$ turbulence model was used to evaluate different combustion models. However, different turbulence models was also re-evaluated after finding appropriate combustion models, as it is known that the SST $k-\omega$ model may not be the best choice for the prediction of jets. The effect of Schmidt number was also evaluated.

2.5.3.1 Evaluation of Combustion Models

Different combustion models were tested, including the eddy-dissipation model (ED), the finite-rate model (FR), the eddy-dissipation/finite-rate model (ED/FR), the eddy-dissipation-concept model (EDC), and the chemical-equilibrium model (CE).

Fig. 2-12: Temperature contours of the H_2 jet flame predicted by the ED model.

The general feature of the diffusion jet flame, predicted by the one-step ED combustion model with SST $k-\omega$ turbulence model, is illustrated in Fig. 2-12. The flame is thin in the zone near the fuel injector at the bottom, but it occupies a large area when the reaction products diffuse downstream in the ambient air. Fig. 2-13 shows the comparisons of axial velocity and temperature between numerical results and the experimental data. All the combustion models tested provide good agreement in the axial velocity along the axis with the experiment, but all of them under-predict the velocity downstream of the flame (at $x/L = 0.5$ and $x/L = 1.0$). As for the temperature, the CE model provides better results in both the axial direction and the radial direction downstream, though relative large discrepancies exist especially in the radial direction (at $x/L = 0.5$ and $x/L = 1.0$). The discrepancies are mainly caused by the inaccurate prediction of velocity field and fuel-air mixing due to the turbulence model, which will be further investigated as follows.

Fig. 2-13: Axial velocity (upper) and temperature (lower) comparisons along the axis, at $x/L = 0.5$, and at $x/L = 1.0$. L is the experimentally observed flame length.

2.5.3.2 Evaluation of the Turbulence Models

In the previous section, the SST $k-\omega$ turbulence model was first used to test the effects of the combustions models. The chemical equilibrium model was found to be the best option to predict the flame temperature. It is known that the SST is not good for jets with strong shear layers. In this section, different turbulence models, including the $k-\omega$ SST, $k-\varepsilon$ realizable, standard $k-\omega$, standard $k-\varepsilon$, standard $k-\varepsilon$ with modified constant [49] (it is another turbulence model specially modified for jets) and the $k-\varepsilon$ RNG model will be tested based on the CE combustion model.

In Fig. 2-14, it can be seen that the results of standard $k-\varepsilon$ modified model matches with the experimental data best in the velocity fields, as the turbulence model constants are specially modified for jets. Regarding the temperature fields, different turbulence models produce comparable predictions. For example, in the axial direction, results of the $k-\varepsilon$

realizable and standard $k-\varepsilon$ modified model agree with the experimental temperature data better in the upstream region ($x/L = 0 \sim 0.6$), while the $k-\omega$ SST and the standard $k-\varepsilon$ predict better downstream ($x/L = 0.6 \sim 1.2$). In the radial direction, at half of the flame length ($x/L = 0.5$), the $k-\omega$ SST model agrees with the experiment well from $r = 0 \sim 15$ mm. But the results of standard $k-\omega$ model are closer to the data from $r = 20 \sim 60$ mm. A similar trend can also be found at the end of the flame ($x/L = 1.0$). Compared with the previous results, both the predictions of velocity and temperature fields have been much improved by the modified standard $k-\varepsilon$ model. Therefore, the standard $k-\varepsilon$ modified turbulence model is selected for the following jet flame simulations.

Fig. 2-14: Axial velocity (upper) and temperature (lower) comparisons along the axis, at $x/L = 0.5$, and at $x/L = 1.0$. L is the experimentally observed flame length.

2.5.3.3 Evaluation of the Turbulent Schmidt Number

The chemical equilibrium combustion model and the standard $k-\varepsilon$ turbulence model with modified constants were found to provide relatively better results for jet flames. They will be used to evaluate the effects of the turbulent Schmidt number, which controls the turbulent fuel air mixing or the distribution of the species in turbulent flows.

The Schmidt number is a non-dimensional number describing the importance of the viscous diffusion and the mass diffusion. It is defined as:

$$Sc = \frac{\nu}{D} = \frac{\text{viscous diffusion}}{\text{mass diffusion}} \quad (2.60)$$

where ν is the kinematic viscosity and D is the transport coefficient of mass. The turbulent Schmidt number is defined as:

$$Sc_t = \frac{\nu_t}{K} \quad (2.61)$$

where ν_t is the eddy viscosity, and K is the eddy diffusivity.

Fig. 2-15 shows the results of four turbulent Schmidt numbers: 0.2, 0.5, 0.7 and 1.0. The mean velocity field is almost not affected at all, while significant changes could be observed in the temperature field when the turbulent Schmidt number varies. In the axial direction, the case with turbulent Schmidt numbers of 0.5 and 0.7 agree best with the temperature from the experimental data. It can also be seen that the temperature profile is shifted to the downstream (see the temperature peak), when the turbulent Schmidt number increases from 0.2 to 1.0. The physical meaning of large Schmidt numbers is that the effect of mass diffusion is not that profound compared with viscous diffusion, and a small Schmidt number means a relatively fast mass diffusion process, or the stoichiometric mixing can be achieved “earlier”. That is the reason why the temperature peak of Schmidt number of 0.2 occurs closer to the fuel injector. In the radial direction, effects of Schmidt number can also be observed easily from the shift of lines, but no straightforward trends could be obtained directly, as a larger Schmidt number is preferable at the location of $x/L = 0.5$ while a smaller one predicts better at $x/L = 1.0$ downstream. When the turbulent Schmidt number is modified, the species distribution has to adjust and the chemical reaction regions will be changed accordingly. The heat release of reactions in turn affects the velocity field. It is found that a variable Sc is able to improve the prediction of flames as reported by Xiao et al. [72].

In summary, the combustion process is a complicated process including fluid dynamics, chemical reaction, heat and mass transportation, and the interactions among them. To predict the combustion process accurately is difficult, but reasonable results still could be obtained through careful considerations of the turbulence model, combustion model, and the turbulent Schmidt number. The three parts will be further checked to find the best combination for a TVC.

Fig. 2-15: Axial velocity (upper) and temperature (lower) comparisons along the axis, at $x/L = 0.5$, and at $x/L = 1.0$. L is the experimentally observed flame length.

2.5.4 Hsu's TVC Combustor

Similar procedures used in the previous jet flames were used in this section to study the combustion in a TVC. Fig. 2-16(a) shows the schematic of a TVC tested by Hsu et al. [4, 5]. Gaseous propane fuel was injected into the cavity through 8 injectors (1.75 mm in diameter) from the after-body in this experiment. The fuel injectors are sandwiched by two sets of air injectors (2.29 mm in diameter). Annular air was provided at a constant flow rate, which corresponds to a velocity of 24 m/s at the annular gap between the fore-body and the casing. The fuel was injected at a constant velocity of 22 m/s and the primary air in the cavity is injected at constant velocity of 24 m/s. The combustor overall equivalence ratio is around 0.21 and the cavity equivalence ratio based on the cavity fuel and air injections is about 4.4.

Fig. 2-16: (a) Schematic of the Hsu's TVC, (b) experimental temperature profiles [5].

In Fig. 2-16(b), the two valleys of the temperature distribution at $r = 10$ and 18 mm (see the line with triangles) correspond to the air injectors inside the cavity which can be used to find the radial direction of measurements. The black symbols in the bottom of Fig. 2-16(b) stand for the temperature fluctuations in the measurements. In order to save the computational time and expense, only $1/8^{\text{th}}$ of the whole combustor was calculated with rotational periodic boundary conditions. The total length of the computational domain is 8 times of the combustor diameter, so that the flow at the exit will be fully developed (based on the Hsu's paper [4, 5]). A three-dimensional steady and pressure-based solver was employed for the simulation. Velocity boundary conditions were used for the air and fuel inlets. The exit was specified as a pressure-outlet boundary condition with the ambient pressure of one atm. Adiabatic and no-slip boundary conditions were implemented for all of the walls. For the mixture properties, the specific heat c_p , thermal conductivity, and viscosity were calculated by a weighted-average mixing law. Whereas for each species the specific heat C_p was computed from a piecewise-polynomial function of temperature. The mass diffusivity, thermal conductivity, and viscosity for each species, which are laminar values essentially, usually are overwhelmed by the turbulence. Hence, these parameters for each species were set as constant.

2.5.4.1 Combustion Models

Five combustion models were evaluated for the new computational domains with the $k - \omega$ SST turbulence model. They are the finite rate (FR) combustion model in which the reaction rate is controlled by the Arrhenius expression or the chemical kinetics, the eddy dissipation (ED) model which computes the reaction rate based on the mixing, the finite rate and eddy dissipation (FR/ED) model that computes the reaction rate using both ways and

picking the slowest one as the final reaction rate, the eddy dissipation concept (EDC) combustion model which assumes that in the small scale the chemical kinetics control the combustion process and then the small scales are integrated to the entire domain from the perspective of turbulent flows, and the chemical equilibrium (CE) combustion model that solves the mixture fraction instead of directly computing the reaction rate as the above four models. It should be noted that in the first four models, the simple one-step reaction chemistry mechanism was assumed. In contrast, the constant pressure chemical equilibrium condition was assumed in the CE model so that there are 19 species involved.

Fig. 2-17: Temperature comparisons with different combustion models.

The temperature predictions at different locations are shown in Fig. 2-17. It can be seen that the FR and the ED/FR model have a relatively large discrepancy from the experiment at all the three locations compared with other models. That means the pure laminar finite rate chemistry cannot represent the turbulent combustion process in the TVC. The other three models (ED, EDC and CE) are able to capture the trend of the temperature distributions relatively well. Among them, the CE model has a relatively low temperature

profile, and match with the experimental results better, especially at the location $z = 40$ mm. As discussed previously, they are due to the contribution of more intermediate species considered.

In terms of computational time, the ED model obtains the results most quickly, and the EDC model is the most time-consuming. As for prediction accuracy, the CE model is the best, and it is followed by the ED model and the EDC model. Considering the EDC model used the simplest one-step reaction mechanism, its accuracy can still be improved when more complex reaction mechanism are implemented. However, the more complex mechanism in EDC model also means longer computational time. For these reasons, the CE combustion model is selected to test the effects of turbulence models in the following sections.

2.5.4.2 Turbulence Models

In the previous cold flow validations, it has been found that the $k-\omega$ SST model is most suitable for cavity flows. This section will evaluate the performance of different turbulence models in combusting flows. Hence five turbulence models will be tested based on the CE combustion model, which provides best agreement with experimental data. The five turbulence models are the $k-\varepsilon$ standard, $k-\varepsilon$ realizable, $k-\varepsilon$ RNG, $k-\omega$ standard and the $k-\omega$ SST turbulence model.

Fig. 2-18: Temperature comparisons with different turbulence models.

Fig. 2-18 shows the temperature predictions with different turbulence models. At the location $z = 5$ mm, they are all close to the measured data except the $k-\omega$ standard turbulence model, which produces a relatively large discrepancy. At the location $z = 24$ mm, the $k-\varepsilon$ models match the experimental results better at the radial location $= 0.01 \sim 0.02$ m, while the $k-\omega$ models over-predict the temperature fluctuations caused by the air and fuel injections. However, at the location $z = 40$ mm, the $k-\omega$ models predicts slightly better at the radial location $= 0.02 \sim 0.03$ m. Although none of the tested turbulence models are able to predict the temperature perfectly, they all capture the trend of the temperature profiles and the magnitudes reasonably well. It seems to be difficult to find a best turbulence model in terms of the temperature comparisons. However, the experimental data shown here only reveal the temperature information inside the cavity, which is only a small part of the overall combustor. The TVC also has a large recirculation zone behind the after-body disk in the main combustor where the adverse pressure gradients exist. It has been known that the $k-\omega$ SST is able to predict flows with adverse pressure gradients well. Besides, for both the

boundary layers near the wall and the regions in the far field, the $k-\omega$ SST can also provide good predictions due to it is a blending model of the $k-\omega$ and the $k-\varepsilon$ model. Hence, the $k-\omega$ SST turbulence model is chosen for the combusting flow calculations.

2.5.4.3 Effects of the Turbulent Schmidt Number

Five different turbulent Schmidt numbers (0.5, 0.7, 0.85, 1.0 and 1.2) were tested with the CE combustion model together with the $k-\omega$ SST turbulence model. The numerical results obtained with different turbulent Schmidt numbers are shown in Fig. 2-19. It is obvious that the turbulent Schmidt number affects the distribution of the temperature inside the cavity. At the location $z = 5$ mm, the large turbulent Schmidt number ($Sc = 1.2$) over-predicts the temperature at the radial location $= 0.005 \sim 0.01$ m. At the location $z = 24$ mm, the temperature fluctuations change with the different Schmidt numbers gradually. A similar observation could be seen at the location $z = 40$ mm. There are no significant changes in these different Schmidt numbers except for the significantly over-predicted temperature at $z = 5$ mm for $Sc = 1.2$. Overall, the Schmidt numbers ($Sc = 0.7$) predict the temperature profiles slightly better than the others.

Fig. 2-19: Temperature distributions with different turbulent Schmidt numbers.

2.5.4.4 Comparison of the Results Obtained from FLUENT and CONVERGE

The results obtained previously are all from ANSYS FLUENT v 14.5. Another flow solver named CONVERGE v2.2 has also been used to simulate the same TVC for comparison. Again, only 1/8th segment of the combustor with rotational periodic boundary conditions were computed. The $k-\varepsilon$ RNG turbulence model was employed for the simulation (it is the only option for RANS in this solver). A detailed chemistry mechanism (325 reactions and 53 species in total) designed by CHEMKIN was used for the finite rate combustion model. The same Schmidt number of 0.7 was chosen for CONVERGE.

Fig. 2-20: Temperature predictions comparison: FLUENT v14.5 and CONVERGE v2.2.

The obtained temperature profiles are shown in Fig. 2-20. It could be seen that the profiles of CONVERGE has a larger discrepancy compared with that of FLUENT v14.5 at $z = 5$ mm. At $z = 24$ and 40 mm, CONVERGE predicts the temperature distribution better, especially at the radial location from 0.02 to 0.04 m. However, the temperature fluctuations due to the injection of air and fuel in the cavity are not shown in the profiles at all. Though there are some discrepancies, the results of temperature fluctuation predicted by FLUENT v14.5 are better than those obtained from CONVERGE using RANS.

2.5.4.5 Comparison with Published Numerical Data

More numerical data of Hsu's TVC were compared with the numerical results from FLUENT. One is from Katta and Roquemore [24], in which both the Direct Numerical Simulation (DNS) and the $k-\varepsilon$ turbulence models were considered with a one-step finite rate combustion model. However, to reduce their simulation difficulties, they assumed the air and fuel injectors were injected from slots instead many small holes. The other data are from Stone and Menon [31] which used a real combustor geometry with LES. A single-step global chemical mechanism was used for combustion. However, the fuel they used in the simulations was methane which has more validated data available instead of the propane that was actually used in both Hsu's experiment and Katta's simulations (the current simulations use gaseous propane as well). The comparisons of the current results and the other numerical data are shown in Fig. 2-21. It can be seen that the results from FLUENT match the experimental data better compared with other published data. For these reasons, FLUENT v14.5 was determined finally to perform the TVC combustor calculations.

Fig. 2-21: The comparisons of our numerical results to the published numerical data.

2.5.5 Prediction of the Lifted Flame

An infinitely fast chemistry model (e.g. the ED and CE combustion model) assumes chemical reaction immediately occurs once fuel and oxidizer meet with each other. For a simple jet diffusion flame, using infinitely fast chemistry models means that flame stabilization is always numerically ensured. Hence in simulations with these combustion models, the flame always starts right at the tip of fuel injector, as shown in Fig. 2-12. In laminar flames or low speed turbulent flames, it is possible that the flame is attached at the rim of the injector. However, when the inlet flow speeds are high, the flame is found to be lifted a certain distance away from the injector as shown in Fig. 2-22. The stabilization mechanism of this lifted flame is the so-called “triple flame” [73], which refers to the two premixed flames (rich premixed and lean premixed) and a diffusion flame. In the triple flame where the fuel flow speed is high, the ignition point of the flame is not attached at the rim, but located at a certain distance downstream. The distance between the injector and the flame is the lift-off distance.

It can also be found that there is a premixing zone between the injector and flame where the fuel and oxidizer mix with each other as shown in Fig. 2-22. Although the fuel is mixed with air in the premixed zone, combustion does not occur as the activation energy of molecules is still not sufficient. In other words, additional heat or ignition energy is needed to initiate combustion. In this type of diffusion flame, the flame stabilization is ensured by the downstream triple flame structure, which provides heat to the upstream mixture. If the flow speed further increases, the lift-off distance will increase as well. When the heat of downstream triple flame cannot sufficiently preheat the incoming mixture, the flame could be blown off.

Fig. 2-22: Diffusion flame stabilized by a triple flame [73].

Fig. 2-23: Lift-off distance predictions against the experimental data from [74], the lift-off distance measured in the experiment is marked as the red dashed line.

In such a diffusion flame with a triple flame structure, the infinitely fast chemistry models which always predict the flame attached at the injector rim is not accurate. The EDC model which considers the activation energy for combustion is able to predict the lifted flame phenomenon. Fig. 2-23 shows the prediction of a propane jet flame [74] in ambient air with three different combustion models: the infinitely fast chemistry model (the ED and CE models) and the EDC model. It could be seen that the infinitely fast chemistry models all predict the flame occurs right at the rim of injector as expected, while the EDC model predicts the flame starts at certain distance downstream. The lift-off distance observed from the experimental is also provided and marked as the dashed line in the plot. The current EDC model with a one-step mechanism slightly under-predicts the flame lift-off distance. However, more accurate prediction of the lift-off distance can be obtained if multiple steps chemistry mechanism is considered.

In a ramjet engine, the flow enters the combustion chamber is compressed by shock waves at the inlet. After the compression process, the Mach number of the flow decreases from supersonic to low subsonic. Meanwhile, the static temperature of the incoming air is increased significantly to around 700 K ~ 900 K. It has been found that the preheated air can reduce the lift-off distance dramatically [75], so it is expected that triple flame structure or the lift-off distance of a jet diffusion flame will be reduced in the ramjet combustor with hot ambient air.

A propane jet flame in 900 K air is performed by using the EDC model, and the predicted temperature contours are shown in Fig. 2-24. For comparison, the flame in ambient air (300 K) is also provided. It could be seen that the lift-off distance decreases in the hot air, and the flame is almost attached to the injector, which is similar to the flame feature predicted by the infinitely fast chemistry models.

Fig. 2-24: Lift-off distances predicted by EDC model at: (a) 300 K and (b) 900 K ambient air.

2.5.6 Rotating or Swirling Flows

Another objective of this thesis is to study the effect of the strong spinning motion on the TVC. Spinning or swirling flows are commonly encountered in gas turbine engines. For example, swirlers are used in the gas turbine combustors to generate highly swirling flows, so that efficient fuel-air mixing and flame stabilization can be achieved inside the combustion chamber with limited space. In this section, computations of a confined jet

without and with swirl have been performed to validate the numerical method for swirling flows.

The benchmark data come from the experiment of Dellenback et al. [76], whose experimental configuration is a simple axisymmetric sudden expansion. Different levels of swirl are generated by additional flow injected through sets of tangential slots in the upstream. The experiment provides detailed mean flow (i.e., non-fluctuating) velocity profiles measured at several different stations for comparison. Rotating or swirling flows are commonly characterized by a non-dimensional swirl number, which is defined as the ratio of axial flux of angular momentum to R times the axial flux of the axial momentum:

$$S = \frac{1}{R} \frac{\iint_A r u_\theta u_x dA}{\iint_A u_x^2 dA} \quad (2.62)$$

where u_x , and u_θ are the local axial and tangential velocities, A is the area, and R is the duct radius. When the duct is an annulus, R is the outer radius of the annulus. When the swirl number increases above a critical value, the high degree of swirl results in a central recirculation bubble, which is a form of vortex breakdown. The vortex breakdown phenomenon is characterized by a bubble-like recirculation structure with an internal stagnation point and reversed flow which are formed when the swirling flow undergoes a sudden expansion. This flow feature is also commonly encountered in gas turbine combustors. The resulting structure is nearly axisymmetric along the rotating axis and is located immediately downstream of the sudden opening. Unlike its non-swirling counterpart in which the recirculating flow is in the forward/reverse directions, the vortex breakdown bubble also has a strong swirling component.

Predicting the swirling flow accurately with CFD is difficult, not only because of the strong coupling of axial motion, swirl motion and the static pressure field, but also due to the anisotropic nature of turbulence in strong swirling flows. In RANS turbulence models, an isotropic turbulence is assumed for all the popular two-equation models, so that accurate predictions of strong swirling flows with these models are far to reach. The Reynolds stress model (RSM), however, includes the anisotropic turbulence and therefore is usually recommended for this type of flow. Another practical difficulty during modeling is that the swirling flow is highly sensitive to the upstream flow conditions or the inflow boundary conditions. If the inflow boundary conditions are not well-defined, significant discrepancies can be obtained.

In this validation, two flow conditions have been studied by using two-dimensional axisymmetric steady computations with RSM: one is a purely sudden expansion flow

without swirl ($S = 0$) at a Reynolds number $Re = 30,000$, the other is a strong swirling flow ($S = 0.6$) at the same Reynolds number.

The velocity fields of the two cases predicted by RSM are shown in Fig. 2-25 and Fig. 2-26. In each figure, the velocity profiles along the radial direction are plotted for different x/D axial locations. To be consistent with the experimental data, all the velocity profiles have been normalized by the axial maximum velocity, which is the centerline value of the inflow axial velocity. To avoid the velocity profiles from different axial slices overlapping with each other, they have also been shifted in the horizontal direction accordingly, so that the horizontal axis here is not the normal single continuous coordinate, but a combination of multiple axes ranging from 0 to 1.0 as shown in Fig. 2-25. The left most profiles are the mean velocity profiles at the inlet, which are implemented in the simulations through the User Defined Functions (UDF) by using the data from the experiments.

When the swirl number is zero ($S = 0$), RSM provides good mean velocity predictions in the whole flow domain as shown in Fig. 2-25, although small deviations can be found at downstream locations along the axis (at around $x/D = 0.6$). When the flow has a strong swirl ($S = 0.6$), the well-known vortex breakdown in swirling flows is characterized by the occurrence of reverse flow on the centerline. The reverse flow region can be seen in Fig. 2-26(a) where the boundary at which the axial velocity turns negative has been drawn for the RSM prediction. For the experimental results indicated by the square symbols in Fig. 2-26(a), the vortex breakdown bubble is also located at around $x/D = 0.2$ to 2.0 , as the axial velocities there have negative values in the region close to the centerline. Because the RSM model includes the anisotropic features of turbulence, it is expected to provide better predictions for strong swirling flows than simpler turbulence models (such as two-equation models). It is also found that the turbulence intensity used in the simulations also affect the predictions. A 20% uniform turbulence intensity, which is a relatively accurate representation of the experimental data, is applied in current study. When other turbulence intensity (say 12 %) was implemented, large discrepancies were observed. The validation results here indicate that the RSM simulation is able to predict the general flow features of the swirling flow, and can be used to investigate the spinning TVC.

Fig. 2-25: Validation results with RSM at swirl number $S = 0$.

(a)

(b)

Fig. 2-26: Validation results with RSM at swirl number $S = 0.6$: (a) axial velocity, (b) tangential velocity.

Fig. 2-27: Validation results of LES for a confined jet without and with swirl.

The validation cases were also studied by LES using the flow solver JETCODE. As discussed previously, swirling flows are sensitive to the inflow boundary conditions. In JETCODE, an adjusted method firstly developed by Schluter et al. [59] was used to implement the inflow conditions for the swirling flows. The main idea of this method is to generate an LES inflow condition database based on the known experimental mean (i.e. non-fluctuating) data and fully developed turbulence statistics. The averaged LES results obtained by JETCODE are shown in Fig. 2-27. It can be seen that good predictions for both the non-swirling and swirling flows have been obtained.

The LES code has advantages in studying rotating flows especially for the unsteadiness of flow, but it also has several limitations. For example, it can only handle a very simple geometry, such as those can be directly obtained by a two-dimensional extrusion or a two-dimensional rotation. In the designed TVC, fuel is injected through several simple

round orifices (which will be present in the flowing chapters), and the existence of these holes invalid the axis-symmetry of the combustor. Therefore, JETCODE can only be used for the cold flow field in which the TVC is axisymmetric. The mixing and combustion performance still have to rely on the commercial software ANSYS FLUENT.

2.6 Summary

The numerical method, verification and validation studies have been present in this chapter. Based on the findings from these numerical results, the selections of the flow solver and numerical models for the TVC are summarized in Table 2-6.

Table 2-6: Selections of the flow solver and numerical models for TVC.

		Flow solver	Turbulence model	Combustion model	Schmidt number
Non-spinning combustor	Flow field characteristics	FLUENT & JETCODE	$k-\omega$ SST & Smagorinsky	-	-
	Fuel mixing	FLUENT	$k-\omega$ SST	-	0.7
	Combustion			CE	
Spinning combustor	Flow field characteristics	FLUENT & JETCODE	RSM & Smagorinsky	-	-
	Fuel mixing	FLUENT	RSM	-	0.7
	Combustion			ED	

As shown in Table 2-6, for the non-spinning combustor, both FLUENT and JETCODE will be used to study the flow field characteristics. The results and discussion will be present in Chapter 3. In Chapter 4, the TVC with fuel injection will be investigated by using FLUENT with the $k-\omega$ SST turbulence model and a Schmidt number of 0.7. The combustion performance of TVC will then be investigated and discussed in Chapter 5, with the chemical equilibrium (CE) model.

Chapter 6 will focus on the effects of spinning motion on TVC. The flow field characteristics without fuel injection and combustion will be studied by using the JETCODE and FLUENT, while the mixing and combustion will still rely on FLUENT. It should also be noted that the ED model instead of the CM model is selected for the spinning combustor. This is because the CM model computes the combustion at a constant pressure condition, but a large radial pressure gradient exists in a strongly spinning combustor which violates the constant pressure combustion assumption. Hence, the ED model will be used for combustion in the spinning TVC.

Chapter 3

Flow Field Characteristics of the TVC in a Mini-Ramjet

The flow field characteristics of TVC in mini-ramjets are investigated in this chapter. The main objectives are to gain insights on the cavity flow field, and also to determine the proper cavity size that can be used for flame stabilization. This chapter will first perform a preliminary ramjet engine design to derive operating conditions for the TVC. The preliminary design is based on a flight vehicle, i.e. a missile, instead projectiles launched by artilleries where the ballistics of projectiles should be considered. Then a detailed numerical study of the flow field characteristics with the derived operating conditions will be followed.

3.1 Inflow Conditions for the Combustor

Realistic and accurate operating conditions, for example the inflow mass flow rate, velocity, pressure and temperature, are important to the combustor design. Directly inheriting the information from a well-designed inlet is certainly the best solution. However, such an inlet is not available in this study. Inflow conditions for the combustor will thus be estimated by the thermodynamic cycle analysis. In this section, a preliminary overall design of an axisymmetric ramjet engine is conducted to obtain inflow conditions for the combustor of interest.

An axisymmetric ramjet engine with a diameter of 100 mm operating at Mach 4 (on-design point) is of interest in this work. The combustor configuration as shown in Fig. 3-1 is proposed based on the first generation TVC proposed by Hsu [5], which has a cavity in the center of the combustor. The proposed length of the engine is about 6 times of the combustor diameter. The estimated weight of the engine is about 200 N (for comparison, M107 155 mm projectile with a length of 605 mm has a weight of 440 N). Fig. 3-1 also provides the reference station locations: station 0 stands for the free stream condition or the undisturbed incoming flow condition, station 3 and 4 are at the combustor inlet and outlet respectively, and station 9 is at the exit of nozzle. It should be noted that the station numbers are used in a

way that is consistent with gas turbines. Hence, the skipped station numbers in Fig. 3-1 mean the stations for compressors, turbines and so on, which do not exist in ramjets.

In the following sections, the proper flight altitude for the ramjet is determined first. Then a stream thrust analysis is performed to evaluate the engine overall performance, and to obtain the combustor operating conditions.

Fig. 3-1: Schematic of the proposed mini-ramjet engine.

3.1.1 Determination of the Flight Altitude

Air-breathing engines, depending on their flight speeds, are usually operating efficiently within a range of certain altitudes, or within a range of a certain dynamic pressure. As shown in a typical flight corridor for air-breathing vehicles in Fig. 3-2, the slender band shows the proper altitudes for the air-breathing engines flying at different Mach numbers with a constant dynamic pressure. For Mach 4, the engine should fly roughly within 18 – 28 km in altitude.

Fig. 3-2: A typical schematic of the air-breathing flight corridor [77].

Fig. 3-3: Zero-lift drag coefficient [78].

Fig. 3-3 is an example of a zero-lift drag coefficient plot for an axisymmetric missile with a nose, a slender body and a boat tail at different flight speeds. As this missile has a similar layout with the designed ramjet, the zero-lift drag coefficient of 0.2 can be selected as an approximation for the ramjet engine at Mach 4.0. However, considering the lift-induced drag and the engine installed drag during the flight, the overall drag coefficient of the engine may increase (say conservatively, by 50%) to 0.3. That means the thrust coefficient of the designed engine in this preliminary stage has to be at least 0.3.

Fig. 3-4: Lift/drag ratios for a circular cross section body and a lifting body flying at a small angle of attack [78].

Another important non-dimensional parameter is the lift/drag ratio. From Fig. 3-4, it can be known that for a circular cross section body like the designed engine, the average lift/drag ratio at Mach 4.0 (with a dynamic pressure of 2000 lb/ft²) is around 2. Hence, the

lift coefficient is 0.6, when the drag coefficient is determined as 0.3. The lift is the force that balances the weight of the flying engine. As the estimated total weight of the small ramjet engine is about 200 N, put them into the lift formula:

$$L = C_L \cdot q_\infty \cdot A \quad (3.1)$$

where L is the lift, C_L is the lift coefficient, q_∞ is the dynamic pressure and A is the reference area. The dynamic pressure can be computed as 41,410 Pa, which corresponds to an altitude of 22.5 km for the considered Mach 4 flight speed. Compared with the flight envelope for air-breathing engine as shown in Fig. 3-2, 41,410 Pa (22.5 km) falls in the operation band, which means the flight altitude for the small ramjet is adequate. Once the altitude is determined as 22.5 km, the static pressure, the static temperature, density and the speed of sound can be easily obtained from the 1976 standard atmosphere [79]. For convenience, they are listed here:

Table 3-1: Standard atmosphere at 22.5 km.

Altitude [km]	Temperature [K]	Pressure [Pa]	Density [kg/m ³]	Speed of sound [m/s]	Viscosity [Pa · s]
22.5	219.15	3699.54	0.0588091	296.767	0.0000144639

3.1.2 Stream Thrust Analysis

When the flight conditions of the ramjet are known, the inflow boundary conditions for the combustor component could be estimated by using the stream thrust analysis [77]. The stream thrust analysis method is based on the classic Brayton cycle analysis, but also considers the mass, the momentum and energy contributed by fuel, the geometry of the combustion chamber and the working condition of the nozzle. The name “stream thrust” comes from the stream thrust function derived from the thrust equation:

$$F = \dot{m}_9 A_9 - \dot{m}_0 A_0 + (p_9 - p_0) A_9 = (pA + \dot{m}A)_9 - (pA + \dot{m}A)_0 - p_0 (A_9 - A_0) \quad (3.2)$$

where F is the engine thrust, p denotes the static pressure, A is the cross section area, and the $\dot{m}$ is the mass flow rate. The subscripts 9 and 0 correspond to the station number as shown in Fig. 3-1. When both sides are divided by the mass flow rate of air, the specific thrust is obtained:

$$\frac{F}{\dot{m}_0} = (1 + f) S a_9 - S a_0 - \frac{RT_0}{U_0} \left(\frac{A_9}{A_0} - 1 \right) \quad (3.3)$$

where $S a_9$ and $S a_0$ are the stream thrust functions, f is the fuel-air mass ratio. One of the obvious advantages of this method is the specific thrust arises naturally in this analysis.

Using the stream thrust analysis to determine the flow conditions at different stations, it is necessary to circumnavigate the entire T - s diagram as shown in Fig. 3-5, beginning from and returning to the known free stream conditions with some assumptions. Firstly, the flow going through the engine is assumed as a simple one-dimensional flow. Secondly, it is assumed that the air is in the equilibrium state at all times, so that all the intensive thermodynamic properties can be specified by two known independent thermodynamic properties. Additionally, the working medium is treated as the calorically perfect gas for each of the four thermodynamic processes. Hence, the specific heat c_p is a constant during each process. However, due to the large temperature variance across the entire processes, different temperature-dependent c_p values will be used for each process based on the arithmetic average temperature between two stations. Thirdly, the combustion process is replaced by a heat addition process and a mass addition process. The amount of heat added to the process is the same as that supplied by the combustion of fuel. The complex chemistry during combustion including the change in the chemical constituents of air is neglected here. Although all of the assumptions are not perfectly met in practice, they capture the key parts and provide a reasonable representation of the nature for the engine.

Fig. 3-5: Schematic of the air-breathing engine temperature-entropy diagram [77].

There are four processes in the Brayton cycle as shown in Fig. 3-5, which are illustrated by the station numbers. The process from point 0 to point 3 is the adiabatic compression process. The free stream static temperature T_0 increases to combustor inlet temperature T_3 after compression with the increase of entropy from s_0 to s_3 due to the irreversible processes like friction and shock waves. An isentropic process which has no entropy change is also illustrated in Fig. 3-5 from point X to point 3. A useful parameter,

adiabatic compression efficiency, can be defined to describe the adiabatic compression process as follows:

$$\eta_c = \frac{h_3 - h_X}{h_3 - h_0} = \frac{C_{pc}(T_3 - T_X)}{C_{pc}(T_3 - T_0)} \leq 1 \quad (3.4)$$

The process from point 3 to point 4 is the constant pressure combustion process. The combustor inlet temperature T_3 increases to the combustor exit temperature T_4 after absorbing the heat released by the combustion of fuel. The assumption of constant pressure is preferable for some reasons. First, it is true for most of the real aero-combustors as the velocity in the burner is reduced as close to 0 as possible for both gas turbine combustors and other high speed combustors (not including the scramjet) so that the pressure inside the burner is almost a constant. Second, from the point of view of aerodynamics, the constant pressure helps to reduce the possibilities of boundary layer separation. Third, it also brings much convenience for mathematical manipulations. In this process, the combustion efficiency is defined as:

$$\eta_b = \frac{\int_3^4 T ds}{f \cdot h_{PR}} \leq 1 \quad (3.5)$$

where f is the fuel-air ratio by mass and h_{PR} is the heating value of the fuel.

The process from point 4 to point 9 is the adiabatic expansion process. The nozzle inlet temperature T_4 drops to the nozzle outlet temperature T_9 when the flow expands from burner static pressure p_4 to nozzle exit pressure p_9 . Similar to what have been mentioned in the adiabatic process, the irreversibility due to skin friction and shocks bring the increases of entropy. A similar adiabatic expansion coefficient is defined as:

$$\eta_e = \frac{h_4 - h_9}{h_4 - h_Y} = \frac{C_{pe}(T_4 - T_9)}{C_{pe}(T_4 - T_Y)} \leq 1 \quad (3.6)$$

The process from point 9 to point 0 is a hypothetical heat release process to close the cycle. The nozzle exit temperature T_9 decreases to the ambient temperature T_0 . This process helps to close the cycle although it does not occur inside the engine.

After dividing the thermodynamic cycle into the aforementioned several processes, they could be analyzed one by one.

Compression component (from station 0 to 3)

1. Calculating the stream thrust function Sa_0 at station 0

$$Sa_0 = \frac{p_0 A_0 + \dot{m}_0 U_0}{\dot{m}_0} = \left(\frac{RT_0}{U_0^2} + 1 \right) U_0 \quad (3.7)$$

2. Calculating the inlet exit temperature T_3 by assuming a compression temperature ratio ψ

$$T_3 = \psi T_0 \quad (3.8)$$

3. Calculating the inlet exit velocity or combustor inlet velocity U_3 by using the conservation of energy during the adiabatic compression process

$$U_3 = \sqrt{U_0^2 - 2C_{pc}T_0(\psi - 1)} \quad (3.9)$$

4. Calculating the static pressure ratio p_3/p_0 across the process

$$\frac{p_3}{p_0} = \left[\frac{\psi}{\psi(1 - \eta_c) + \eta_c} \right]^{(c_{pc}/R)} \quad (3.10)$$

5. Calculating the area ratio A_3/A_0 across the process by the conservation of mass

$$\frac{A_3}{A_0} = \psi \cdot \frac{p_0}{p_3} \cdot \frac{V_0}{U_3} \quad (3.11)$$

6. Calculating the stream thrust function Sa_3 at station 3

$$Sa_3 = \left(\frac{RT_3}{U_3^2} + 1 \right) U_3 \quad (3.12)$$

Combustion component (from station 3 to 4)

The combustion processes has been assumed to occur under the constant pressure condition and it is replaced by two processes: the heat addition and mass addition. Some additional parameters need to be described before the calculation.

U_{fx}/U_3 : ratio of fuel injection axial velocity to combustor inlet velocity U_3

U_f/U_3 : ratio of fuel injection total velocity to combustor inlet velocity U_3

$C_f \cdot A_w/A_3$: effective drag coefficient of combustion chamber

h_f : absolute sensible enthalpy of fuel before entering combustor

T° : reference temperature to calculate the absolute enthalpy

1. Calculating the combustor exit velocity U_4 based on the conservation of momentum

$$U_4 = U_3 \left[\frac{1 + f \cdot \frac{U_{fx}}{U_3}}{1 + f} - \frac{C_f \cdot \frac{A_w}{A_3}}{2(1 + f)} \right] \quad (3.13)$$

2. Calculating the combustor exit temperature T_4 based on the conservation of energy

$$T_4 = \frac{T_3}{1+f} \left\{ 1 + \frac{1}{C_{pb}} \left[\eta_b f h_{PR} + f h_f + f C_{pb} T^\circ + \left(1 + f \cdot \frac{U_f^2}{U_3^2} \right) \frac{U_3^2}{2} \right] \right\} - \frac{U_4^2}{2C_{pb}} \quad (3.14)$$

3. Calculating the area ratio A_4/A_3 across the process by the conservation of mass

$$\frac{A_4}{A_3} = (1+f) \cdot \frac{T_4}{T_3} \cdot \frac{U_3}{U_4} \quad (3.15)$$

4. Calculating the stream thrust function Sa_4 at station 4

$$Sa_4 = \left(\frac{RT_4}{U_4^2} + 1 \right) U_4 \quad (3.16)$$

Expansion component (from station 4 to 9)

1. Calculating the nozzle throat area A^* using the isentropic relation

$$A^* = A_4 \frac{U_4}{\sqrt{\gamma_e RT_4}} \left(\frac{\gamma_e + 1}{2} \right)^{\frac{\gamma_e + 1}{2(\gamma_e - 1)}} \left(1 + \frac{\gamma_e - 1}{2} \cdot \frac{U_4^2}{\gamma_e RT_4} \right)^{-\frac{\gamma_e + 1}{2(\gamma_e - 1)}} \quad (3.17)$$

2. Calculating the nozzle exit temperature T_9 across the adiabatic compression process

$$T_9 = T_4 \left\{ 1 - \eta_e \left[1 - \left(\frac{p_9}{p_0} \cdot \frac{p_0}{p_4} \right)^{(R/c_{pe})} \right] \right\} \quad (3.18)$$

3. Calculating the nozzle exit velocity U_9 by using the conservation of energy during the adiabatic compression process

$$U_9 = \sqrt{U_4^2 + 2c_{pe}(T_4 - T_9)} \quad (3.19)$$

4. Calculating the area ratio A_9/A_0 across the process by the conservation of mass

$$\frac{A_9}{A_0} = (1+f) \cdot \frac{p_0}{p_9} \cdot \frac{T_9}{T_0} \cdot \frac{U_0}{U_9} \quad (3.20)$$

For launching and aerodynamic considerations, the nozzle exit diameter D_9 should not be larger than the maximum diameter of the engine (it is the combustor diameter D_4 here).

5. Calculating the stream thrust function Sa_9 at station 9

$$Sa_9 = \left(\frac{RT_9}{U_9^2} + 1 \right) U_9 \quad (3.21)$$

Engine performance evaluation

1. Calculating the specific thrust $F / \dot{m}_0$

$$\frac{F}{\dot{m}_0} = (1+f)Sa_9 - Sa_0 - \frac{RT_0}{U_0} \left(\frac{A_9}{A_0} - 1 \right) \quad (3.22)$$

2. Calculating the specific impulse $F / \dot{m}_f g$

$$\frac{F}{\dot{m}_f g} = \frac{F}{\dot{m}_0 g} \cdot \frac{\dot{m}_0}{\dot{m}_f} = \frac{F}{\dot{m}_0 g} \cdot \frac{1}{f} \quad (3.23)$$

3. Calculating the thrust F

$$F = \frac{F}{\dot{m}_0} \cdot \dot{m}_0 = \frac{F}{\dot{m}_0} \cdot (\rho_0 U_0 A_0) \quad (3.24)$$

4. Calculating the thermal efficiency and overall efficiency

$$\eta_{thermal} = \frac{\text{Engine mechanical power}}{\text{Chemical energy rate}} = \frac{(1+f) \frac{U_e^2}{2} - \frac{U_0^2}{2}}{f h_{PR}} \quad (3.25)$$

$$\eta_{propulsive} = \frac{\text{Thrust power}}{\text{Engine mechanical power}} = \frac{2}{\frac{U_e}{U_0} + 1} \quad (3.26)$$

$$\eta_{overall} = \frac{\text{Thrust power}}{\text{Chemical energy rate}} = \eta_{thermal} \cdot \eta_{propulsive} \quad (3.27)$$

For a flight vehicle, the rule for the specific thrust is the larger the better. The specific impulse of a ramjet using hydrocarbon fuels is around 1500 s. Compared with thrust, the specific thrust and specific impulse are specific properties, which do not depend on the mass of the property. They can thus be used for compare performance of engines with different sizes. The engine with larger specific thrust or specific impulse is more efficient. However, higher specific impulse value defined in terms with the mass of fuel does not directly imply a large thrust (i.e. ion thruster), as the engine structure is not considered in this parameter. The overall efficiency is defined as a ratio of thrust power to energy flow rate from the fuel. Here lists only several parameters that matter most for the study; more performance measurement parameters can be found in Heiser's book [77]. During the design, it is necessary to go back to the stream thrust analysis several times to modify the parameters such as compression temperature ratio, fuel-air ratio and nozzle pressure ratio until the performance parameters satisfy the design requirements.

Finally, the parameters (including the inlet area A_3 and other flow conditions) at station 3, can be obtained from the analysis and they will be used for the following numerical experiments. For the current study (one on-design flight speed), the operating conditions of the combustor are shown in Table 3-2 below.

Table 3-2: Operating conditions of the designed combustor.

Input	22.5 km	unit
T_0	219.2	K
T_3/T_0	4.012	-
$f(C_3H_8)$	0.035	-
P_9/P_0	2.8	-
$\eta_c \eta_b \eta_e$	0.85	-
output		
$F/m_0g(I_{sp})$	427.7(1245)	Ns/kg (s)
$C_F(150\% C_D)$	0.3 (0.3)	-
T_3	902.9	K
P_3	1.46693	bar
M_3	0.3287	-
D_3	0.062	m
$\dot{m}_0(\dot{m}_f)$	0.3448 (0.012)	kg/s
$A_0(D_0)$	0.0049 (0.08)	m^2 (m)
Equivalence ratio ϕ	0.546	-
$\eta_{thermal}$	0.33	
$\eta_{propulsive}$	0.91	
$\eta_{overall}$	0.3	

Noted the relative low thermal and overall efficiencies are mainly due to the low compression, combustion and expansion efficiencies assumed

Uniform pressure and temperature profiles can be assumed at the combustor inlet. In a real ramjet engine, the uniform inlet boundary conditions may not be realistic as the flow has gone through shock waves and the geometric change of subsonic diffusers in supersonic inlets. A thick boundary layer and flow distortions are also always encountered at the exit of ramjet inlets. However, it is common to assume a uniform flow for simplicity in analysis. Moreover, an experimental study using the direct-connect facility to test the combustor is intended to be followed, which means uniform flow conditions have a superiority as they are readily replicated in the planned experiment. The schematic of the designed combustor for numerical computations is shown in Fig. 3-6. It can be seen that a convergent-divergent nozzle is attached at the end of combustor to accelerate the flow to supersonic speeds. After

obtaining the proper inflow conditions, numerical simulations will be performed in the following sections.

Fig. 3-6: Schematic of the designed combustor.

3.2 Classification of Flow Regimes in TVC

The flow field characteristics of a locked vortex after-body have raised the interest of many researchers, because the aerodynamic base drag is reduced when a cavity is attached behind. Four flow regimes have been characterized by Little and Whipkey [20] and Briones et al. [70] through experimental and numerical studies. Fig. 3-7 shows the four regimes when fluid flows over a cavity formed by a forebody and afterbody disk: the wake backflow, the unsteady cavity vortex, the steady cavity vortex and the compressed cavity vortex. They can be easily distinguished from the streamlines and the location of the stagnation point.

The first flow regime is obtained when the after-body disk is close to the fore-body, which causes the formation of a large recirculation zone downstream of the after-body. The recirculation zone spills over the after-body and causes a vortex at the opening of the cavity which in consequence generates another weaker counter-rotating vortex inside the cavity. There is a freestanding stagnation point in the flow (as opposed to be on a wall surface), which indicates the presence of opposed wake backflows [70]. When the after-body is moved further downstream, the second flow regime is encountered where the flow still spills over the after-body but the generated vortex becomes larger and moves into the cavity. The freestanding stagnation point is still in the flow. For both the first and second flow regimes, the vortices generated between the fore-body and the after-body are not fully surrounded by the cavity and are still connected to the downstream recirculation zone, hence they are under the effect of wake flow downstream of the after-body. In the experiments, they are found to be unstable [20].

When the after-body is at a certain distance from the fore-body such that there is no flow spilling over the after-body, the third flow regime takes place which is called the steady vortex regime. In this case, the fluid in the main stream flows smoothly over the cavity and the cavity is filled with a large vortex. The stagnation point in this regime is now on the top surface of the after-body, indicating the cavity vortex is disconnected with the wake and is no longer strongly influenced by the recirculation zone downstream of the after-body. For this reason, the vortex in the cavity is found to be trapped stably inside.

Fig. 3-7: Four typical cavity flow regimes. a) Wake back flow, b) unsteady cavity vortex, c) steady cavity vortex, and d) compressed cavity vortex. Left column: taken from Ref. [70]. Right column: the current simulations.

The last flow regime named as compressed cavity vortex is encountered when the cavity length is larger than that of the steady vortex regime. The cavity vortex is elongated and compressed by the main flow, and the stagnation point is on the front surface of the after-body, which means the flow in the main stream goes directly into the cavity. The direct flow impingement on the front surface of the after-body leads to a large increase in pressure on the surface that contributes to a large drag, which is undesirable for combustor applications. It is also found that the compressed cavity vortex is unstable, because of the strong and continuous exchange of fluid between the cavity and the main flow [20, 70].

For a combustor application where fuel-air mixing is important, the wake backflow, the unsteady cavity vortex, and the compressed cavity vortex can significantly promote fuel-air mixing in the cavity, due to their unsteady nature and the strong flow interactions with the outside region. However, they do not work well in providing a stabilized flame in the cavity, because the vortex is unstable which impairs the stability of combustion, as reported by Hsu et al. [5]. When combustion occurs in the cavity, if the cavity vortex is not stable or may “shed” out the cavity, the flame can shed out with the vortex which leads to an unsuccessful flame stabilization. Though the steady cavity vortex regime has less mass exchange with the mainstream flow in comparison with the others, its stable cavity vortex is perfect for anchoring a pilot flame in the cavity. Therefore, the steady cavity vortex flow regime, which is most appropriate for the fuel-air mixing and flame stabilization in the combustor operation, will be investigated further in the following.

3.3 Grid Sensitivity Analysis

Since the geometry of the designed combustor is axisymmetric, two-dimensional axisymmetric structured grids generated by Gridgen v15.0 are used to study the flow field characteristics. The schematic of the grid is shown in Fig. 3-8. The grid points are clustered densely near the wall to resolve the boundary layers. At the inlet, the first layer thickness in the wall normal direction is scaled to 10^{-6} m to achieve a y^+ value smaller than one. In the main combustor, the grid points are also clustered in the shear layer region to calculate the large gradients.

Fig. 3-8: Schematic of the two dimensional grid.

A grid independence study was performed to ensure the results are not sensitive to the grid resolution. Three different grids were considered: a coarse grid (9,934 cells), a medium grid (40,870 cells), and a fine cells (166,725 cells). The static pressure distribution along the combustor axis and the casing are plotted in Fig. 3-9. It could be seen that the coarse mesh provides a relatively large pressure distribution among the three grids, while the results of the medium mesh are almost identical to the results obtained from the fine mesh. To justify the validity of the axisymmetric assumption for the cavity type flow, a three-

dimensional grid with periodic boundaries representing a $1/8^{\text{th}}$ segment of the combustor is also investigated. Again, the static pressure was chosen as the parameter to compare between the two-dimensional and three-dimensional results, as shown in Fig. 3-10. The results reveal that the axisymmetric assumption here is acceptable, as there is no obvious difference in the static pressure along the axis. Hence, the two-dimensional axisymmetric grid with 40,870 cells was used to study the cold flow field characteristics.

Fig. 3-9: Static pressure comparison between three different 2-D axisymmetric grids: (a) along the combustor axis; (b) along the combustor casing.

Fig. 3-10: Static pressure comparison between 2-D axisymmetric and 3-D periodic simulations: (a) along the combustor axis; (b) along the combustor casing.

3.4 Flow Field Characteristics

Although the optimum cavity size has been reported by Little et al. [20] and Hsu et al. [4], it is only suitable for the low speed regimes (around 40 m/s). For ramjet combustors, whose inflow velocity is very large and has a magnitude around 200 m/s, therefore we need to find the appropriate cavity size that can trap a stable cavity vortex. A good example that helps to understand this issue is that the recirculation zone size changes when the air flows over a backward facing step with different speeds. In the experiments of Little et al. [20] and Hsu et al. [30], the stable vortex is found by varying the cavity dimensions and observing the stability of the resulting vortex. It is easy and straightforward in the experiments; however it is difficult to be carried out in the current numerical simulations. Fortunately, a stable cavity vortex is associated with low drag as indicated in Little and Whipkey's work [20], which provides a way to find the cavity size from the measurable quantity – drag. According to the discussion in the previous section, it is known that the wake back flow and the flow that direct impinges to the after-body surface would make the cavity vortex unstable because of the additional perturbations from the wake and incoming flow. These additional perturbations come from unsmooth streamlines and hence also contribute to additional drag.

A set of steady simulations including different cavity dimensions has been performed. Four different after-body diameters (corresponding to a diameter ratio $D_2/D_1 = 0.7, 0.8, 0.85$ and 0.9 respectively) with a varying cavity length H were investigated. The drag reduction $\Delta C_{d,p}$ which is obtained by comparing the drag to that for the geometry without the after-body and spindle is presented in Fig. 3-11(a). As the skin friction is only a small portion of the drag, it is neglected and only the pressure drag is counted in the calculations. For different after-body diameters, it can be seen that there is a lowest point in each drag reduction coefficient line. Among the four different after-body diameters, the overall lowest drag is obtained at $D_2/D_1 = 0.85$ and $H/D_1 = 0.5$. As indicated by Little et al. [20], this cavity size with minimum drag corresponds to a stable cavity vortex. The streamlines inside the combustor with this cavity size are shown in Fig. 3-11(b). It could be found that the cavity vortex configuration is quite similar to the steady cavity vortex regime mentioned earlier, and the stagnation point locates on the tip of upper surface of the after-body. This cavity size is supposed to be the one that traps a stable vortex inside.

Fig. 3-11: (a) Drag reduction for different cavity sizes; (b) streamlines with optimum cavity size.

3.5 Full Three-Dimensional Unsteady Results

In the previous section, an optimal cavity size that can trap a table vortex inside has been found with the help of a set of two-dimensional axisymmetric steady RANS simulations. Although the vortex structure and the location of the stagnation point show evidence that the cavity vortex is trapped inside, three-dimensional unsteady simulations of the optimal cavity were also performed by using unsteady RANS and LES to confirm whether the cavity vortex is trapped well inside.

Fig. 3-12: Flow vectors of unsteady RANS results.

Unsteady RANS calculations were performed for a fully three-dimensional geometry with around 2.7 million cells. And solutions with time steps of 10^{-6} and 10^{-5} s were obtained. To know whether there are any significant unsteady flow phenomena (i.e. cavity pulsations), an animation of the flow fields near the cavity was generated. The animation results shown that when the cavity vortex structure is established, the flow in the cavity is almost steady except for some small non-axisymmetric eddies in the cavity corner. Fig. 3-12 is a snapshot of the animation, and it shows the instantaneous velocity vectors in the cavity region for a time step of 10^{-5} s. It is observed that the large cavity vortex formed within the cavity is almost steady, which is quite similar to the 2-D steady results. Furthermore, there is no cavity pulsation found and this large cavity vortex is always trapped well in the cavity. The three-dimensional unsteady RANS results confirm that a stable cavity vortex situation has been achieved.

An LES calculation with 5.2 million cells was also performed by using JETCODE. An animation of vorticity contours was generated for the LES results as well. Compared with the unsteady RANS results which have been averaged, the LES solution can provide more unsteady features of the flow. For example, in the instantaneous vorticity contours as shown in Fig. 3-13, it can be seen that vorticity is generated inside the cavity and near the wall. A round green circle with relatively large vorticity magnitude is generated at the upper part of the cavity, which is a vortex core and indicates the cavity vortex is formed. The vorticity generated in the boundary layer upstream of the cavity propagates downstream. Part of it enters the cavity and rotates with the cavity vortex and the other flows directly over the cavity. Furthermore, the animation shows that the cavity is filled with vorticity when the time goes on, but no cavity pulsation is observed. These findings from the LES results also confirm that the vortex generated by the current cavity size is trapped well inside the cavity.

Fig. 3-13: Instantaneous vorticity contours from LES results.

3.6 Summary

The flow field characteristic of the TVC in mini-ramjets has been investigated in this chapter, with the purpose to understand the flow fields, and also to find the proper cavity size which is able to trap a stable cavity vortex for flame stabilization.

A preliminary overall design of the mini-ramjet engine operating at Mach 4 was conducted to obtain the engine flight conditions. The operating conditions of the TVC component were then obtained by using the stream thrust analysis. Although in real ramjets, the combustor inflow is very complex which usually includes shocks, thick boundary layers and non-uniform flows, a simple uniform inflow for the designed combustor was assumed, as a direct-connected ground testing for the designed combustor is planned in the future. The operating conditions of the TVC are summarized in Table 3-3 for convenience. It also should be noted that they are needed to be revised in the spinning combustor, which will be discussed more in Chapter 6.

The numerical results show that a successful TVC design depends on whether a cavity vortex is trapped well for the flame stabilization. Previous literature found proper sizes which can trap a stable vortex in the low-speed flow regimes (~ 40 m/s). This chapter has focused on the appropriate flow speeds suitable for ramjet application (~ 200 m/s). Both the two-dimensional steady RANS calculations and the three-dimensional unsteady RANS and LES simulations have shown that a vortex has been established and trapped well in the combustor cavity, when it has a size of $D_2/D_1 = 0.85$ and $H/D_1 = 0.5$. This cavity size will be used for the following mixing and combustor study.

Table 3-3: Combustor operating conditions.

T_3	902.9	K
P_3	1.46693	bar
Ma_3	0.3287	-
D_3	0.062	m
$\dot{m}_0 (\dot{m}_f - propane)$	0.3448 (0.012)	kg/s
ϕ	0.546	-

Chapter 4

Gaseous Fuel Injection and Mixing

Combustion of Gaseous fuels occurs by exchanging atoms between molecules of reactants through molecular collisions. For a combustion system, if the molecular-level mixing (i.e. micro-mixing) of the fuel and air is not achieved, chemical energy of the fuels cannot be fully released through combustion. The homogeneous mixture at the molecular level is called a micro-mixture, which can be obtained slowly by diffusion. Additional devices that generate macro-mixing through stirring the flow or generating turbulences are commonly used to promote the mixing process, i.e. the use of swirlers, sudden expansions and bluff bodies. These macromixing increases the contact surface of mixtants so that the rate of micro-mixing can be improved. Furthermore, turbulence which is characterized by three-dimensional random eddies can more drastically decrease the segregation scale. Therefore, the rate of mixing can be significantly increased by turbulence.

The small size of the designed combustor (100 mm in diameter) provides limited space and time for the fuel-air mixing process, and the space constraints also indicate utilization of advanced injectors are impossible, as these injectors usually have relatively large sizes. For example, the air-blast type fuel injector usually has a length of around 40 mm, which is almost a half of the designed combustor diameter. Therefore, the simplest round orifice is selected as fuel injectors. Moreover, the current small TVC design as shown in Fig. 3-1 indicates that there is no by-pass air directly injected into the cavity as in Hsu's TVC for gas turbine engines. In other words, in the current study, only the fuel is injected to the combustor through simple round orifice injectors.

The objective of this chapter is to investigate proper fuel injection strategies with simple round orifice injectors. Main interests focus on the cold flow fuel-air mixing performance. The equivalence ratios of the combustor in different fuel delivery schemes are evaluated, especially in the cavity and main combustor region where the cavity pilot flame and main combustor flame locate.

4.1 3-D Grid and Grid Sensitivity Study

As the fuel is injected through small holes which invalidate the axisymmetric of the combustor, three-dimensional computations are required. To ensure the simulation results are not dependent on the grid resolution, a grid sensitivity study for the three-dimensional mesh was performed. To save the computational cost and time, the rotational periodic boundary conditions were applied to a $1/8^{\text{th}}$ segment of the designed combustor, in which the injectors are distributed circumferentially uniformly. All the structured grids used are generated by GRIDGEN v15.

Three different grids without considering the fuel injection are considered first: a coarse grid (255,148 cells), a medium grid (769,494 cells), and a fine grid (1,851,960 cells). The static pressure distributions along the combustor axis and along the combustor casing are plotted in Fig. 4-1. It could be seen that the medium mesh provides almost the same distribution compared with the results of the fine mesh.

Fig. 4-1: Static pressure comparison between three different 3-D axisymmetric grids:
(a) along the combustor axis; (b) along the combustor casing.

For round fuel injectors, there are different ways to generate the mesh for this type of geometry as shown in Fig. 4-2. The first approach known as the four corners approach splits the circle into four elements. In this approach, the grid is easy to generate, but the cells in the four corners are distorted as the corner angle is 180 degree, which means the skewness of the structured grid there is large. Another approach separates the area inside the circle to five small blocks by inserting a square (or a rectangle). The irregularity in this grid is smaller, compared with the four corners approach. Besides, the irregularity will not get worse, no

matter how many cells are used in this approach. Hence, this approach was adopted for the fuel injector in this study.

Fig. 4-2: Grid topology for the round fuel injector: (a) four corners approach and (b) center square approach.

The final grid used for the mixing and combustion study is obtained by combining the medium mesh (769,494 cells without considering the fuel injectors) and the grids for injector generated by the center square approach. There are shear layers between the jet and the main flow when the fuel is injected out from holes. Hence, the grid points were clustered towards the injector wall to resolve the strong gradients. Accordingly, the grids in the upstream and downstream region close to the injector were also refined. A schematic of the final mesh is shown in Fig. 4-3, which has 1.1 million cells.

Fig. 4-3: Schematic of the 3-D periodic grid for the combustor and the grid for the fuel injector.

4.2 Fuel injection Schemes Investigations

The fuel used in the current combustor is gaseous propane (C_3H_8). Several different fuel injection schemes were investigated. It should be noted that again, the fuel injectors in all of the schemes were uniformly distributed in the circumferential direction.

4.2.1 Injecting Locations

Four injection locations were tested as listed in Table 4-1. The first three injecting schemes (Cases 1 - 3) deliver the fuel directly into the cavity at different locations, while the fourth case injects the fuel in the upstream of the cavity. The number of the fuel injectors, injector size, mass flow rate, injection velocity, and momentum flux ratio are provided in Table 4-1 as well. The first three injection schemes have a large momentum flux ratio (~ 40), mainly because the fuel is injected inside the cavity, in which the flow velocity is much smaller than the main flow in the combustor annular passage.

The computational results of the streamlines and the equivalence ratio contours in the cavity are shown in Fig. 4-4. The equivalence ratio of 1.0 is the stoichiometric condition that leads to a complete combustion process in which both fuel and oxidizer are mutually consumed completely. An equivalence ratio greater than 1.0 is a fuel-rich mixture condition where there is fuel left after the reaction. Similarly, the equivalence ratio below 1.0 is a fuel-lean mixture condition in which there is oxidizer left after combustion. The maximum adiabatic flame temperature for a given fuel and oxidizer combination occurs for a stoichiometric mixture. The additional fuel or oxidizer in the fuel-rich or fuel-lean mixture decreases the flame temperature. Hence, a stoichiometric mixture in the cavity is desired for the combustor to obtain a strong pilot flame.

Table 4-1: Injection conditions at different locations.

Injecting location	Number of injectors	Injector diameter [mm]	Mass flow rate per injector [kg/s]	Injection velocity [m/s]	Momentum flux ratio
Case 1: fore-body $r = 0.03$ m	8	3.0	0.001513	78	~ 40
Case 2: after-body $r = 0.03$ m	8	3.0	0.001513	78	~ 40
Case 3: after-body $r = 0.008$ m	8	3.0	0.001513	78	~ 40
Case 4: upstream $z = -0.025$ m	8	2.8	0.001513	92	1.09

Fig. 4-4: Streamlines and equivalence ratio contours in the cavity for different injecting locations:
 (a) case 1, (b) case 2, (c) case 3, and (d) case 4.
 The arrow indicates the fuel injection location and direction.

Fig. 4-4(a) illustrates the fuel injection from the fore-body in case 1. As indicated by the streamlines, the trapped cavity vortex is not enhanced but destroyed by the fuel injection. Some fuel mixes with the incoming flow in a thin layer at the opening of the cavity and then spills out. The other part of the fuel is transported to the bottom of the cavity. When the fuel is injected from the top of the after-body as shown in Fig. 4-4(b), the cavity vortex still exists, but it is squeezed by the fuel stream to be smaller. In case 3, when the fuel is injected from the bottom of the after-body as shown in Fig. 4-4(c), the fuel injection direction follows the initial rotating direction of the cavity vortex. Hence, the previous stable cavity vortex is enhanced by the fuel stream, which reinforces the strength of the cavity vortex. However, it can be seen from the equivalence ratio contours that there is too much fuel remaining in the

cavity (equivalence ratio > 3.0) for the first three schemes. Such a large equivalence ratio indicates a local fuel-rich condition, and combustion cannot occur in the cavity.

In case 4, the fuel is injected to the main flow upstream of the cavity, instead of inside the cavity. Injecting the fuel in the upstream has several advantages. First, the fuel has more time to mix with the air because the injection is “early”; second, when the fuel is injected normal to the main flow where the flow speed is relatively high, the strong shear force and jet interaction can help to promote mixing significantly. For these reasons, rapid fuel-air mixing at the molecular level may be achieved in the designed combustor. As shown in Fig. 4-4(d), the equivalence ratios in the cavity are reduced drastically to around 1.0 compared with the former cases, and the cavity vortex is still trapped in the cavity well.

The equivalence ratio contours for the entire combustor are shown in Fig. 4-5. For cases 1, 2, and 3, the equivalence ratios inside the cavities are too high for combustion, as discussed previously. That means the intention of anchoring a pilot flame (the flame to ignite the mixture and to stabilize the flame) inside the cavity cannot be realized. However, in the main combustor zone, case 3 in Fig. 4-5(c) does provide better equivalence ratios behind the after-body than cases 1 and 2. For example, the stoichiometric ratio (equivalence ratio of 1.0) in case 3 occurs between $z = 0.1 \sim 0.15$, while it starts around $z = 0.2$ in cases 1 and 2. Considering the fuel injection in case 3 enhances the strength of the cavity vortex (see the streamlines in Fig. 4-4 c)), this also increases the fuel residence time inside the cavity, which leads to better mixing performance in the main combustor region. The stoichiometric contour line in case 3 occurs close to the after-body, which indicates that combustion would occur at that location. Hence, the flame is expected to be successfully obtained behind the after-body in case 3, though there would be no pilot flame in the cavity.

For case 4, in which the fuel is injected in the upstream, a good mixing performance is not only provided in the cavity but also in the main combustor region. The upstream injected fuel penetrates into the main stream and mixes with the air. When the mixture flows over the cavity, part of it enters the cavity and the rest flows past the after-body directly to the main combustor region. The equivalence ratios in the cavity have dropped to around 1.0. In the main combustor, the stoichiometric contour line occurs just behind the after-body and the other region has a lean mixture with an equivalence ratio of around 0.5. This indicates that in case 4 a pilot flame can exist in the cavity, and hot temperature zones can also be obtained in the main combustor.

Fig. 4-5: Equivalence ratio contours in the main combustor:
(a) case 1, (b) case 2, (c) case 3, and (d) case 4.

There are several reasons contributing to the good mixing performance of the upstream injection scheme in the combustor. One is because the injection is “early” as mentioned previously. Besides, the contact surface between the fuel and air in the upstream injection is much larger than the other cases. When the fuel is injected directly into the cavity, the contact surface of fuel and air is confined in the shear layer at the opening of the cavity. Therefore, the fuel can only slip out the cavity by following the velocity tangential to the main stream. However, when the fuel is injected perpendicularly to the main flow, it can penetrate into and interact with the main flow, and hence gains much more contact surface with the air. This is essentially a way to promote micro-mixing by manipulated macro-mixing. Besides, the flow structure of the fuel jet normal to the main flow also helps the fuel-air mixing process. A typical flow structure for a jet in cross flow is illustrated in Fig. 4-6. The counter-rotating vortex pair continuously rolls the surrounding air into the core of the jet. And the horseshoe vortex and the wake vortices also can increase the mixing rate. Those are the main reasons for the better mixing of the jet in crossflows.

Fig. 4-6: Flow structures of a jet in crossflow.

Furthermore, the distributions of turbulent kinetic energy (TKE) in the combustor are also provided to gain insights on the turbulence effects on mixing performance. As shown in Fig. 4-7, the distributions of TKE in case 3 (cavity bottom fuel injection) and case 4 (upstream fuel injection) are plotted in cold flows. Compared with the cavity bottom injection scheme, the upstream injection significantly increases TKE in the annular passage from 100 J/kg to over 1000 J/kg. Meanwhile, TKE in the cavity is also increased by 50% from around 200 J/kg to 300 J/kg. In the main combustor, however, the distributions of TKE for both schemes are comparable. These results indicate that the upstream fuel injection scheme provides a better mixing performance than the cavity bottom injection scheme.

Fig. 4-7: Distributions of turbulent kinetic energy: (a) case 3, and (b) case 4.

Previous works on supersonic combustors in scramjets have also noted the benefits of upstream fuel injection. Although the current study involves a subsonic combustor, the observed results in the scramjet works do support similar benefits in ramjet combustors as well. Sun et al. [80] studied the mixing and combustion characteristics of a scramjet combustor with fuel injection upstream of a cavity flame-holder. They found that the interaction of the fuel jet with the shear layer at the opening of the cavity plays an important role in the mixing and flame-holding processes. The well-known “counter-rotating vortex pair” of the jet in crossflow rolls up the shear layer when the jet flows over the cavity, so that the mass exchange between the cavity and the main stream is promoted. If a flame is established in the cavity, the interaction also transports hot products to the jet and the main stream to ignite the unburned mixture. Several other studies [81, 82] have also found that upstream injection can help to suppress cavity oscillations, as it enhances the shear layer growth rate which in turn helps to stabilize the cavity vortex [83].

4.2.2 Upstream Injection: Effects of Fuel/Air Momentum Flux Ratio

It was found that the upstream fuel injection scheme can provide a stoichiometric mixing cavity. However, it does not mean any upstream fuel injection schemes are able to achieve that. During the study, it is found that the fuel/air momentum flux ratio plays a crucial role in distributing the fuel in the cavity and main combustor. The fuel/air momentum flux ratio is defined as follow:

$$q = \frac{\rho_f u_f^2}{\rho_a u_a^2} \quad (4.1)$$

where ρ_f denotes the fuel density, u_f denotes the fuel injecting velocity, ρ_a is the density of incoming air, and u_a is the incoming air velocity. This non-dimensional parameter is a measure of the importance of the fuel stream momentum flux and the perpendicularly flowing main stream momentum flux.

To study the effect of the fuel/air momentum flux ratio on the mixing, four upstream injection schemes with various injection velocities (see Table 4-2) have been investigated. The fuel-air ratio is maintained for the four cases, and the various momentum flux ratio is realized by changing the fuel injection velocity through adjusting the injector diameter. As shown in Table 4-2, the momentum flux ratio increases from 0.52 to 8.85 for case 5 to case 8. It should be noted that the case 4 discussed previously has been renamed as case 7 here, for the convenience of comparison and explanation in this section. In the following context, case 7 will be mentioned instead of case 4.

Table 4-2: Upstream fuel injection with different momentum flux ratios.

Injecting location	Number of injectors	Injector diameter [mm]	Mass flow rate per injector [kg/s]	Injection velocity [m/s]	Momentum flux ratio
Case 5: upstream $z = -0.025$ m	8	3.36	0.001513	63.1	0.52
Case 6: upstream $z = -0.025$ m	8	3.2	0.001513	69.8	0.63
Case 7: upstream $z = -0.025$ m	8	2.8	0.001513	91.6	1.09
Case 8: upstream $z = -0.025$ m	8	1.64	0.001513	260	8.85

The equivalence ratio contours of the cavity in a plane that goes through the fuel injector are shown in Fig. 4-8. As marked at the top of subfigures, the momentum flux ratio increases from $q = 0.52$ in Fig. 4-8(a) to $q = 8.85$ Fig. 4-8(d). It can be seen that the fuel penetrates into the main stream deeper when the momentum flux ratio increases (see the equivalence ratio contours, the high concentration region of the stream is getting closer to the combustor casing). In Fig. 4-8(d), the fuel stream in case 8 directly impinges on the combustor casing when the momentum flux ratio increases to 8.85. However, the equivalence ratio in the cavity decreases when the momentum flux ratio is increased. In Fig. 4-8(a), the lean mixed region (the region enclosed by the line of equivalence ratio of 1.0) is only a smaller portion of the cavity, while it gets larger in Fig. 4-8(b) and even occupies almost the whole cavity in Fig. 4-8(c). For the largest momentum flux ratio in Fig. 4-8(d), the equivalence ratio is lower and much smaller than 0.5.

Fig. 4-8: Equivalence ratio contours for different fuel-air momentum flux ratios:
(a) case 5, (b) case 6, (c) case 7, and (d) case 8.

It can be found that when the fuel/air momentum flux ratio is small, the fuel stream tends to bend itself to accommodate to the main flow and the fuel jet is close to fore-body surface and the cavity. The consequence is that, the fuel is more easily to enter the cavity. On the other hand, the fuel jet with a large momentum flux ratio has a large penetration depth. The fuel stream then flows away from the fore-body and the cavity. In this situation, there would be less chance for the fuel to enter the cavity.

Excess fuel or oxidizer in the fuel-rich or fuel-lean mixtures decreases the flame temperature. A stoichiometric mixture which provides the maximum flame temperature is most desirable for the flame stabilization purpose in the cavity. As shown in Fig. 4-8(a), case 5 most probably has a fuel-rich mixture, because most region of the cavity has an equivalence ratio larger than one. The cavity in case 6 as shown in Fig. 4-8(b) has an

equivalence ratio of around 1.0 (probably a bit fuel-rich), while case 7 has a equivalence ratio of 1.0 (probably a bit fuel-lean) as shown in Fig. 4-8(c). In case 8, there is too little fuel in the cavity, and combustion could not happen.

To have a more throughout understanding of the mixing performance in the cavity for cases 5, 6, and 7, equivalence ratio contours in three addition axial planes in the cavity for each fuel scheme are shown in Fig. 4-9, Fig. 4-10, and Fig. 4-11. Similarly, case 5 in Fig. 4-9 has a fuel-rich mixture especially in the fuel injection plane, case 6 in Fig. 4-10 has an equivalence ratio around 1.0, and the case 7 Fig. 4-11 more likely has a fuel-lean mixture. These equivalence ratios are all within the flammable limits of propane, so that the three fuel injection schemes may all be able to provide a combustion zone in the cavity and a hot source for flame stabilization. Furthermore, it can be known that an upstream scheme with any fuel/air momentum flux ratio between 0.52 and 1.09 (corresponding to case 5 to 7) can provide a similar mixing condition in the cavity.

Fig. 4-9: Equivalence ratio contours of case 5 at axial planes for $q = 0.52$.

Fig. 4-10: Equivalence ratio contours of case 6 at axial planes for $q = 0.63$.

Fig. 4-11: Equivalence ratio contours of case 7 at axial planes for $q = 1.09$.

The equivalence ratio contours in the main combustor are shown in Fig. 4-12. For cases 5 - 7, the main combustor has a stoichiometric mixture just downstream of the after-body, and the equivalence ratio in the large recirculation region behind the after-body has a value slightly smaller than 1.0. For case 8 which has a large momentum flux ratio, the wake region behind the after-body has a very small equivalence ratio, which is smaller than 0.25.

Fig. 4-12: Mixing performance in the main combustor: (a) case 5, (b) case 6, (c) case 7, and (d) case 8.

4.2.3 Effects of Reducing the Fuel Mass Flow Rate in Case 3

As shown previously, if all of the fuel is injected into the cavity, it turns out that combustion cannot happen because there is too much fuel inside the cavity. However, how much fuel is just appropriate to obtain a stoichiometric mixture for the designed cavity, so that a hot pilot cavity flame can be obtained?

It has been found that the injection from the bottom of after-body (case 3) is able to increase the strength of the cavity vortex. Hence, four cases based on case 3 as listed in Table 4-3 are further investigated. The fuel mass flow rate injected into the cavity in the four cases, are reduced to $1/2$, $1/4$, $1/8$, and $1/16$ of the initial fuel mass flow rate. It should be noted that, as the injector size is still based on case 3, the reduction of fuel mass flow rate also means a smaller injection velocity.

Table 4-3: Bottom injection with reduced mass flow rate.

Injecting location	Number of injectors	Injector diameter [mm]	Fuel fraction to case 3	Mass flow rate per injector [kg/s]
Case 9: after-body $r = 0.008$ m	8	3.0	1/2	0.000757
Case10: after-body $r = 0.008$ m	8	3.0	1/4	0.000378
Case11: after-body $r = 0.008$ m	8	3.0	1/8	0.000189
Case12: after-body $r = 0.008$ m	8	3.0	1/16	0.000095

Fig. 4-13: Streamlines and equivalence ratio contours of:
(a) case 9, (b) case 10, (c) case 11, and (d) case 12.

Indicated by the streamlines as shown in Fig. 4-13, the cavity vortex does not affected much by the reduction of fuel injection. When the mass fuel rate of fuel is reduced, the injecting velocity decreases as well. Hence, the fuel jets with large velocities (see the relatively straight streamlines along the jet trajectory in Fig. 4-13(a) and Fig. 4-13(b)) strengthen the vortex, while the fuel jets with smaller velocities (see Fig. 4-13(c) and Fig. 4-13(d)) adapt themselves to the corner vortex and slowly transport the fuel to the whole cavity region. Equivalence ratio contours of these cavities are also shown in Fig. 4-13. When the injected fuel is reduced to half as shown in Fig. 4-13(a), the entire cavity still is filled with a fuel-rich mixture with an equivalence ratio of larger than 3.0. For the case with 1/4 injected fuel in Fig. 4-13 (b), the equivalence ratio in the cavity center is reduced to around 2.0. When the fuel is reduced to 1/8 as shown in Fig. 4-13(c), the equivalence ratio of the mixture is further reduced, but it is still fuel-rich at the bottom of cavity. When the fuel is reduced to 1/16 as shown in Fig. 4-13(d), the cavity has an overall equivalence ratio of around 1.0, and the large equivalence ratios only exist around the fuel injector.

This investigation also provides a way to estimate the entrainment of air from the main flow to the cavity. It is difficult to know the net air entrainment into the cavity, not only because the air is entering and leaving the cavity simultaneously, but also because the interface between the cavity and the main flow is difficult to define. However, if the fuel is injected into the cavity with a small disturbance to the original cavity vortex until a stoichiometric mixture is achieved, the amount of air can then be easily computed from the mass of injected fuel. As shown in Fig. 4-13 (d), it can be seen that an overall stoichiometric mixture is obtained in case 12. The net air entering the cavity can thus be obtained by calculation based on its amount of fuel injected (1/16 of the initial fuel mass flow rate). Moreover, it could be better if the temperature of the cavity can be compared with the adiabatic flame temperature as well (this will be present in the next chapter).

4.3 Summary of the Mixing Schemes

The following fueling schemes have been investigated in this Chapter:

1. 100% fuel injection directly into the cavity (cases 1, 2, and 3)

All of the injections schemes provide an overly fuel-rich cavity (equivalence ratio is much larger than 3.0), which means a pilot cavity flame cannot be obtained. If a stoichiometric mixture in the cavity is desired, the amount of fuel into cavity has to be reduced. Among the three schemes, only case 3, which injected fuel from the bottom of the after-body, is able to provide slightly better mixing in the main combustor. Hence, case 3 will be further investigated by the combustion analysis in the following chapter.

2. 100% fuel injection in the upstream of the cavity (cases 5, 6, 7, and 8)

Injecting upstream of the cavity is found to be an effective way to reduce the amount of fuel in the cavity. In these schemes, fuel streams can penetrate into the main flow, and promote mixing significantly. It is also found that the fuel/air momentum flux ratio (q) is an important parameter that directly controls the amount of fuel into the cavity and into the main combustor. If q is large, the fuel stream is ejected away from the fore-body and less fuel goes into the cavity. If q is small, the fuel stream is close to the surface of the fore-body, and hence has more chances to enter the cavity. For the current combustor operating conditions and the gaseous propane fuel, when q is around 0.6 to 1.09, a near stoichiometric mixture in the cavity and main combustor can be achieved. Case 6 and 7 will be investigated in the combustion analysis.

3. Cavity bottom injection: reduce the fuel mass flow rate (cases 9, 10, 11, and 12)

Injecting directly into the cavity, but reducing the amount of fuel is also a way to achieve stoichiometric mixing in the cavity. It is found that $1/16^{\text{th}}$ of the total fuel can provide a stoichiometric mixture in the cavity for the current TVC. This injection scheme also provides a way to estimate the main flow air entrainment to the cavity. Case 9, 10, 11, and 12 will be investigated in the following chapter to see whether a hot pilot flame can be obtained.

Chapter 5

Combustion Performance

The combustion performance of the designed TVC will be evaluated in two ways: first, to check whether the pilot cavity flame is obtained, as it is crucial to the TVC design; second, to check how good the axial combustion efficiency is. The existence of the pilot flame can be directly observed from temperature contours, while for the combustion efficiency, there are many ways to describe it quantitatively, which will be discussed further in this chapter.

5.1 Definition of Axial Combustion Efficiency

Combustion efficiency is an important parameter that describes the completeness of the chemical reactions in combustors. Depending upon what is concerned most, it is expressed in different forms.

5.1.1 Species-Based Efficiency

When the fraction of fuel that has burnt to completion is concerned, species-based combustion efficiency can be an appropriate measure. It is widely used in situations where the amount of certain species is available to measure (experimentally or numerically). For example, the most simple combustion mechanism for a methane/air mixture is the one-step reaction, which has the following form:

The combustion efficiency can be calculated by tracking how much fuel (methane) is left in the flowing way:

$$\eta = 1 - \frac{\text{unburnt methane}}{\text{initial total methane}} \quad (5.2)$$

or by tracking how much product (CO_2 e.g.) is generated, it can also be expressed as:

$$\eta = \frac{\text{amount of CO}_2 \text{ generated}}{\text{total amount of CO}_2 \text{ generated for complete reaction}} \quad (5.3)$$

The meaning of the species-based combustion definition is straightforward and it is also easy to compute. For the simple one-step combustion mechanism, tracking the unburnt reactant or the generation of product is essentially the same, as the reactants will be directly converted to the final products (no intermediate species) when combustion occurs. However, the real reaction mechanism in an aero-engine is never the simple one-step one. In the multiple-step combustion mechanism, some problems for the computation of the species based efficiency may arise. Because the reactant-based efficiency (Eq. (5.2)) is most probably not the same with the final product-based efficiency (Eq. (5.3)), as many intermediate species (i.e., CO) are involved. Another drawback of the species-based definition is that, it does not provide any information related to the heat release, the most important parameter for the fuel in propulsion devices.

5.1.2 Temperature-Rise Based Efficiency

As the amount of chemical heat release directly contributes to the thrust, another widely-used definition of combustion efficiency is the temperature-rise based combustion efficiency, as expressed in the following equation:

$$\eta(z) = \frac{\tilde{T}(z) - \tilde{T}_{inlet}}{\tilde{T}_{adibatic} - \tilde{T}_{inlet}} \quad (5.4)$$

where $\tilde{T}(z)$ is the mass-averaged static temperature at a certain axial plane, $\tilde{T}_{inlet}$ denotes the mass-averaged static temperature for both fuel and air at combustor inlet, and $\tilde{T}_{adibatic}$ stands for the adiabatic flame temperature. In the definition, the denominator is the maximum temperature rise when all the fuel and air are burnt to chemical equilibrium. Hence, the efficiency at a certain axial plane is the ratio of the static temperature rise to the maximum temperature rise.

It should be noted that several assumptions have been made behind this definition. The first is the change of flow velocity during the combustion process is negligible, so that all of the heat release from chemical reaction only leads to a rise of static enthalpy. The second assumption is that the specific heat at constant pressure for the mixture c_p is a constant during this heat addition process. Therefore, the change of static enthalpy can be expressed as a change of static temperature directly.

5.1.3 Energy-Based Efficiency

For the ramjet combustor in the current study, it would be better to use an energy-based combustion efficiency, which is similar to the temperature-rise based efficiency

mentioned above, but also takes the change of velocity and temperature-dependent c_p into account. The definition in mathematical form is:

$$\eta(z) = \frac{(1+f)h_t(z) - h_{t3}}{f h_{PR}} \quad (5.5)$$

where f is the fuel-air ratio, h_t is the total enthalpy, h_{PR} is the heating value of the fuel. In this definition, the numerator is the total enthalpy change due to combustion, and the denominator is the total energy release from the burning of the fuel. As the total enthalpy includes both the static enthalpy and the flow velocity, it provides a better representation of the energy change of the flow due to combustion. Moreover, a temperature-dependent c_p can be used to compute the static enthalpy, which thus further improves the accuracy of this definition.

5.2 Effects of Injecting Locations and Momentum Flux Ratios

Based on the previous investigation of different mixing schemes, three mixing schemes, cases 3, 6 and 7 associated with the different injection locations (case 3 versus case 6) and different momentum flux ratios (case 6 versus case 7), are chosen for performing the combustion computations. For convenience, the details of the three cases are listed here.

Table 5-1: Three injection schemes from mixing results for combustion simulations.

Injecting location	Number of injectors	Injector diameter [mm]	Mass flow rate per injector [kg/s]	Injection velocity [m/s]	Momentum flux ratio
Case 3: after-body $r = 0.01$ m	8	3.0	0.001513	78	~ 40
Case 6: upstream $z = -0.025$ m	8	3.2	0.001513	69.8	0.63
Case 7: upstream $z = -0.025$ m	8	2.8	0.001513	91.6	1.09

The three cases are chosen for the following reasons. In the mixing studies, case 3 has a fuel-rich cavity but a relatively better mixing in the main combustor. It would be good to confirm through numerical simulations that the cavity flame cannot be obtained in the cavity due to the local fuel-rich condition as discussed previously in the mixing study. For cases 6 and 7 which have a comparable mixing performance in the cavity and in the main combustor, their combustion performance will show which momentum flux ratio is more suitable to provide a better overall combustion performance.

The temperature contours of the three cases are illustrated in Fig. 5-1. In case 3, a main combustor flame exists behind the after-body, but there is no pilot flame inside the cavity (the temperature inside the cavity is only around 700 K). In case 6, an intense main

combustor flame in the main combustor behind the after-body can be observed from the temperature contours. More importantly, a hot flame with a temperature of 2000 K is anchored inside the cavity. This pilot flame plays a crucial role in the design of small-size TVC combustors. It provides continuous heat to the main flow going pass the cavity. The high temperature products and unburned reactants in the cavity are transported out to the downstream to ignite the other unburned mixture. Furthermore, the pilot flame is protected by the cavity instead of being exposed to the main flow which means the ignition source is stable and the combustion is robust. Compared with case 6, case 7 has an even higher cavity flame, whose peak temperature is almost the same as the adiabatic flame temperature (2450 K). The mixing results in the cold flow (see Section 4.2.2.) show that the equivalence ratio in the cavity of case 7 is slightly lower than the stoichiometric. However, such a high temperature cavity flame, which only happens at the stoichiometric mixture, indicates the heat addition from combustion has further promoted the mixing in the cavity.

Fig. 5-1: Temperature contours in the combustor: (a) case 3, (b) case 6, and (c) case 7.

The distributions of TKE in combustor flows of two different fuel injection schemes (cavity bottom injection case 3 and upstream injection case 7) are provided in Fig. 5-2. It can be seen that there is a significant increase of TKE in the annular passage for the upstream fuel injection scheme case 7, and the distribution of TKE in the cavity region is increased by 2 – 3 times. Furthermore, TKE at the center of the recirculation zone in the main combustor is also increased by 30 % from 1400 J/kg to 1800 J/kg. Compared with the TKE distributions in cold flows as shown in Fig. 4-7, the heat release of combustion not only increases the TKE in the annular passage and the cavity, but also has significantly increased

the TKE in the main combustor. The higher TKE indicate that the upstream fuel injection scheme provides a better mixing performance than the cavity bottom injection scheme.

Fig. 5-2: Distributions of turbulent kinetic energy: (a) case 3, and (b) case 7.

Axial combustion efficiency quantitatively characterizes the overall performance of the combustor. The plots of axial energy-based combustion efficiency for the three cases are shown in Fig. 5-3. The horizontal axis is normalized by the combustor inner diameter 0.1 m. It can be seen that the upstream injection scheme (case 6 and case 7) have better efficiency than the cavity-only injection (case 3) over the whole range of the axial distance.

Fig. 5-3: Combustion efficiency.

In case 3, combustion starts from the cavity region, while in the other two cases chemical reactions occurs earlier, right at the fuel injector. It is also not surprising to find that case 3 has the lowest combustion efficiency at the end of combustor, as the fuel mixing surface is much smaller than the other two, in which they are jets in cross flow. For case 7, complete combustion is almost achieved at the end of the combustor. Although case 6 provides a similar mixing performance in the cavity (near stoichiometric but a little fuel-rich in cold flows), it finally only provides a combustion efficiency of around 80%. Considering that case 7 has a larger fuel/momentum flux ratio than case 6, it means that the deeper penetration helps a better mixing in the main combustor. However, it does not mean that an even larger momentum flux ratio would be more helpful for the current TVC, because the further increase of the momentum flux ratio reduces the amount of fuel into the cavity so that the pilot cavity flame may not be obtained.

5.3 Effects of Reduced Fuel Mass Flow Rate in the Cavity

Several reduced cavity injection schemes are selected to confirm whether 1/16 of the total fuel into the cavity can achieve a hot pilot flame. The selected schemes are listed in Table 5-2.

Table 5-2: Reduced cavity injection schemes for the combustion simulations.

Injecting location	Number of injectors	Injector diameter [mm]	Fuel fraction to case 3	Mass flow rate per injector [kg/s]
Case10: after-body $r = 0.008$ m	8	3.0	1/4	0.000378
Case11: after-body $r = 0.008$ m	8	3.0	1/8	0.000189
Case12: after-body $r = 0.008$ m	8	3.0	1/16	0.000095

Fig. 5-4: Temperature contours of the reduced cavity fuel schemes:
(a) case 10, (b) case 11, and (c) case 12.

Temperature contours of the selected schemes are shown in Fig. 5-4. Case 10 with $1/4^{\text{th}}$ of the total fuel still cannot generate a cavity flame due to too much fuel. In case 11 with $1/8^{\text{th}}$ of total fuel, a cavity flame is observed in part of the cavity, but the flame temperature is relatively low (below 2000K). This is because the fuel is still bit much, and the additional fuel decreases the temperature of the cavity flame. For case 12 with $1/16^{\text{th}}$ of the total fuel, a very hot cavity flame is obtained, and its temperature is almost the same as the maximum temperature of the adiabatic flame, 2450K.

In the previous Chapter 4, it was discussed that the amount of fuel into the cavity which provides stoichiometric mixing can be used to estimate the entrainments of air from the main flow. Such a high temperature flame in Fig. 5-4(c) confirms that the stoichiometric mixing is achieved inside the cavity. Besides that, some other important parameters for combustor can also be estimated from this amount fuel which provides adiabatic flame temperature. The first one is the power of the cavity flame, expressed as follows:

$$p = Q \times \dot{m} / 16 = 4.4 \text{ kW} \quad (5.6)$$

The second one is the combustion intensity, defined as:

$$\begin{aligned} \text{combustion intensity} &= \frac{\text{heat release rate}}{\text{combustion volume} \times \text{pressure}} \text{ kW/m}^3 \text{ atm} \\ &\approx 2.1 \times 10^4 \text{ kW/m}^3 \text{ atm} \end{aligned} \quad (5.7)$$

As the obvious differences for various kinds of combustor, it is sometimes inappropriate to directly compare the performance of different systems based on efficiency alone. The combustion intensity is a quantity that measures the combustor performance

regardless of the size difference. According to the definition, it includes the effects of heat release rate, combustor geometric volume, and combustion pressure. Needless to say, the rate of heat release is determined by the amount of fuel burnt. The combustor volume also matters to the intensity, because the larger the combustor volume the easier to achieve good mixing, complete combustion, and high efficiency. The pressure also has a strong influence on the chemical reactions rates (imagining how the reaction rate of combustor increases in a high pressure system, as the chance of effective molecular collisions increases significantly), therefore it is also considered here. Hence the combustion intensity is a good parameter for the designer to know how good the small combustor is, by considering various parameters together. Different from the combustion efficiency (higher is better), combustion intensity is a parameter that should fall in a certain range. Because a larger combustion intensity also means much a higher standard or requirement for the materials, heat transfer, and cooling. Hence, the choice of the combustion intensity for a combustor should be consistent with the state-of-art capability of heat transfers and materials. On the other hand, if the combustion intensity is too low, it may indicate the combustor is poorly designed. For the current design of the small TVC ramjet, the combustion intensity in the cavity has a value of around $2.1 \times 10^4 \text{ kW/m}^3$, which falls in the typical range of aircraft systems, $2 \sim 5 \times 10^4 \text{ kW/m}^3$ [84], which means the combustion intensity of the current combustor is well-designed.

5.4 Angled Injection and Upstream Injection Locations

Angled fuel injection has been extensively studied in the scramjet community [83, 85], as it helps to enhance mixing and reduce total pressure loss by generating a weaker shock, compared with normal injections in supersonic flows. Since the flow in the current combustor is fully subsonic, there are no shock waves, and the total pressure loss due to injection is relative small. However, the injection angle may also affect the combustor overall performance, because the fuel stream trajectory changes with different angles.

Three injection angles (90° , 60° , and 30°) have been investigated based on case 7, in which the fuel is injected perpendicularly to the main flow (90° injection). The other two injection angles incline respectively at 60° and 30° to the main flow direction. Meanwhile, the momentum flux ratio for the gaseous propane fuel, 1.09, is maintained for the three cases.

As shown in Fig. 5-5(a), the results reveal that, for the maintained momentum flux ratio, injection at 90° yields higher combustion efficiency than the other two. However, only small differences exist at the combustor exit. When the injection angle becomes smaller, the fuel penetration depth decreases so that the fuel stream is closer to the center-body. Similar

to the momentum flux ratio discussed previously, a small injection angle also means that the fuel enters the cavity more easily, which leads to a local fuel-rich cavity. When the fuel goes further downstream, turbulent diffusion gradually balances the effects of angled injection. However, it should be noted that the best injection angle of 90° observed here may not apply to other TVC designs with a different fuel-air momentum flux ratio, as the current analysis is based on the optimum momentum flux of 1.09. When the fuel-air momentum flux ratio is different, e.g., in the case of a choked fuel nozzle, which has a large fuel injection velocity, the best angle will most probably not be 90° . In that situation, fuel injection schemes with different angles should be conducted to seek the optimum angle providing the best combustion performance.

Fig. 5-5: Combustion efficiency of injection schemes with (a) different injection angles, (b) different upstream injection locations

The fuel in case 7 is injected at $z = -0.025$ m. To gain insights on the effect of upstream injection location, another two locations at $z = -0.04$ m and $z = -0.01$ m with the same momentum flux ratio were studied. Fig. 5-5(b) displays the resulting combustion efficiency at the different injection locations. The combustion efficiency lines start from different locations, as the injectors have been shifted upstream and downstream. When the fuel is injected “early”, it has more time to mix and hence combustion occurs early. However, there is almost no difference in combustion efficiency when the flow approaches the combustor exit. The change in fuel residence time by varying the injection location is just around 0.07 ms, which is very small. Compared with the combustor length, the small variations of injector location here almost do not affect the final overall combustor performance. The fuel injectors are designed to be located at the exit of the engine inlet or at

the combustor entrance. This region is within a short length for the mini-engine considered here. However, things will be different if the fuel is injected in the front part of the engine inlet, e.g., injecting fuel at the inlet cone. In that situation, a premixed mixture might be assumed as the incoming flow for the combustor, which is outside the scope of this study.

5.5 Effect of Combustor Length

Thrust-weight ratio is an important performance parameter for engine designers. The larger the thrust-weight ratio, the better is the engine. There are two ways to improve this parameter, one is to increase the thrust when the weight is fixed, and the other is to reduce the weight meanwhile maintaining the thrust. Cutting the engine length is a common way to reduce the engine weight, and to improve the thrust-weight ratio.

The initial combustor length for the designed combustor here is set at 0.25 m ($z/D = 2.5$). If the combustor length is too short, chemical energy of the fuel may not be released completely. If the combustor length is too long, the fuel has burnt completely before exiting the combustor, which indicates the long combustor length is wasted. To examine the impact of combustor length on the current TVC design, computations for another two combustor lengths ($z/D = 3.0$, and 3.5) based on case 7 have been carried out.

The resulting combustion efficiencies are presented in Fig. 5-6. The figure shows that there is little change when the combustor length is increased. For $z/D = 2.5$, the efficiency at the combustor exit is around 93%. When the length is increased to three times of the combustor diameter, the efficiency finally reaches around 100%. For $z/D = 3.5$, the combustion efficiency is almost overlapped with that of $z/D = 3.0$ in the front of the combustor, but it decreases slightly at the rear.

Fig. 5-6: Combustion efficiencies with different combustor lengths.

The streamlines inside the combustor with different lengths are also provided in Fig. 5-7. It can be seen that the large recirculation zone behind the after-body does not change much with the different combustor lengths, which indicates there is no forced circulation due to the confinement of the nozzle (not shown in Fig. 5-7). Hence, the interaction effect between the nozzle and the combustor is small.

Fig. 5-7: Streamlines for combustor with different length:
(a) $z/D = 2.5$, (b) $z/D = 3.0$, and (c) $z/D = 3.5$.

5.6 Summary

Combustion simulations for the selected injection schemes from the previous chapter have been performed. The numerical results show that:

1. Upstream injection with a momentum flux ratio of around 1.09 (case 7) for the current TVC with propane fuel has a good combustion performance. With this momentum flux ratio, a hot pilot flame in the cavity and a strong main combustor flame can be obtained.

2. If injecting the fuel directly into the cavity, $1/16^{\text{th}}$ of the total fuel mass flow rate can provide a stoichiometric mixture and a hot cavity flame. This is also an indirect way to estimate the air entrainment from the main stream into the cavity.

3. Angled injection shows that normal fuel injection (90 degree) is best for the present configurations. Upstream fuel injection locations have little effect on the final combustion efficiency.

4. Two additional combustor lengths investigated show that they have little effect on the combustion efficiency, but longer combustor reduced the engine thrust-weight ratio.

Chapter 6

Effect of Spinning Motion

6.1 Introduction

Under some circumstances, the combustor may be required to operate at high spinning speed. For example, it could be a combustor installed in a spin-stabilized gun-launched projectile [86], in which all of the mechanical parts are rotating together at a spinning speed rate around 10,000 rpm to 30,000 rpm. In this situation, the strong spinning motion brings centrifugal and Coriolis forces, which may have a profound effect on the cavity flow field, fuel-air mixing, and combustion performance. Hence, there is a crucial need to study the effects of high spinning motion on the designed combustor.

Unlike the fin-stabilized projectile whose flight stability is ensured by the control moments generated by fins, the spin-stabilized projectile requires a high spinning motion to obtain gyroscopic stability. For example, the 155 mm M549 projectile has a spinning rate of around 13,000 rpm when it flies at Mach 2.0 [87]. A non-dimensional spinning rate which is defined as PD/u , is more commonly used for the spinning projectiles. P is the spinning rate (rad/s), D is the diameter, and u is the free stream velocity. For the 155 mm round, it has a value of 0.314. If the same non-dimensional spinning rate is adopted for the current designed 100 mm combustor for Mach 4.0, the absolute spinning rate should be around 35,580 rpm. For convenience, a 30,000 rpm spinning rate is chosen for the study in this Chapter.

6.2 Literature Review

To our knowledge, the TVC under spinning motion has not received much attention so far. Only several studies associated with swirling flows, rotating cavity and effects of spinning on combustion can be found in low speed applications.

In gas turbines engines, it is common to supply cooling air to the compressor and turbine blades. The cooling air flows over cavities that are formed by the compressor or turbine disks. Farthing et al. [88, 89] experimentally studied the flow structure and heat transfer in rotating cavities with axial cooling through-flow. It was found that vortex

breakdown occurred inside the cavities. The vortex breakdown phenomenon is characterized by a bubble-like recirculation structure with an internal stagnation point and reversed flow with swirling, which is commonly encountered in gas turbine combustors. The breakdown vortex structure is nearly axisymmetric along the rotating axis and is located immediately downstream of the sudden opening. Unlike its non-swirling counterpart in which the recirculating flow is in the forward/reverse directions, the vortex breakdown bubble also has a strong swirling component. Lewis et al. [90, 91] experimentally studied the combustion in a centrifugal-force field. Their experimental results showed that the rate of combustion can be significantly increased by the centrifugal forces. Katta et al. [92] numerically studied the centrifugal force effects on flame structure in an ultra-compact combustor (UCC). In the UCC, a cavity is located circumferentially around the combustor. The cavity, together with swirling flows introduced inside the cavity, is used to improve the mixing and combustion efficiency. Their simulation results showed that a thinner and shorter flame was obtained in the UCC, but the flame structure was not affected significantly.

In flows which are rotating or moving in a curved path, secondary flows are commonly encountered. The secondary flow is a relatively minor flow, compared with the primary flow or the main flow. Although it is not the dominant one in the whole flow field, its effects are sometimes quite obvious and important. A typical secondary flow occurs in a teacup experiment [93]. The tea leaves have a larger density than the liquid, so they are at the bottom of the teacup. If using a spoon to stir the tea to make the tea leaves and the liquid spin around the cup together, a forced vortex is formed. Someone may expect that the tea leaves move outwards to the perimeter because of the centrifugal force induced by the swirling flow. However, the real phenomenon is that the tea leaves gather at the center of the bottom, as shown in Fig. 6-1 (a). This is the well-known tea leaves paradox.

Fig. 6-1: The secondary flow in a teacup stirred to spin [94].

Albert Einstein firstly solved this paradox in 1926. When the tea is moving in the circular path, a centripetal force towards the center is needed to maintain the circular motion. It is the radial pressure gradient that provides the centripetal force. In mathematical form, it can be expressed as:

$$-\frac{dp}{dr} = \rho \frac{u_\theta^2}{r} \quad (6.1)$$

where u_θ is the tangential velocity. At the bottom in the boundary layer, the velocity of the fluid is slowed by the viscosity, while the pressure gradient is still dominated by the primary flow, which is the spinning motion. Once the tangential velocity decreases, the centripetal force needed ($\rho \frac{u_\theta^2}{r}$) is smaller than the pressure gradient provided. Hence, the radius of the moving path has to be decreased to reach a new balance of centripetal force and the pressure gradient. Thus, the fluid at the bottom essentially moves inwards to the center. When the fluid reaches the center, it has no way but goes up. A circulation in the vertical plane is thus created, as shown in Fig. 6-1 (b). The tea leaves at the bottom are under the effect of the boundary layer, so they also move to the cup center. However, they are denser than the liquid and cannot go up with the fluid, so the final phenomenon is they are gathering together at the center of the cup. In this experiment, the vertical circulation as indicated in Fig. 6-1 is the secondary flow.

In aerospace engineering, secondary flows are also common in the flows that go through gas turbine blades. As shown in Fig. 6-2, the blades form many bends which turn the main flow direction. Similarly, viscous effects due to the bottom wall will also generate a secondary flow along the direction indicated by the arrows in Fig. 6-2.

Fig. 6-2: The secondary flows in gas turbine blades [95].

6.3 Numerical Approach

6.3.1 Flow Solvers and Numerical Models

As mentioned at the end of Chapter 2, both RANS and LES were used to investigate the spinning combustor. For convenience, the table summarizing the numerical flow solvers and models is reproduced as shown in Table 6-1. For the cold flow field characteristics without fuel injection and combustion, both the LES JETCODE and the FLUENT were applied. However, the LES code cannot handle the non-axisymmetric geometry when the fuel injectors exist in fuel injection and combustion. Hence the FLUENT with the Reynolds Stress turbulence model was used to study the fuel mixing and combustion.

Table 6-1: Numerical flow solvers and models used in the spinning combustor.

		Flow solver	Turbulence model	Combustion model	Schmidt number
Spinning combustor	Flow field characteristics	FLUENT & JETCODE	RSM & Smagorinsky	-	-
	Fuel mixing	FLUENT	RSM	-	0.7
	Combustion			ED	

In the previous non-spinning combustor, the chemical equilibrium (CE) combustion model was used for the numerical simulations. It provides more realistic and accurate results, because more intermediate species are introduced. However, it is not suitable for the prediction in spinning combustors. The reason is that the chemical equilibrium model assumes a constant pressure in the combustion chamber to compute the temperature and products based on thermodynamic relations. In the non-spinning combustor, the pressure in the combustor does not change much, and thus it can be assumed as constant-pressure combustion. In the spinning combustor, however, the pressure field is related to the spinning motion, which is not uniform (see Eq. (6.1)). Hence, the eddy dissipation (ED) combustion model, which is not dependent on the pressure, has been selected to perform the combustion simulations for the spinning combustor.

6.3.2 Computational Domain and Boundary Conditions

Multi-block structured grids for three-dimensional simulations were generated by GRIDGEN v15 for the spinning combustor. The grid topologies and the rotational periodic boundaries are shown in Fig. 6-3. To save computational time, only a $1/8^{\text{th}}$ segment combustor with rotational periodic boundary conditions is considered. In the wall normal

direction, the first layer thickness is scaled to 10^{-6} m in order to achieve a y^+ value smaller than one. To ensure the calculated results are not dependent on the grid, a grid independent study was conducted. Three grid sizes were examined: coarse, medium, and fine, with 225,148, 769,494, and 1,893,240 cells, respectively. The temperature profiles along the radial direction in the cavity region were selected for comparison. It was found that the maximum temperature difference between the medium and the fine grids along the radius was no larger than 1.5%. Furthermore, the static pressure along the combustor centerline was also compared, and the maximum pressure difference is not larger than 1%. The number of cells of the grid used in this work is around 0.8 million with 45 points in the circumferential direction of the $1/8^{\text{th}}$ segment combustor.

Fig. 6-3: Grid topology of the $1/8^{\text{th}}$ segment combustor.

6.3.3 Discussion about the Combustor Inflow Conditions in Spinning Motion

For the non-spinning combustor in the previous chapters, uniform pressure and temperature profiles were assumed at the combustor inlet. In the designed TVC, struts are used to connect the center body to the combustor casing as shown in Fig. 6-4, so that the center body can be placed at the right location. When the engine spins, the combustor inflow conditions could be quite complex due to the flow/wall and flow/strut interactions in the supersonic inlet. Therefore a careful consideration of the inflow for the spinning combustor is needed before performing the numerical simulations.

Fig. 6-4: Schematic of the struts in the design ramjet.

When the ramjet engine spins, the mass flow rate of air going through the engine is the same as the non-spinning one, because the capture area of the inlet does not change. However, the velocity, pressure and temperature distributions at the combustor inlet would not be the same. The incoming flow tends to rotate with the solid walls of the engine due to viscous effects. Furthermore, the struts in the designed engine will also rotate at the same speed with the whole engine. In the non-spinning motion, the struts are usually neglected for simplicity because of their relatively insignificant effects to the flow [96]. However, when the spinning rate is high (i.e. 30,000 rpm), the struts may act like compressor blades in the gas turbine, which do work on the incoming flow and force it to spin. As a result, the inflow at the combustor inlet in the spinning combustor must have a tangential velocity component. Also, due to the high spinning rate, the flow may quickly achieve an equilibrium state, in which the fluid is rotating together with the struts without further tendency to change. In other words, the incoming flow would rotate with the engine at the same spinning rate. In this study, the engine spins in a solid body rotation at 30,000 rpm. Hence, the same spinning rate is assumed for the flow at the combustor inlet for the aforementioned reasons.

When the combustor inflow is assumed to be in a solid body rotation at 30,000 rpm, the inlet velocity profiles can be obtained, i.e. the tangential velocity in solid body rotation is proportional to the radius which is a non-uniform profile. Furthermore, we could also have some understanding on the pressure and temperature profiles of the inflow due to the existence of tangential velocity. For example, there must be a pressure gradient along the radial direction that keeps the fluids moving tangentially, which means the inlet pressure profile should be non-uniform as well. The inlet total temperature profile also becomes radius-dependent because of the non-uniform velocity profile. In addition, the other properties (e.g., density) have to adjust themselves to satisfy the ideal gas equation. These coupled non-uniform properties provide many difficulties for the specification of the inflow boundary in numerical computations.

A solution to solve this problem is to set the inflow boundary conditions in a relative reference frame, which spins at the same rate (30,000 rpm) with the engine. In this rotating reference frame, the non-uniform tangential velocity component would vanish. The relative total temperature (total temperature in the relative reference frame) is thus not radius-dependent, and it becomes the same as the total temperature in the previous non-spinning combustor. The mass flow rate of incoming air, as discussed previously, remains the same whether spinning is considered or not, so it can still be implemented in the rotating reference frame. Therefore, the reference frame spinning rate (30,000 rpm), inflow mass flow rate in the relative reference frame and the relative total temperature in the relative reference frame will be directly set at the combustor inlet in the flow solver. The other flow properties (e.g.

density, pressure and velocity) will then be computed based on them. Another advantage of using the relative reference frame is that the fuel injection conditions, the relative fuel total temperature and fuel mass flow rate obtained previously can be directly implemented in the spinning reference frame.

6.3.4 Treatments of Numerical Boundary Conditions

For ANSYS FLUENT

As discussed previously, setting the inflow boundary conditions in the spinning reference frame is able to avoiding the strong coupling between the inflow parameters. In FLUENT there is an option available to specify the motion of reference frame, so it is straightforward to implement these inflows. The assigned inlet profiles in FLUENT are provided here to check whether the inflow conditions of interest have been obtained. As shown in Fig. 6-5(a), it can be seen that the radial velocity is almost zero along the radius, while the tangential velocity increases linearly along the radius which is the solid body rotation, and the axial velocity has small variations. The pressure profiles plotted in Fig. 6-5(b) shows that it increases quadratically with the radius, which also obey the pressure equations (6.1). Hence, the inlet boundary profiles of interest have been obtained.

Fig. 6-5: Specified inlet profiles in FLUENT: (a) velocity profiles, and (b) pressure profiles.

For JETCODE

JETCODE only deals with incompressible flow, which means the density in the calculations will be treated as a constant. For low speed of flows with a Mach number smaller than 0.3 (in the current combustor, the flow Mach number is around 0.3), the

incompressible flow assumption can still be made for the combustor without fuel injection and combustion.

Practical difficulties have been encountered in implementing inflow conditions in the spinning reference frame in the LES code. Hence, modification of the inflow conditions has been made as follows: the inflow is in solid-body rotation while the wall is still stationary. Although this modification makes the boundary conditions not exact as what is desired initially (both the wall and flow are rotating together), but it is also beneficial to gain insights on the flow structure of swirling flows over the cavity.

The boundary conditions for the mean quantities specified at the inlet of LES simulation are listed as follows:

$$\text{Radial velocity equation: } u_r = 0 \quad (6.2)$$

$$\text{Tangential velocity equation: } u_\theta = \Omega r \quad (6.3)$$

$$\text{Axial velocity equation: } u_z = \text{uniform} \quad (6.4)$$

6.4 Results and Discussion

6.4.1 Cold Flow Field Characteristics Predicted by LES

LES results of the designed combustor without and with spinning have been obtained by the JETCODE. In the calculations, there are no fuel injections and combustion. Hence, the LES calculations only focus on the cold flow field characteristics of the combustor in spinning motion. The vorticity, streamlines, and the vortical structures in the cavity were investigated.

Fig. 6-6: Vorticity contours of the combustor: (a) 0 rpm, (b) 30,000rpm.

Fig. 6-6 illustrates the vorticity magnitude contours in the cavity of the combustor. The vorticity contours in the non-spinning combustor are shown in Fig. 6-6(a). In the near wall regions, the magnitude of the vorticity is relatively high due to the viscous effects. In the front tip of the cavity, the vorticity generated in the boundary layer of the surface of the fore-body is transported downstream to the cavity. A vortex core marked by a red circle as shown in Fig. 6-6 (a) can be seen clearly. When the combustor spins at 30,000 rpm as shown in Fig. 6-6(b), there is no obvious vortex core, but the overall level of vorticity magnitude is significantly larger than that of the non-spinning combustor.

The time-averaged results of LES were also obtained. Fig. 6-7 shows the two-dimensional streamlines in the combustor. The velocity is normalized by a reference velocity $U_{\text{reference}}$, which is the average velocity magnitude at the combustor inlet. The streamlines for the non-spinning combustor is plotted in Fig. 6-7(a). It could be seen that a well-trapped vortex similar to the one obtained from the previous RANS results (in Chapter 3), has been obtained in the cavity. For comparison, the streamlines in the combustor with 30,000 rpm are shown in Fig. 6-7(b). One of the obvious differences comes from the cavity vortical structure. The cavity vortex in the spinning combustor has split into two vortices: one at the opening of the cavity, and the other at the bottom of the cavity. Another obvious feature of the spinning combustor is the very short recirculation zone behind the after-body. As indicated by the horizontal axis, the recirculation zone has been reduced to a half of its initial length in the non-spinning combustor.

(a)

(b)

Fig. 6-7: Two-dimensional streamlines for the combustor: (a) 0 rpm, (b) 30,000rpm.

The three-dimensional streamlines viewed in the relative reference frame, which can clearly show how the flow moves in the combustor in non-spinning and spinning motions, are also obtained. As shown in Fig. 6-8, the spinning combustor has more obvious three-dimensional flow features than the non-spinning combustor, especially in the cavity region. From the point of view of mixing, a strong three-dimensional flow could promote fuel-air mixing. It can also be seen that the flow close to the combustor axis is still rotating in Fig. 6-8(b), even viewed in the relative reference frame. This means that the spinning rate of the flow near the combustor axis is larger than 30,000rpm.

(a)

(b)

Fig. 6-8: Three-dimensional streamlines of the combustor viewed in the relative reference frame:
LES averaged results: (a) 0 rpm, and (b) 30,000 rpm.

6.4.2 RANS Results

As the JECTCODE only studies the cold flow field characteristics of a swirling flow, ANSYS FLUENT 14.5 has been used to study the spinning TVC on the flow field characteristics, mixing and combustion in the combustor.

6.4.2.1 Flow Field Characteristics

Two-dimensional streamlines with pressure contours in cold flows are plotted in the plane cutting through the fuel injector for both the non-spinning and spinning TVC. In the non-spinning motion as shown in Fig. 6-9(a), a well-trapped vortex can be observed inside the cavity and a large recirculation zone behind the after-body is established, and the pressure in the combustor is almost uniform. However, quite different flow patterns and pressure distributions can be found in the combustor spinning at 30,000 rpm as shown in Fig. 6-9(b). First, the single vortex in the cavity is destroyed and split into several vortices. Secondly, the recirculation zone size behind the after-body is reduced significantly: in the non-spinning TVC the recirculation zone is formed between the after-body and $z/D = 2.0$, while it appears between the after-body and $z/D = 0.7$ for the spinning case.

(a)

(b)

Fig. 6-9: 2-D streamlines with pressure contours in cold flows: (a) 0 rpm, and (b) 30,000 rpm.

Two-dimensional streamlines in combustor flows are also provided. In the non-spinning motion as shown in Fig. 6-10(a), the single cavity vortex and the recirculation zone behind the after-body can still be observed. Compared with the cold flows, the length of the recirculation zone is reduced, as its stagnation point is moved to $z/D = 1.5$. In the spinning combustor as shown in Fig. 6-10(b), multiple vortices are established in the cavity: one at the opening of the cavity and the others are inside the cavity and at the bottom of cavity. Similar vortex breakdown structures can also be observed in the wake region when combustion occurs. Furthermore, compared with the cold flow, the recirculation zone size behind the after-body is slightly increased, as it is located between the after-body and $z/D = 1.2$.

(a)

Fig. 6-10: Two-dimensional streamlines in combustor flows: (a) 0 rpm, and (b) 30,000 rpm.

Three-dimensional streamlines in combustor flows viewed in the relative reference frame are plotted in Fig. 6-11. Compared with the nearly two-dimensional flow in the non-spinning combustor as shown in Fig. 6-10(a), significant three-dimensional flow features are observed in the cavity and in the wake region when there is spinning as shown in Fig. 6-11(b). Moreover, it can be seen from Fig. 6-11(b) that the flow near the combustor centerline (see the region around $z = 0.12 - 0.14$ m) still rotates, even viewing in the rotating reference frame (same as the LES results presented previously). As the rotating reference frame spins at 30,000 rpm in a direction indicated by the large arrow in Fig. 6-11(b), the rotating streamlines there means the flow rotates at an even higher spinning rate.

Fig. 6-11: 3-D streamlines in combustor flows: (a) 0 rpm, and (b) 30,000 rpm.
(Viewed in the relative reference frame and the large arrow denotes the reference frame rotating direction)

The observed “multiple vortices” in the cavity as shown in Fig. 6-10(b) could be regarded as the combined results of two motions, one is the cavity vortex driven by the main

axial flow, and the other is the rotation caused by the Coriolis force $-2\vec{\Omega} \times \vec{u}_{rel}$. Imagine a small fluid element of the main stream entering the cavity, in the stationary case it follows the path as shown in Fig. 6-11(a), from point 1 $\rightarrow$ 2 $\rightarrow$ 3 $\rightarrow$ 4 $\rightarrow$ 1. In the spinning combustor as shown in Fig. 6-11(b), a radial velocity pointing to the centerline exists when the particle enters the cavity. Due to the radial velocity component, the Coriolis force tends to make the fluid element rotate so that a tangential velocity component is generated even in the relative reference frame. The existence of the tangential velocity leads to a strong three-dimensional flow inside the cavity, so that the fluid element cannot follow the path as in the non-stationary combustor. For better visualization, one single streamline entering the cavity from the main flow is plotted in Fig. 6-12. The three-dimensional flow effects generated by the Coriolis force can be seen clearly in Fig. 6-12(a). Fig. 6-12(b) shows the section view from A-A, and it can be seen that the selected streamline only covers the opening of the cavity which corresponds to the outer cavity vortex as shown in Fig. 6-10(b). Hence, the entrainment from the main flow forms the opening vortex, and the other vortices as shown in Fig. 6-10(b) are thus weaker vortices driven by the main vortex at the cavity opening.

Fig. 6-12: Single 3-D streamline in combustor flow for the spinning TVC: (a) bird's eye view of the selected streamline, and (b) section view of the selected streamline (viewed in the relative reference frame).

The centrifugal force, $-\vec{\Omega} \times (\vec{\Omega} \times \vec{r})$, which is always pointing outward along the radial direction, is a force that tends to draw the subject away from the rotation axis. For a steady spinning motion with a constant spinning rate, it satisfies the simple radial equilibrium equation as follows:

$$-\vec{\Omega} \times (\vec{\Omega} \times \vec{r}) = \nabla (\Omega^2 r^2 / 2) = \Omega^2 r = -\frac{1}{\rho} \frac{\partial p}{\partial r} \quad (6.5)$$

This equation indicates that the centrifugal force in steady rotation is balanced by the pressure field. In other words, a radial pressure gradient is induced by the centrifugal force. Because this force in steady rotation is a function of relative position only, it is thus usually combined with the gravity or pressure term in the momentum equation.

The small recirculation zones downstream of the after-body as shown in Fig. 6-9(b) and Fig. 6-10(b) are essentially due to the significant change in the pressure field in order to balance the centrifugal force. In the non-spinning combustor, the recirculation zone is a consequence of the total pressure loss due to flow separation at the sudden expansion, which induces a flow reversal at the after-body. In contrast, for the spinning combustor, as the flow expands into a larger area downstream of the after-body, the fluid that enters into a smaller radius must spin at a faster rate in order to conserve angular momentum. The higher velocity then leads to a decrease in static pressure if viscous losses can be neglected. In essence, the higher angular velocity (i.e., centrifugal force) in the vicinity of the after-body must be balanced by a pressure gradient as described by Eq. (6.5). The lower pressure in the vicinity of the after-body thus draws in downstream fluid and reduces the size of the recirculation zone.

The non-dimensional form of the Navier-Stokes equation in the rotating reference frame can be expressed as follows (though the flow in the current TVC is compressible, the incompressible form is adopted for easier analysis):

$$\frac{\partial \vec{u}_{rel}^*}{\partial t^*} + \left[\frac{U_{rel}}{\Omega L} \right] (\vec{u}_{rel}^* \cdot \nabla^*) \vec{u}_{rel}^* = -\nabla^* p_{red}^* + \left[\frac{\nu}{\Omega L^2} \right] \nabla^{*2} \vec{u}_{rel}^* - 2^* \vec{k} \times \vec{u}_{rel}^* \quad (6.6)$$

where $\vec{u}_{rel}$ is the relative velocity vector, $\vec{\Omega}$ denotes the angular velocity vector, $\vec{k}$ is the unit vector in the direction of rotation, ν is the kinematic viscosity, U_{rel} is the reference velocity, and L is the reference length scale. The superscript * represents dimensionless quantities. Because the centrifugal force is always pointing along the radial direction and is related to the pressure gradient as expressed in Eq. (6-5), it thus has been combined with the pressure gradient to form the reduced pressure p_{red} through:

$$-\frac{1}{\rho} \nabla p - \vec{\Omega} \times (\vec{\Omega} \times \vec{r}) = -\frac{1}{\rho} \nabla p_{red} \quad (6.7)$$

The term $U_{rel}/\Omega L$ in Eq. (6-6) is known as the Rossby number Ro , which measures the importance of inertial force and the Coriolis force. A large Ro means the inertial force is more important, whereas a very small Ro value indicates the Coriolis force dominates the flow. Similarly, $\nu/\Omega L^2$ which is known as the Ekman number Ek , is a measure of the ratio of the viscous to Coriolis force. For the flow inside the cavity, if the mean velocity and the

cavity depth are selected as the reference velocity and reference length, the Rossby and Ekman numbers are around 0.3 and 1×10^{-5} respectively. The Rossby number tells that the Coriolis force is more important in the cavity. Therefore it is the main cause of the multiple-vortex structure in the cavity. Because the Ekman number is very small, it indicates that viscous forces are relatively unimportant for the main flow and hence the viscous effects outside the boundary layers and the shear layers may be neglected.

When combustion occurs, the heat release may change the flow through density and viscosity variations. Revisiting the Ekman number with a larger viscosity for hot flow, i.e. at 1800 K, it provides a value of around 7×10^{-5} . Though Ek has been increased due to the high temperature, it is of the same order as that in cold flow and is still a small number. When Ek is very small, it means that the Coriolis force overwhelms the viscous force. In the current spinning combustor, the Coriolis force is thus 4 to 5 orders higher than the (laminar) viscous force. And since the turbulent viscosity in combustion is typically 2 to 3 orders of magnitude larger than the laminar viscosity, Coriolis effects should still be larger than turbulence effects, except in the boundary and shear layers. Therefore the small Ekman number suggests that the importance of the turbulence may be overshadowed by Coriolis effects, since turbulence is a viscous phenomenon.

But a more explicit way to see the relative importance of turbulence and the Coriolis force effects is to use another non-dimensional, the eddy diffusivity number K (also called exchange coefficient) [97]. It is a dimensionless number that quantifies turbulent diffusion rate. The ratio of eddy diffusivity coefficient to the laminar kinematic viscosity is equal to Reynold number ($K/\nu = Re$). With the help of this relation, a new non-dimensional number can be constructed by multiplying the Ekman number by the Reynolds number as follows:

$$Ek \times Re = \frac{\nu}{\Omega^2 L} \times \frac{K}{\nu} = \frac{K}{\Omega^2 L} \quad (6.8)$$

This new dimensionless number directly relates the eddy diffusivity to the Coriolis effects. In the current combustor, the Ek is around 7×10^{-5} and the Re is around 6×10^4 . Put them into the above formula, the new dimensional number has a value of 4, which is of order one. Therefore, it means that Coriolis effects are as important as the eddy diffusivity (or turbulence).

The high spinning rate of the flow near the combustor center can be understood from the conservation of angular momentum. Angular momentum L , which is defined as the product of the rotational inertial and the angular velocity around the axis, is expressed as:

$$L = I\Omega \quad (6.9)$$

where I is the rotational inertial, which is equal to r^2m for a single particle. For a spinning object, if the rotational inertial suddenly decreases, the angular velocity will increase to conserve angular momentum. In the current combustor, the flow experiences expansion when it flows past the center bodies to the main combustor. When it expands into the combustor center, the radius of its moving path decreases so that the rotational inertial also decreases. To conserve the angular momentum, the angular velocity has to increase to a rate that is larger than 30,000 rpm.

6.4.2.2 Fuel Stream Trajectory and Combustion Performance

The three-dimensional fuel stream trajectories when combustion occurs viewed in the rotating reference frame are plotted in Fig. 6-13. In the non-spinning motion, the fuel stream goes straightly through the combustor (see Fig. 6-13(a)). However, they are bent under the effects of Coriolis force, $-2\vec{\Omega} \times \vec{u}_r$, in the rotating reference frame when there is spinning motion (see Fig. 6-13(b)). The fuel stream expands to the combustor center when it passes over the after-body so that a radial velocity component pointing to the combustor centerline is introduced. Therefore, the trajectory is bent to the right under the effect of the Coriolis force according to the right-hand rule. Furthermore, as shown in Fig. 6-13(b), it can be seen that the fuel stream rotates with a spinning rate higher than 30,000 rpm in the main combustor. As discussed previously, this is caused by the conservation of angular momentum.

Fig. 6-13: Full three dimensional fuel stream trajectories viewed in the relative reference frame:
(a) 0 rpm combustor flow and (b) 30,000rpm combustor flow.

Fig. 6-14: Turbulent kinetic energy in the combustor flows: (a) 0 rpm and (b) 30,000 rpm.

Fuel-air mixing is directly controlled by the turbulence in the combustor, but the non-dimensional analysis revealed that the spinning effect is as important as the eddy diffusivity. To gain insights on how the mixing is affected by the spinning motion, the distributions of the turbulent kinetic energy are provided in Fig. 6-14. In the non-spinning combustor as shown in Fig. 6-14(a), the turbulent kinetic energy downstream of the

transverse fuel injector is increased due to the strong shear between the fuel injection and the main flow. Inside the cavity and the large recirculation zone behind the after-body, the turbulent kinetic energy is also increased. In the stationary combustor, the overall level of the turbulent kinetic energy is around 1000 – 1500 J/kg. In the spinning combustor as shown in Fig. 6-14(b), however, the level of turbulent kinetic energy is significantly increased to around 2500 J/kg, especially in the main combustor. These numbers indicate that the effects of Coriolis force are in the same order of the eddy diffusivity, which is consistent with the above magnitude analysis. In the cavity, the distribution of kinetic energy also follows the cavity vortex motion in both non-spinning and spinning combustor. For example, the kinetic energy in the non-spinning cavity as shown in Fig. 6-14(a) is transported to the bottom of the cavity along the trapped vortex. However, the kinetic energy is confined within the opening of the cavity in the spinning combustor, which is the primary vortex generated by the entrainment of the main flow as discussed previously.

When combustion takes place, the density of the flow drops due to the heat release. The density difference in the spinning motion will induce another sort of secondary flow caused by centrifugal force. Still considering a fluid element in spinning flows, when its density suddenly decreases because of combustion, the centripetal force exerted on the fluid particle is smaller than the radial pressure gradient as follows:

$$\rho \frac{u_{\theta}^2}{r} < -\frac{dp}{dr} \quad (6.10)$$

Hence, the fluid element will be pushed by the pressure gradient towards the combustor center line until to a position that the centripetal force equals to the pressure gradient again. For example, in the temperature contours of the non-spinning and spinning combustor as shown in (a), the entire main combustor is filled with high temperature flows, and even in the region close to the combustor casing. However, in the spinning combustor as shown in (b), the hot flow region in the main combustor is concentrated more to the axis, as the temperature close to the combustor casing is not as high as the non-spinning one. From the temperature contours, it can also be found that the hot cavity pilot flame with high temperature of 2400 K exists for both non-spinning and spinning combustors. Furthermore, the hottest temperature region in the non-spinning combustor is just located behind the after-body as shown in Fig. 6-15(a), while it occurs slightly downstream in the spinning main combustor as indicated in Fig. 6-15(b).

Fig. 6-15: Temperature contours of the combustor: (a) non-spinning and (b) spinning.

The spinning combustor has a more significant three-dimensional flow features due to the Coriolis force. From this point of view, fuel-air mixing is improved by the spinning motion. However, the small recirculation zone in the main combustor generated by the centrifugal force impairs fuel-air mixing. Hence, axial combustion efficiencies are plotted in Fig. 6-16 to gain insights on the overall combustion performance. From the efficiency plot, it could be seen that the spinning combustor has better efficiency (up to 30%) in the region from $z/D = 0 - 2.0$, which means that the three-dimensional flow overcomes the deficiency of the short recirculation zone in this region. When it comes close to the end of combustor around $z/D = 2.0 - 3.0$, the combustion efficiency in the spinning motion is comparable and slightly lower than that of the stationary one. This is because the effects of the short recirculation zone and the secondary flow which confines hot flows more close to the axis are getting stronger. At the exit of the combustor, almost the same combustion efficiency has been achieved for both stationary and spinning combustor.

Fig. 6-16: Combustion efficiency for the non-spinning and spinning combustor.

6.5 Summary

This chapter investigated the effects of a strong spinning motion on the designed ramjet TVC by using LES and RANS simulations. The combustor inflow with solid-body rotation has been assigned at the relative reference frame for the spinning combustor.

The numerical results reveal that Coriolis effects dominate the flow in the TVC under such a high spinning rate (30,000 rpm). Vortex breakdown happens inside the cavity as the single cavity vortex that would occur in non-spinning motion splits into multiple vortices when the combustor rotates. The multi-vortices structure brings strong three-dimensional flows and promotes fuel-air mixing in the cavity. A hot cavity pilot flame is thus obtained. But in the main combustor, the spinning motion also generates a small recirculation zone and makes the hot flows concentrate more on the combustor axis, which in turn impairs fuel-air mixing, and affects the location of hottest temperature region. However, the deficiency of the short recirculation zone is remedied by the stronger kinetic energy distribution due to the spinning. Overall, the spinning motion locally increases the combustion efficiency in the cavity region, but almost the same combustion efficiency is achieved at the exit of the combustor in both the non-spinning and spinning motion.

Chapter 7

Conclusions and Future Work

7.1 Conclusions

A mini-ramjet combustor using the trapped-vortex concept has been investigated numerically in this thesis. The principal objective is to understand the working process of TVC in mini-ramjets, also with the intention to provide guidance for design.

The results and discussion in the preceding chapters have presented highlights on numerical model validation, “trapped” cavity vortex for flame stabilization, fuel injection schemes and mixing, overall combustion performance, and the effects of spinning motions.

Through this study, the procedures of designing the mini-ramjet TVC have been obtained and summarized as follows:

- (1) Derive the combustor operating conditions using stream-thrust analysis from the flight conditions of interest (flight altitude and Mach number);
- (2) Determine cavity sizes that can lock a vortex inside;
- (3) Investigate fuelling schemes which can provide stoichiometric fuel-air mixing in the cavity;
- (4) Evaluate combustor performance from the combustion efficiency, cavity flame temperature and main combustor flame length;
- (5) Further improve injection schemes and determine the final combustor length.

In step 2, the measurable base drag can be used as an indicator to obtain the proper cavity size which traps a vortex. That with a small drag corresponds to the appropriate cavity size. Among the fuel injection schemes investigated, those injecting directly into the cavity cannot hold a pilot flame due to an overly fuel-rich cavity. However, the fueling strategy by injecting “early” in the upstream is found to be rather effective. The fuel-air momentum flux ratio controls the penetration of the fuel stream and the amount of fuel entering the cavity. When the momentum flux ratio is between 0.61 and 1.0, a mixture equivalence ratio of around 1.0 can be obtained in the cavity. However, combustion simulation results reveal that the injection scheme with a momentum flux ratio of around 1.0 has a better combustion

performance for the gaseous fuel propane selected here. The angled injection results indicate the normal injection with 90 degree upstream the cavity provides the best performance inside the combustor. However, the final overall combustion efficiency is not much affected. The current designed combustor can be as compact as $z/D = 2.5$, which indicates a good engine thrust-weight ratio.

Another objective of the thesis is to study the effects of a spinning motion (at 30,000 rpm) on the designed mini-ramjet TVC, as the combustor may be installed on a spin-stabilized projectile. The numerical results reveal that Coriolis effects dominate the flow in the TVC under such a high spinning rate (30,000 rpm). Vortex breakdown happens inside the cavity as the single cavity vortex that would occur in non-spinning motion splits into multiple vortices when the combustor rotates. The multi-vortices structure brings strong three-dimensional flows and promotes fuel-air mixing in the cavity. A hot cavity pilot flame is thus obtained. But in the main combustor, the spinning motion also generates a small recirculation zone and makes the hot flows concentrate more on the combustor axis, which in turn impairs fuel-air mixing, and affects the location of hottest temperature region. However, the deficiency of the short recirculation zone is remedied by the stronger kinetic energy distribution due to the spinning. Overall, the spinning motion locally increases the combustion efficiency in the cavity region, but almost the same combustion efficiency is achieved at the exit of the combustor in both the non-spinning and spinning motion.

7.2 Suggestions for Future Work

The current work shows that the TVC is a promising candidate for miniature ramjet combustors with a short length. However, considerable work is still required before the practical application of the TVC in ramjets.

Firstly, a LES of the mixing and combustion for the spinning combustor should be considered, if a more accurate prediction of the flow physics is of interest, although the current RANS has satisfied the need for the design. Secondly, more practical liquid fuel (like kerosene and JP-type) can be studied and applied in TVC in the future, because the liquid fuel has a larger energy density in volume which is more attractive in practical applications. Thirdly, a direct-connect type ground testing experiment should be conducted to test the performance of the designed combustor. Fourthly, the shape of the cavity can be studied or optimized, and the interaction of the cavity acoustics and the combustion can be investigated as well.

Furthermore, the current study does not consider the effects of the supersonic inlet. If a real supersonic inlet is implemented, the combustor inflow may include thick boundary

layer, non-uniform flows, reflected shocks and inlet-combustor interactions. Hence, an inlet should be added to consider more practical operating conditions for the combustor.

REREFENCES

1. Fry, R. S., "A Century of Ramjet Propulsion Technology Evolution", *Journal of Propulsion and Power*, Vol. 20, No. 1, 2004, pp. 27-58.
2. Ramgen Power Systems, "Development and Testing of a Pre-Prototype Mach 2 Ramgen Engine Final Report (DOE Award No. DE-FC26-00NT40915)", 2001: Department of Energy, Bellevue, WA.
3. Ramgen Power Systems, "Development and Testing a Pre-Prototype Ramgen Engine Budget Period 3 Final Report (DOE Award No. DE-FC26-00NT40915)", 2003: Department of Energy, Bellevue, WA.
4. Hsu, K.-Y., Gross, L. P., Trump, D., and Roquemore, W. M., "Performance of a Trapped-Vortex Combustor", *AIAA paper 95-0810*, 1995.
5. Hsu, K.-Y., Goss, L. P., and Roquemore, W. M., "Characteristics of a Trapped-Vortex Combustor", *Journal of Propulsion and Power*, Vol. 14, No. 1, 1998, pp. 57-65.
6. Dugger, G. L., "Ramjets", *AIAA Selected Reprinted Series*, Vol. VI, New York, June 1969.
7. <http://en.wikipedia.org/wiki/Ramjet>.
8. Rancourt, D., Picard, M., Denninger, M., Plante, J.-S., Chen, J., and Yousefpour, A., "Rim-Rotor Rotary Ramjet Engine, Part 1: Structural Design and Experimental Validation", *Journal of Propulsion and Power*, Vol. 28, No. 6, 2012, pp. 1293-1303.
9. Picard, M., Rancourt, D., Plante, J.-S., and Brouillette, M., "Rim-Rotor Rotary Ramjet Engine, Part 2: Quasi-One-Dimensional Aerothermodynamic Design", *Journal of Propulsion and Power*, Vol. 28, No. 6, 2012, pp. 1304-1314.
10. Zukoski, E. E., Ph.D. Thesis, "Flame Stabilization on Bluff Bodies at Low and Intermediate Reynolds Numbers", *California Institute of Technology*, 1954.
11. Longwell, J. P., Chenevey, J. E., Clark, W. W., and Frost, E. E., "Flame Stabilization by Baffles in A High Velocity Gas Stream", *Symposium on Combustion and Flame, and Explosion Phenomena*, Vol. 3, No. 1, 1949, pp. 40-44.
12. Penner, S. S. and Williams, F., "Recent Studies on Flame Stabilization of Premixed Turbulent Gases", *Applied Mechanics Reviews*, Vol. 10, No. 6, 1957.
13. Gao, G. and Ning, H., "Theoretical and Experimental Study in Stability of The Barchan Dune Vortex Flame", *Journal of Engineering Thermophysics (in Chinese)*, Vol. 3, No. 1, 1982.
14. <http://zenstoves.net/How.htm>.
15. Ahmed, K., Ph.D. Thesis, "Flame Stabilization in a Model Ramjet Combustor Using a Transverse Slot Jet", *University at Buffalo, State University of New York*, 2009.
16. Ben-Yakar, A. and Hanson, R. K., "Cavity Flame-Holders for Ignition and Flame Stabilization in Scramjets: An Overview", *Journal of Propulsion and Power*, Vol. 17, No. 4, 2001, pp. 869-877.
17. Ben-Yakar, A., Ph.D. Thesis, "Experimental Investigation of Mixing and Ignition of Transverse Jets in Supersonic Crossflows", *Stanford University*, December 2000.
18. Shouse, D. T., "Trapped Vortex Combustor Technology", *MITE Workshop on Goals and Technologies for Future Gas Turbines*, at The Georgia Institute of Technology Manufacturing Research Center, 2000.
19. Mair, W. A., "The Effect of a Rear-Mounted Disc on the Drag of a Blunt-Based Body of Revolution", *Aeronautical Quarterly*, Vol. 10, No. 4, 1965, pp. 350-360.
20. Little, B. H. and Whipkey, R. R., "Locked Vortex Afterbodies", *Journal of Aircraft*, Vol. 16, No. 5, 1979, pp. 296-302.
21. Gharib, M. and Roshko, A., "The Effect of Flow Oscillations on Cavity Drag", *Journal of Fluid Mechanics*, Vol. 177, No., 1987, pp. 501-530.

22. Roquemore, W. M., Dale, S., Dave, B., Art, J., Clayton, C., Beverly, D., Hsu, K. Y., Katta, V., Sturgess, G., and Illari, V., "Vortex Combustor Concept for Gas Turbine Engines", *AIAA paper 2001-0483*, 2001.
23. Steele, R. C., "Trapped Vortex Combustion", *The Gas Turbine Handbook*, U.S. Department of Energy, National Energy Technology Laboratory, 2006.
24. Katta, V. R. and Roquemore, W. M., "Numerical Studies of Trapped-Vortex Combustor", *AIAA paper 96-2660*, 1996.
25. Katta, V. R. and Roquemore, W. M., "Numerical Studies on Trapped-Vortex Concepts for Stable Combustion", *Journal of Engineering for Gas Turbines and Power*, Vol. 120, No. 1, 1998.
26. Katta, V. R. and Roquemore, W. M., "Study on Trapped-Vortex Combustor - Effect of Injection on Flow Dynamics", *Journal of Propulsion and Power*, Vol. 14, No. 3, 1997, pp. 273-281.
27. Katta, V. R. and Roquemore, W. M., "Study on Trapped-Vortex Combustor - Effect of Injection on Dynamics of Non-Reacting and Reacting Flows in a Cavity", *AIAA paper 97-3256*, 1997.
28. Sturgess, G. J., Hsu, K. Y., Sturgess, G., and Hsu, K. Y., "Entrainment of Mainstream Flow in a Trapped-Vortex Combustor", *AIAA paper 97-0261*, 1997.
29. Sturgess, G. J. and Hsu, K.-Y., "Combustion Characteristics of a Trapped Vortex Combustor", *RTO AVT Symposium on Gas Turbine Engine Combustion, Emission and Alternative Fuels*, Lisbon, Portugal, October 12-16, 1998.
30. Hsu, K.-Y., Carter, C., Katta, V., and Roquemore, W., "Characteristics of Combustion Instability Associated with Trapped-Vortex Burner", *AIAA paper 99-0488*, 1999.
31. Stone, C. and Menon, S., "Simulation of Fuel-Air Mixing and Combustion in a Trapped-Vortex Combustor", *AIAA paper 2000-0478*, 2000.
32. Burrus, D. L., Johnson, A. W., Roquemore, W. M., and Shouse, D. T., "Performance Assessment of a Prototype Trapped Vortex Combustor for Gas Turbine Application", *Proceedings of ASME TURBO EXPO 2001*, New Orleans, Louisiana, June 4-7, 2001.
33. Bucher, J., Edmonds, R. G., Steele, R. C., Kendrick, D. W., Chenevert, B. C., and Malte, P. C., "The Development of a Lean-Premixed Trapped Vortex Combustor", *Proceedings of ASME TURBO EXPO 2003*, Atlanta, Georgia, July 16-19, 2003.
34. http://www.ramgen.com/tech_vortex_conventional.html.
35. Hendricks, R. C., Shouse, D. T., Roquemore, W. M., Burrus, D. L., Duncan, B. S., Ryder, R. C., Brankovic, A., Liu, N.-S., Gallagher, J. R., and Hendricks, J. A., "Experimental and Computational Study of Trapped Vortex Combustor Sector Rig With Tri-Pass Diffuser", *NASA/TM-2004-212507*, 2004.
36. Nardo, A. D., Calchetti, G., Mongiello, C., Giammartini, S., and Rufoloni, M., "CFD Modelling of an Experimental Scaled Model of a Trapped Vortex Combustor", *Proceedings of the European Combustion Meeting 2009*, 2009.
37. Liu, Y. Y., Li, R. M., Liu, H. X., and Yang, M. L., "Effects of Fueling Scheme on the Performance of a Trapped Vortex Combustor Rig", *AIAA paper 2009-4831*, 2009.
38. Li, Q., Pan, Y., Tan, J. g., and Wang, Z. g., "Experiment Research of Ramjet with Cavity-Based Flameholder", *AIAA paper 2009-5049*, 2009.
39. He, X. M. and Zhang, J. Y., "An Investigation on the Performance of the Ramjet Trapped-Vortex Combustor", *AIAA paper 2013-3768*, 2013.
40. Zhang, J. Y., He, X. M., Jin, Y., and Wang, L., "Experimental and Numerical Investigations on Liner Cooling Characteristics of a Trapped Vortex Combustor", *AIAA 2013-3770*, 2013.
41. Fei, L. Q., Zhang, Y., Niu, Z. G., and Ling, W. H., "Numerical Simulation On Trapped-Vortex Combustor Model for Ramjet Engine", *Journal Of Propulsion Technology (in Chinese)*, Vol. 32, No. 5, 2011.

42. Chen, S., Chue, R., Tue, T. N., and Yu, S., "Numerical Investigations of Small-Size High Throughflow Speed Combustors with Trapped-Vortex Concept", *AIAA paper 2013-3769*, 2013.
43. Chen, S., Chue, S. K. R., Schlüter, J. U., Ngguyen, T. T., and Yu, C. M. S., "Numerical Investigation of a Trapped Vortex Miniature Ramjet Combustor", *Journal of Propulsion and Power*, Vol. 31, No. 3, 2015, pp. 872-882.
44. Hinze, J. O., "*Turbulence*", McGraw Publishing Co., New York, 1975.
45. Menter, F. R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Application", *AIAA Journal*, Vol. 32, No. 8, 1994, pp. 1598-1605.
46. Gibson, M. M. and Launder, B. E., "Ground Effects on Pressure Fluctuations in the Atmospheric Boundary Layer", *Journal of Fluid Mechanics*, Vol. 86, No., 1978, pp. 491-511.
47. Smagorinsky, J., "General Circulation Experiments with the Primitive Equations.", *Monthly Weather Review*, Vol. 91, No. 3, 1963, pp. 99-164.
48. Germano, M., Piomelli, U., Moin, P., and Cabot, W., "A Dynamic Localization Model for Large-Eddy Simulation of Turbulent Flows", *Physics of Fluids*, Vol. A, No. 3, 1991, pp. 1760-1765.
49. Magnussen, B. F. and Hejertager, B. H., "On Mathematical Modeling of Turbulent Combustion with Special Emphasis on Soot Formation and Combustion", *Symposium (International) on Combustion*, Vol. 16, No. 1, 1977, pp. 719-729.
50. Magnussen, B. F., "On the Structure of Turbulence and a Generalized Eddy Dissipation Concept for Chemical Reaction in Turbulent Flow", *19th AIAA Aerospace Science Meeting, AIAA paper 81-0042*, 1981.
51. Magnussen, B. F., "The Eddy Dissipation Concept - A Bridge between Science and Technology", *ECCOMAS Thematic Conference on Computational Combustion*, Lisbon, 2005.
52. Corin, S., "*The Scramjet Engine, Process and Characteristics*", Cambridge Aerospace Series, Cambridge University Press, 2009.
53. Pierce, C. D. and Moin, P., "Large Eddy Simulation of a Confined Coaxial Jet with Swirl and Heat Release", *AIAA Paper 98-2892*, 1998.
54. Pierce, C. D., Ph.D. Thesis, "Progress-Variable Approach for Large-Eddy Simulation of Turbulent Combustion", *Stanford University*, 2001.
55. Germano, M., Piomelli, U., P., M., and Cabot, W., "A Dynamic Subgrid-Scale Eddy Viscosity Model", *Physics of Fluids A*, Vol. 3, No. 7, 1991, pp. 1760-1765.
56. Lilly, D. K., "A Proposed Modification of the Germano Subgrid-Scale Closure Method", *Physics of Fluids A*, Vol. 4, No. 3, 1992, pp. 633-635.
57. Pierce, C. D. and Moin, P., "Method for Generating Equilibrium Swirling Inflow Conditions", *AIAA Journal*, Vol. 36, No. 7, 1998, pp. 1325-1327.
58. Schlüter, J. U., "Static Control of Combustion Oscillations by Coaxial Flows: An LES Investigation", *Journal of Propulsion and Power*, Vol. 20, No. 3, 2004, pp. 460-467.
59. Schlüter, J. U., Pitsch, H., and Moin, P., "Large Eddy Simulation Inflow Conditions for Coupling with Reynolds-Averaged Flow Solvers", *AIAA Journal*, Vol. 42, No. 3, 2004, pp. 478-484.
60. Schlüter, J. U., Pitsch, H., and Moin, P., "Outflow Conditions for Integrated Large-Eddy Simulation/Reynolds-Averaged Navier-Stokes Simulations", *AIAA Journal*, Vol. 43, No. 1, 2005, pp. 156-164.
61. Zhang, X. Q., Theissen, P., and Schlüter, J. U., "Towards Simulation of Flapping Wings Using Immersed Boundary Method", *International Journal for Numerical Methods in Fluids*, Vol. 71, No. 4, 2013, pp. 552-536.
62. Zhang, X. Q., Chai, V., and Schlüter, J. U., "Reynolds-Number Dependency of the Leading Edge Vortex on a Fast Pitch-up Flat Plate", *AIAA paper 2011-3794*, 2011.
63. Sarkar, A. and Schlüter, J. U., "Numerical Investigation of the Turbulent Energy Budget in the Wake of Freely Oscillating Elastically Mounted Cylinder at Low

- Reduced Velocities", *Journal of Fluids and Structures*, Vol. 43, No., 2013, pp. 441-462.
64. Sarkar, A. and Schlüter, J. U., "Large Eddy Simulations of Turbulent Mixing Layers Excited with Two Frequencies", *Turbulence and Combustion*, Vol. 92, No. 3, 2014, pp. 651-671.
 65. Devenport, W. J. and Sutton, E. P., "An Experimental Study of Two Flows Through an Axisymmetric Sudden Expansion", *Experiments in Fluids*, Vol. 14, No. 6, 1993, pp. 423-432.
 66. Bardina, J. E., Huang, P. G., and Coakley, T. J., "Turbulence Modeling, Validation, Testing and Development", *NASA-TM-110446*, 1997.
 67. Merz, R. A., Page, R. H., and Przirembel, C. E. G., "Subsonic Axisymmetric Near-Wake Study", *AIAA Journal*, Vol. 16, No. 7, 1978, pp. 656-662.
 68. Depres, D., Reijasse, P., and Dussauge, J. P., "Analysis of Unsteadiness in Afterbody Transonic Flow", *AIAA Journal*, Vol. 42, No. 12, 2004, pp. 2541-2550.
 69. Meliga, P. and Reijasse, P., "Unsteady Transonic Flow Behind an Axisymmetric Afterbody with Two Boosters", *AIAA paper 2007-4564*, 2007.
 70. Briones, A. M., Zelina, J., and Katta, V. R., "Flame Stabilization in Small Cavities", *AIAA Journal*, Vol. 48, No. 1, 2010, pp. 224-235.
 71. <http://www.sandia.gov/TNF/DataArch/H2HeData.html>.
 72. Xiao, X., Hassan, H. A., and Baurle, R. A., "Modeling Scramjet Flows with Variable Turbulent Prandtl and Schmidt Numbers", *AIAA Journal*, Vol. 45, No. 6, June 2007, pp. 1415-1423.
 73. Thierry, P. and Denis, V., "*Theoretical and Numerical Combustion, Second Edition*", Philadelphia, Edwards, 2005.
 74. Rokke, N. A., Hustad, J. E., and Sonju, O. K., "A Study of Partially Premixed Unconfined Propane Flames", *Combustion and Flames*, Vol. 97, No. 1, 1994, pp. 88-106.
 75. Takahashi, F., Schmoll, W. J., and Katta, V. R., "Attachment Mechanisms of Diffusion Flames", *Symposium (International) on Combustion*, Vol. 27, No. 1, 1998, pp. 675-684.
 76. Dellenback, P. A., Metzger, D. E., and Neitzel, G. P., "Measurements in Turbulent Swirling Flow Through an Abrupt Axisymmetric Expansion", *AIAA Journal*, Vol. 26, No. 6, 1988, pp. 669-681.
 77. Heiser, W. H. and Pratt, D. T., "*Hypersonic Airbreathing Propulsion*", AIAA Education Series, 1994.
 78. Fleeman, E. L., "*Tactical Missile Design-second edition*", AIAA Education Series, 2006.
 79. <http://www.pdas.com/atmos.html>.
 80. Sun, M. B., Wang, Z. G., Liang, J. H., and Geng, H., "Flame Characteristics in Supersonic Combustor with Hydrogen Injection Upstream of Cavity Flameholder", *Journal of Propulsion and Power*, Vol. 24, No. 4, July-August 2008.
 81. Peng, S. W. and Dolling, D. S., "Passive Control of Pressure Oscillation in Hypersonic Cavity Flow", *AIAA paper 96-0444*, 1996.
 82. Zhang, X., Rona, A., and Edwards, J. A., "The Effects of Trailing Edge Geometry on Cavity Flow Oscillation Driven by a Supersonic Shear Layer", *The Aeronautical Journal*, Vol. 102, No. 1013, 1988, pp. 129-136.
 83. Jeong, E., O'Byrne, S., Jeung, I., and Houwing, A. F. P., "Investigation of Supersonic Combustion with Angled Injection in a Cavity-Based Combustor", *Journal of Propulsion and Power*, Vol. 24, No. 6, 2008, pp. 1258-1268.
 84. Cohen, H., Rogers, G. F. C., and Saravanamuttoo, H. I. H., "*Gas Turbine Theory, second edition*", Longman Group Limited, 1972.
 85. Ebrahimi, H. B., Malo-Molina, F. J., and Gaitonde, D. V., "Numerical Simulation of Injection Strategies in a Cavity-Based Supersonic Combustor", *Journal of Propulsion and Power*, Vol. 28, No. 5, 2012, pp. 991-999.

86. Verra, R. G. and Mayer, A. E. H. J., "The Role of the TNO Free Jet Test Facility in Solid Fuel Ramjet Projectile Development", *AIAA paper 2005-3828*, 2005.
87. Kline, R., Herrmann, W. R., and Oskay, V., "A Determination of the Aerodynamic Coefficients of the 155 mm, M549 Projectile", *Picatinny Arsenal Technical Report No. 4764*, 1974.
88. Farthing, P. R., Long, C. A., Owen, J. M., and Pincombe, J. R., "Rotating Cavity with Axial Throughflow of Cooling Air: Heat Transfer", *Journal of Turbomachinery*, Vol. 114, No. 1, 1992, pp. 229-236.
89. Farthing, P. R., Long, C. A., Owen, J. M., and Pincombe, J. R., "Rotating Cavity with Axial Throughflow of Cooling Air: Flow Structure", *Journal of Turbomachinery*, Vol. 114, No. 1, 1992, pp. 237-246.
90. Lewis, G. D., "Combustion in a Centrifugal-Force Field", *Symposium (International) on Combustion*, Vol. 13, No. 1, 1971, pp. 625-629.
91. Lewis, G. D., Shadowen, J. H., and Thayer, E. B., "Swirling Flow Combustion", *Journal of Energy*, Vol. 1, No. 4, 1977, pp. 201-205.
92. Katta, V. R., Blunck, D., and Roquemore, W. M., "Effects of Centrifugal Forces on Flame Stability in an Ultra-Compact Combustor", *AIAA paper 2013-1046*, 2013.
93. "Illustrated Experiments in Fluid Mechanics: the NCFMF Book of Film Notes", National Committee for Fluid Mechanics Films, The MIT Press, 1972.
94. https://en.wikipedia.org/wiki/Tea_leaf_paradox.
95. Greitzer, E. M., Tan, C. S., and Graf, M. B., "Internal Flows: Concepts and Applications", Cambridge Engine Technology Series (No. 3), Cambridge University Press, 2004.
96. Lee, H. J., Lee, B. J., Kim, S. D., and Jeun, I.-S., "Flow Characteristics of Small-Sized Supersonic Inlets", *Journal of Propulsion and Power*, Vol. 21, No. 2, 2011, pp. 306-318.
97. Tennekes, H. and Lumley, J. L., "A first Course in Turbulence", The MIT Press, 1972.