EFD/CFD Activities in Research for Reusable Launch Vehicles

再使用型宇宙往還機に向けた研究におけるEFD/CFDの取り組み

Yasuhiro TANI, Shigeru ASO, Kentaro HAYASHI, Kei INOUE,
Kohei YAMAGUCHI and Takuro ISHIDA
谷 泰寛、麻生 茂、林健太郎、井上 慶、山口耕平、石田拓郎

Department of Aeronautics and Astronautics, Kyushu University
Fukuoka, Japan

Second Workshop of Integration of EFD and CFD, 23-24 Feb. 2009

Space Transportation Systems Engineering Laboratory, Kyushu University

---

Contents

1. Research Area in our Laboratory
2. Experimental Facilities and CFD Tools
3. Aerodynamic characteristics of RLV configuration
4. Aerodynamic heating reduction research
5. Fuel/Air mixing of SCRAM jet engine combustor

Space Transportation Systems Engineering Laboratory, Kyushu University

Research Area in our laboratory

- High and Low Speed Aerodynamic Characteristics of Reusable Launch Vehicle (RLV)
- Reduction of aerodynamic heating during reentry phase for the RLVs
- Future aerospace propulsion system (SCRAM-jet engin, PDE)
- Ecological aircraft for future commuter air transportation

Research activities on RLVs

- **Aerodynamic characteristics**

  fuselage cross sectional configuration

- **Reduction of Aerodynamic heating**

  opposing jet, Film cooling

- **Engines for hypersonic flight**

  scramjet engine, Pulse detonation engine

Experimental Facilities in use

Test facilities in Department of Aeronautics and Astronautics, Kyushu univ.

- Low Noise Low speed wind tunnel
- Supersonic wind tunnel
- Transonic wind tunnel

Test facilities in Space Transportation Systems Lab.

- Detonation driven Expansion tube
- Free piston shock tunnel
- Shock Tube

Other test facilities in use

- ISAS/JAXA Supersonic wind tunnel and Transonic wind tunnel

Former Wind tunnels in Hakozaki Campus

2m Low speed wind tunnel

15cm Supersonic wind tunnel (Mach 4)

Kyushu Univ. Low Noise Wind Tunnel

Low Noise Test Section (No.1)

Göttingen type, in anechoic chamber

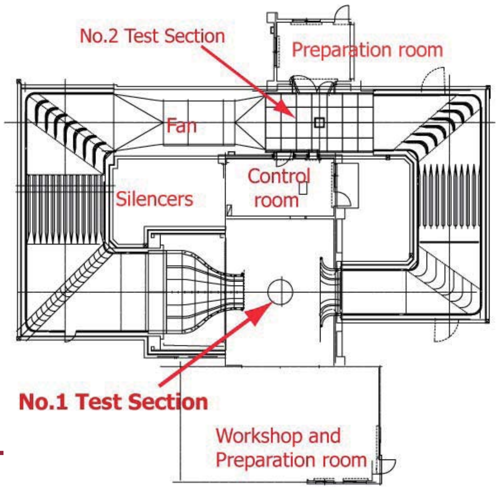
2 m width octagonal, 5 m length

Max 60 m/s, 65 dB at 40 m/s

Large Closed Test Section (No.2)

3.5m x 3.5 m (max 15m/s),

30m/s with 3.5m x 1.5m insert



Space Transportation Systems Engineering Laboratory, Kyushu University

Kyushu Univ. Transonic Wind Tunnel

Blow down type Transonic wind tunnel

Mach 0.3～1.3

150mm x 450mm closed test section with slit walls

Space Transportation Systems Engineering Laboratory, Kyushu University

Kyushu Univ. Supersonic Wind Tunnel

Blow down type Supersonic wind tunnel

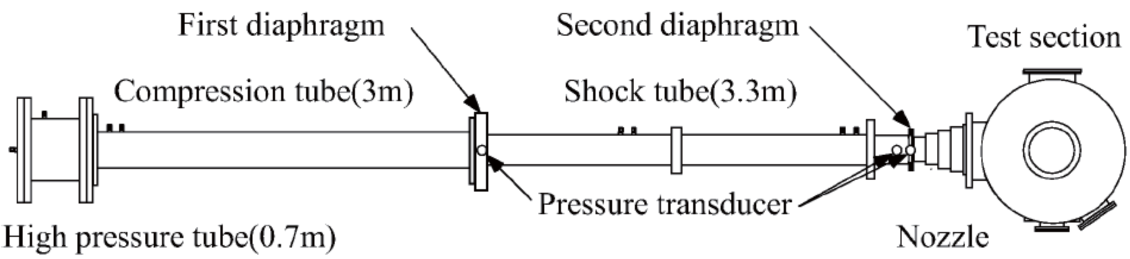
- Mach 2.5 and 3.5
- 250x200mm closed test section

Kyushu Univ. Transonic Wind Tunnel

High enthalpy Flow test apparatus

- Free piston shock tunnel
- Detonation driven Expansion tube
- Normal Shock Tube

Free piston shock tunnel



High pressure tube : L 0.7 m, D 136.6 mm

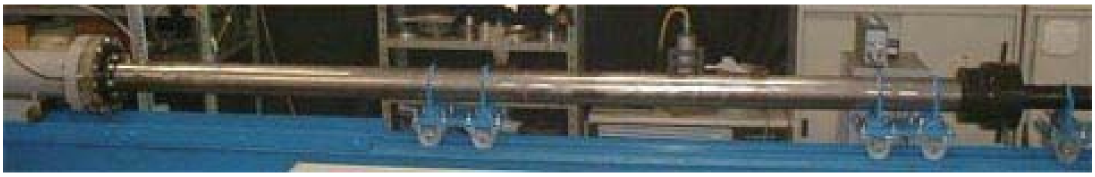
Compression tube : L 3.0 m, D 70 mm

Piston : Mass 1.13 kg, L 49 mm

Shock tube : L 3.3 m, D 60 mm

Nozzle : Exit Diameter 270 mm,Area ratio 190, Design Mach 8

Test section : Volume 1m^3



CFD Tools

2D / 3D Navier-Stokes code (in house)

- Compressible Full Navier-Stokes / Euler
- Structured grid with Multi-Block formulation
- AUSM-DV scheme, LU-ADI
- Turbulence models
  - Wilcox k-ω 2eq. model
  - Spalart-Allmaras 1eq. model
  - Baldwin-Lomax algebraic model
- Chemical reaction

Grid Generation

- Gridgen
- Transfinite/Elliptic grid generator (In house)

CFD Tools

Design tool

- PANAIR
- XFOIL
- Vortex lattice code
- Newtonian Flow code
- DATCOM

Background (1)

Space tourism market has been more active.

- VIRGIN GALACTIC
  - → Space Ship Two and White Knight Two
- ROCKET PLANE [GLOBAL]
  - → Rocket Plane XP

http://www.virgingalactic.com/

http://www.rocketplane.com/

↓

New Technology and Development are required for Space Transportation System

Current Systems

'Expendable Rockets' & 'Space Shuttle (partly reuse)'

↓

High Economic Efficiency
Global Environment

Reusable Launch Vehicle (RLV)

Background (2)

Considering the aerodynamic heating
and structural weight

⬇

RLV's Wings are small compared with fuselages relatively

- Worse aerodynamic characteristics in subsonic region
- Fuselages have more effects on aerodynamic performance

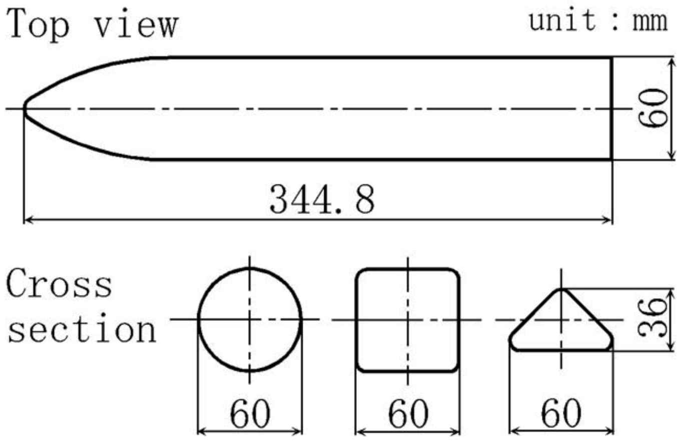
Required aerodynamic characteristics & performance for RLV

- **High $C_{L\max}$** ・・・ Low landing speed, rate of descent
- **High $(L/D)_{\max}$** ・・・ Shallow flight path angle
  (Increment of downrange & cross range)
  Low rate of descent
- **Maneuverability, Safety, & Reliability**

Simplified fuselage models

Wind tunnel test models

- simplified fuselage model
- three different cross section



Transonic and Supersonic Wind Tunnel

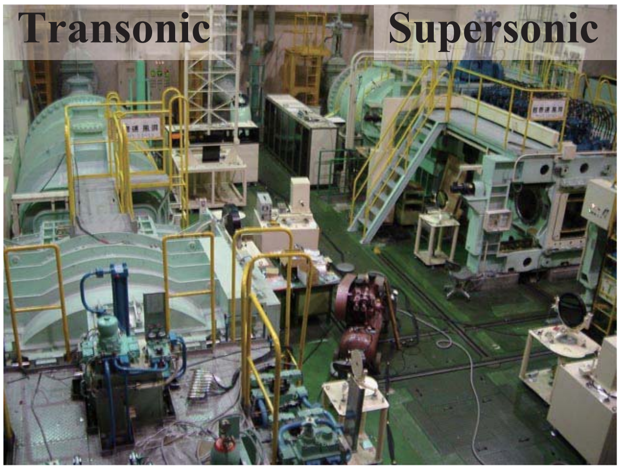
ISAS/JAXA Wind Tunnels

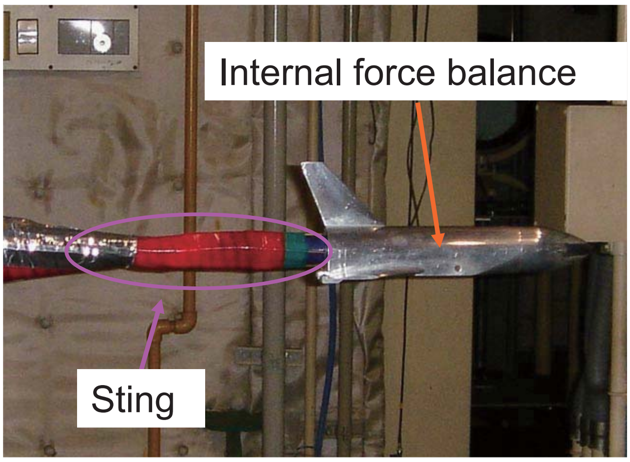
Transonic ： $M_\infty = 0.3 \sim 1.3$

Supersonic ： $M_\infty = 1.5 \sim 4.0$

Blow down type

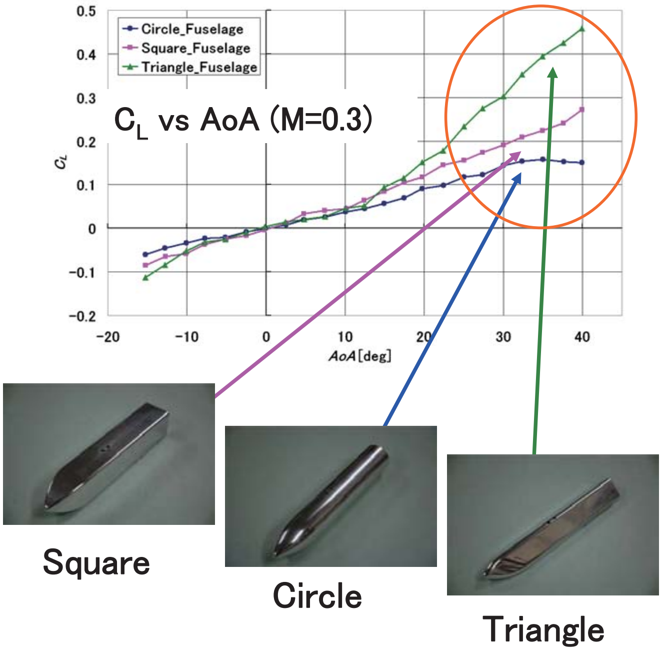
Test section：600 mm ×600 mm

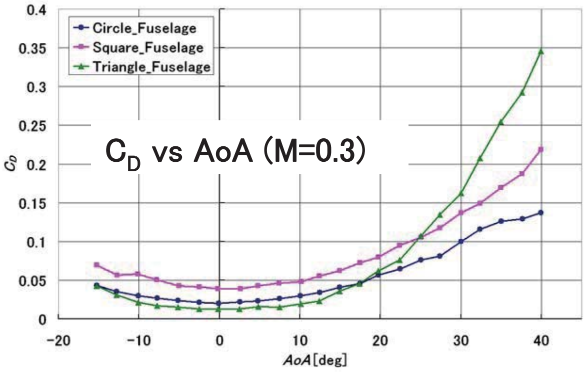


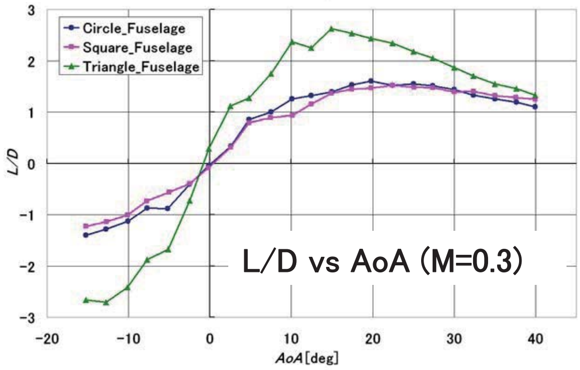


Wind tunnel test result (subsonic)

Significant differences on aerodynamic characteristics, highest Lift for triangle model.



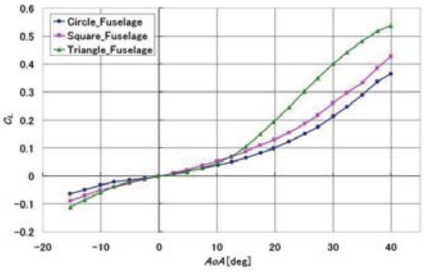




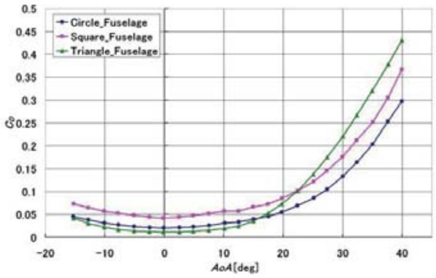
KYUSHU UNIVERSITY 2011
100th Anniversary

Wind tunnel test result (transonic)

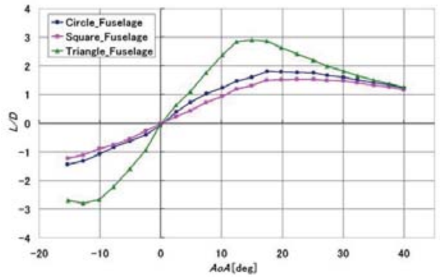
九州大学
KYUSHU UNIVERSITY



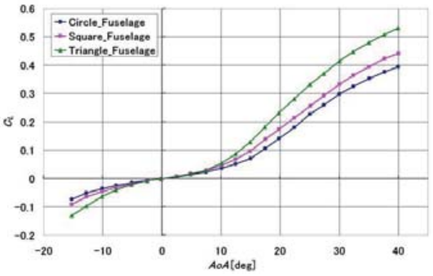
C_L vs AoA (M=0.9)



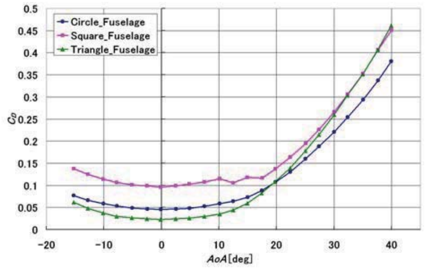
C_D vs AoA (M=0.9)



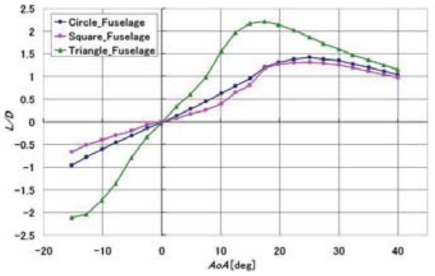
L/D vs AoA (M=0.9)



C_L vs AoA (M=1.3)



C_D vs AoA (M=1.3)



L/D vs AoA (M=1.3)

Space Transportation Systems Engineering Laboratory, Kyushu University

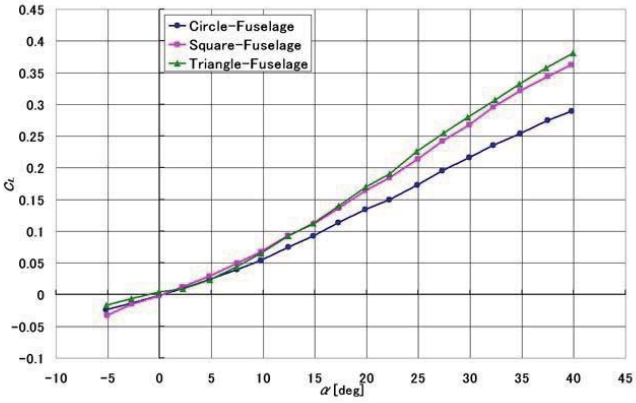
KYUSHU UNIVERSITY 2011
100th Anniversary

Wind tunnel test result (supersonic)

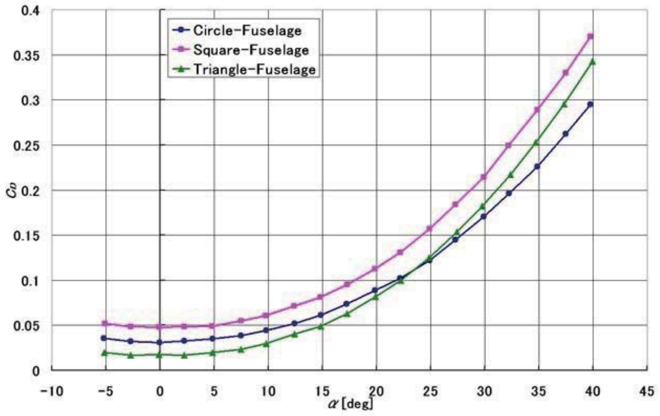
九州大学
KYUSHU UNIVERSITY

$M_\infty = 4.0$

$C_L - \alpha$ curves,



$C_D - \alpha$ curves



Cannot observe the significant differences on aerodynamic characteristics compared to subsonic region

Space Transportation Systems Engineering Laboratory, Kyushu University

Wind tunnel test result (flow pattern)

[Top View]

Oil Flow Visualization (M=4.0, AoA=30deg)

Oil Flow Visualization (M=0.3, AoA=30deg)

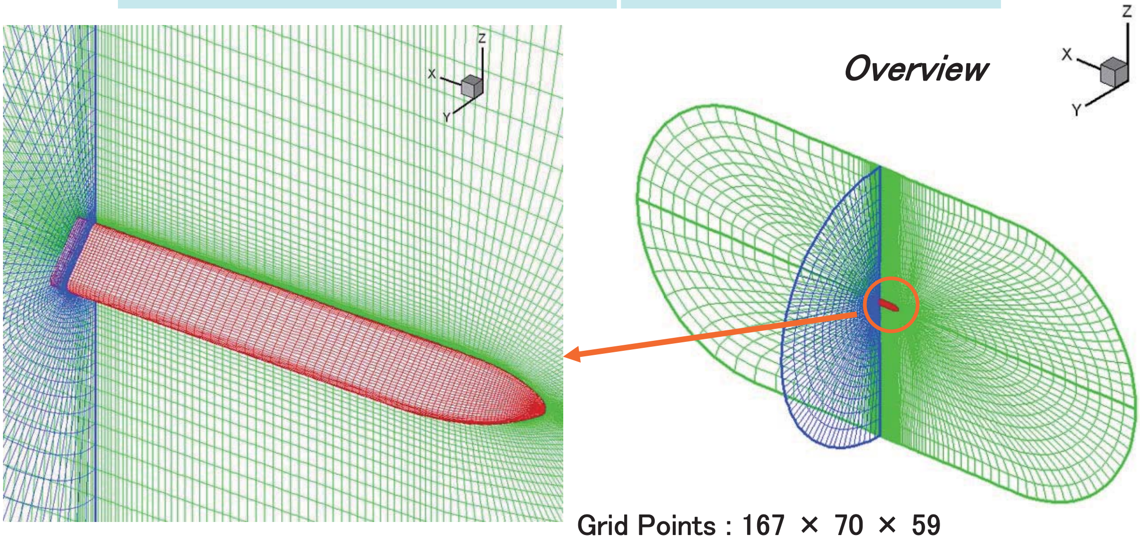
Should be clarify the effect of the separated vortices on the aerodynamic forces

Numerical analysis : scheme

Governing Equation	3D Full Navier-Stokes Equation (RANS)
Convective terms	AUSM-DV scheme
Viscous terms	2nd order central difference
Time integration	Euler Explicit method
Turbulence model	k-ω two equation model

Flow condition and Computational Grid

Free Stream Mach Number	0.29
Total Pressure	1.51 × 10^5 Pa
Total Temperature	293.2 K
Angle of Attacks	30 deg



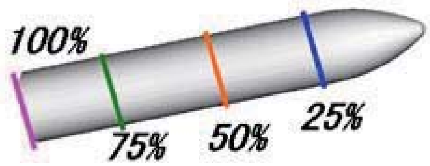
Grid Points : 167 × 70 × 59

Space Transportation Systems Engineering Laboratory, Kyushu University

---

Comparison of CFD and Flow visualization

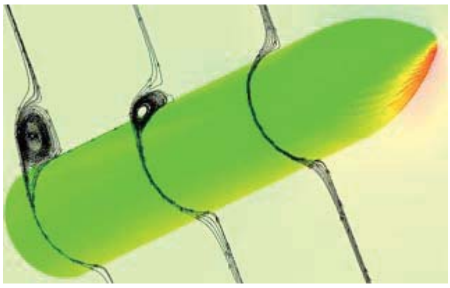
(Circle Fuselage, AoA=30deg)



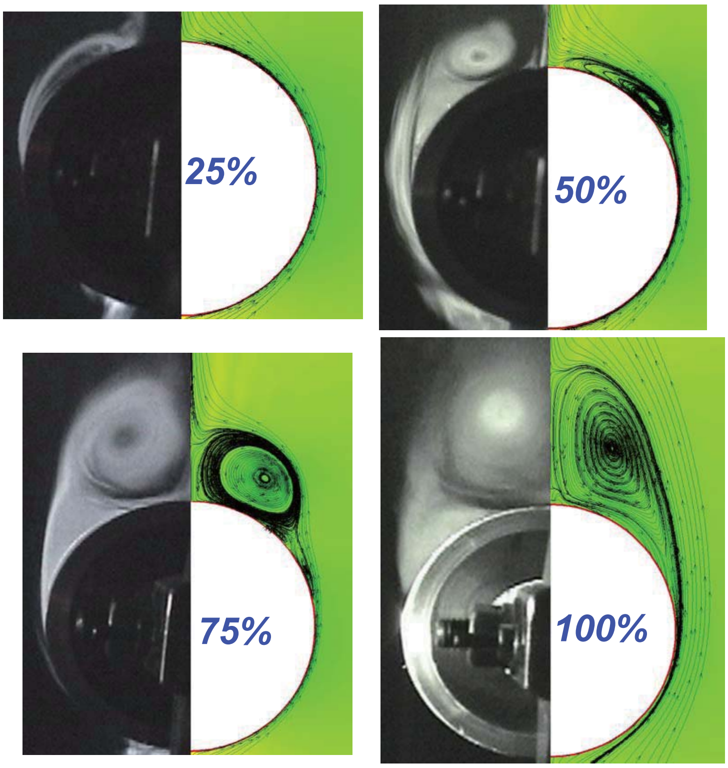
Aerodynamic Forces

Exp. : C_L=0.14, C_D=0.10

CFD : C_L=0.14, C_D=0.12



Growth of vortices & surface pressure

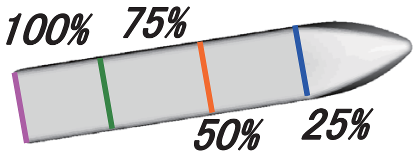


・*Smoke : 3.5 m/s (Re=8.6×10^4), CFD : M = 0.3 (Re=3.2×10^6)*

Space Transportation Systems Engineering Laboratory, Kyushu University

Comparison of CFD and Flow visualization

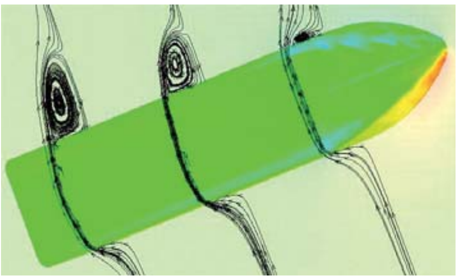
(Square Fuselage, AoA=30deg)



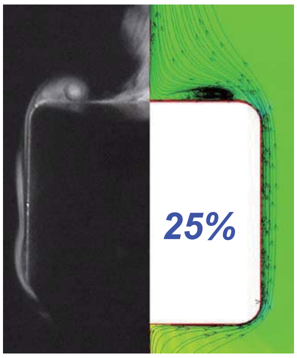
Aerodynamic Forces

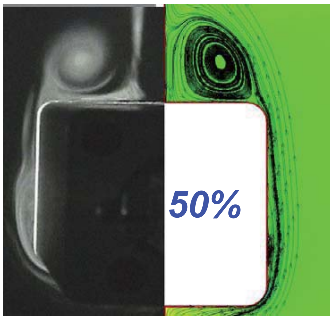
Exp. : C_L=0.19, C_D=0.14

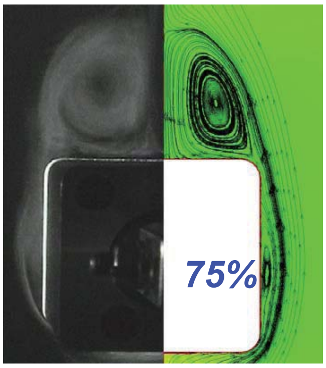
CFD : C_L=0.22, C_D=0.17

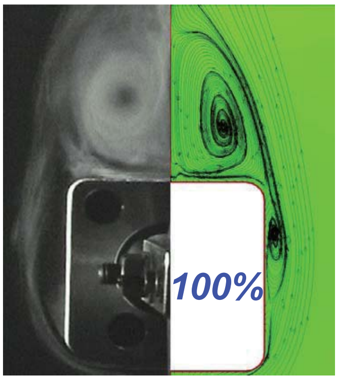


Growth of vortices & surface pressure





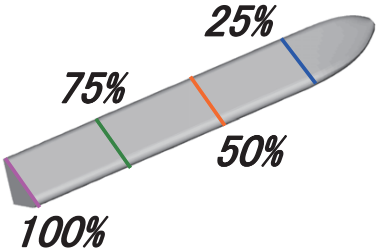




Space Transportation Systems Engineering Laboratory, Kyushu University

Comparison of CFD and Flow visualization

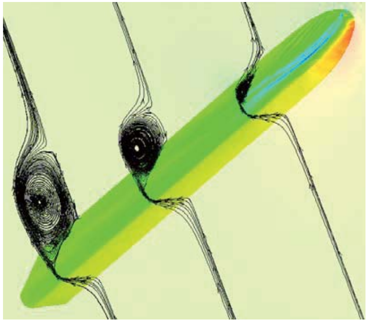
(Triangle Fuselage, AoA=30deg)



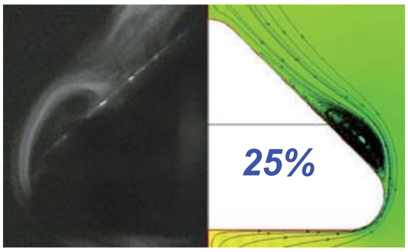
Aerodynamic Forces

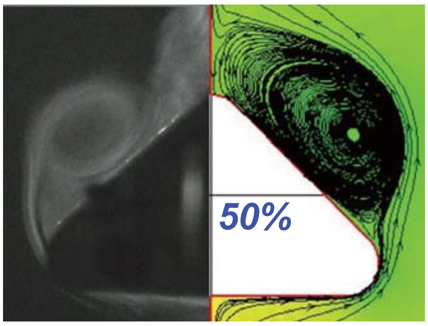
Exp. : C_L=0.30, C_D=0.16

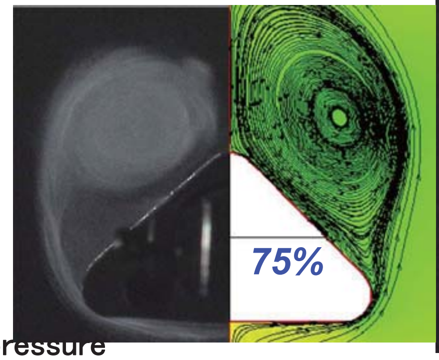
CFD : C_L=0.31, C_D=0.18

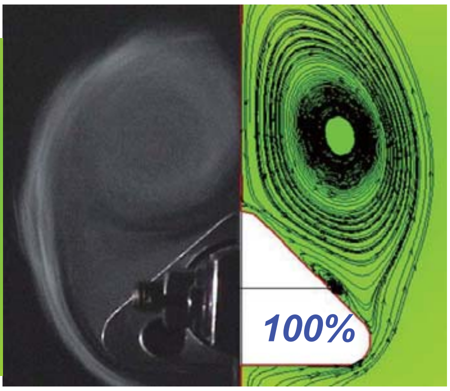


Growth of vortices & surface pressure





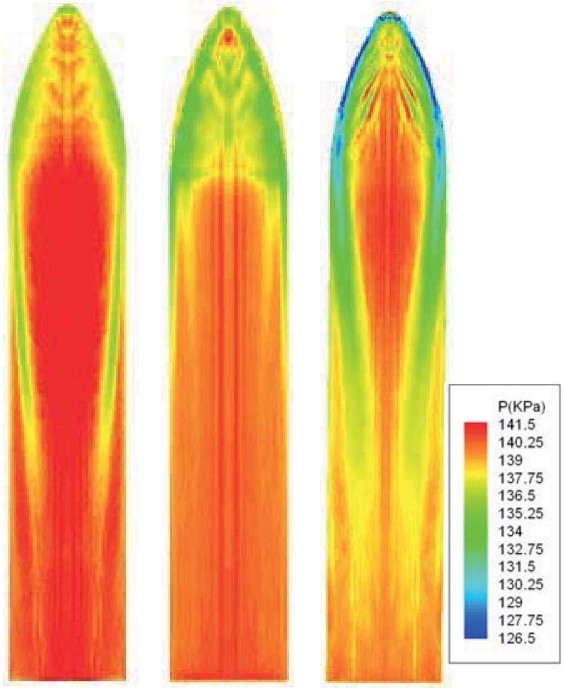




Space Transportation Systems Engineering Laboratory, Kyushu University

Stream lines on Upper Surface

Pressure Contour on Upper Surface



Circle **Square** **Triangle**

⇩ Largest low pressure region

Stream lines on Upper Surface

Circle **Square** **Triangle**

Requirement for CFD

- higher accuracy for estimation of flow separation, aerodynamic forces, etc.
- reduction of computational time

Requirement for EFD

- improvement of wall interference correction, base drag correction, etc.
- spatial measurement
- non-contact measurement

Background .. again

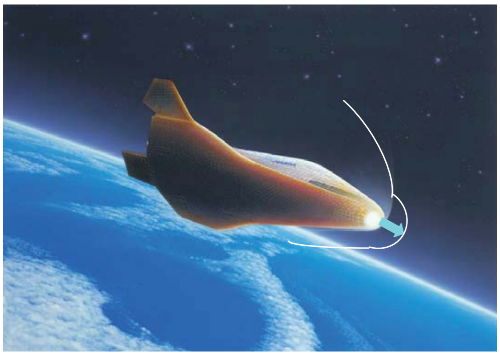
- In designing Reusable Launch Vehicle (RLV)
  → Aerodynamic Heating at reentry and supersonic flight

- Important Problems
  - Aerodynamic Heating at stagnation point
  - Increase of Aerodynamic Heating by transition of boundary layer

Courtesy of JAXA

Necessity of Aerodynamic Heating Reduction

Example of TPS



Courtesy of JAXA

Passive method

Heat absorption

Heat-resistant material

Active method

Mass transfer cooling
- Liquid
- Gas
  - Transpiration cooling
  - Film cooling
  - Cooling by opposing jet

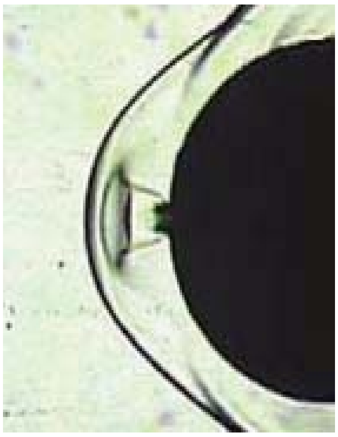
Nose region

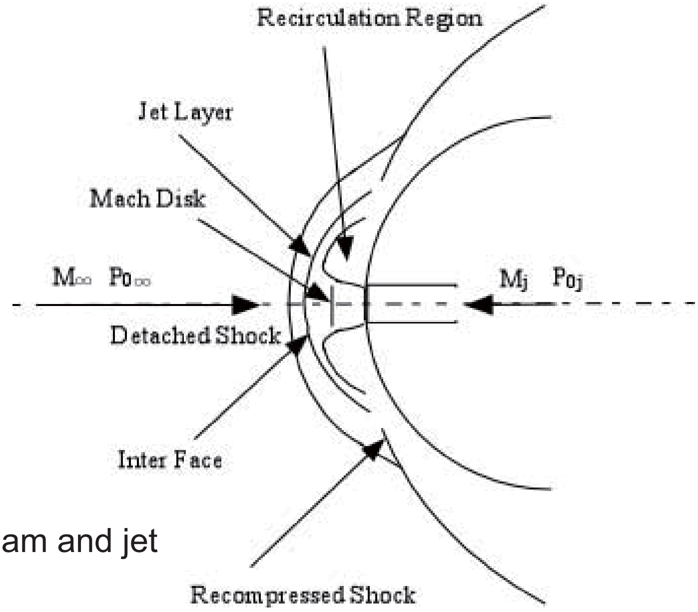
Mechanical method

Intermediate method

Ablation

The Flow Field of the Opposing Jet





The governing parameters

- The Mach numbers of free stream and jet
- The diameter of the jet orifice
- The ratio of the total pressure of jet to that of free stream, PR

$$PR = \frac{p_{0j}}{p_{0\infty}}$$

(the ratio of total pressure of opposing jet to that of free stream)

Experimental Research

- Measurement of the heat flux
- Visualization of the flow field with Schlieren method
- Kyushu univ. supersonic wind tunnel

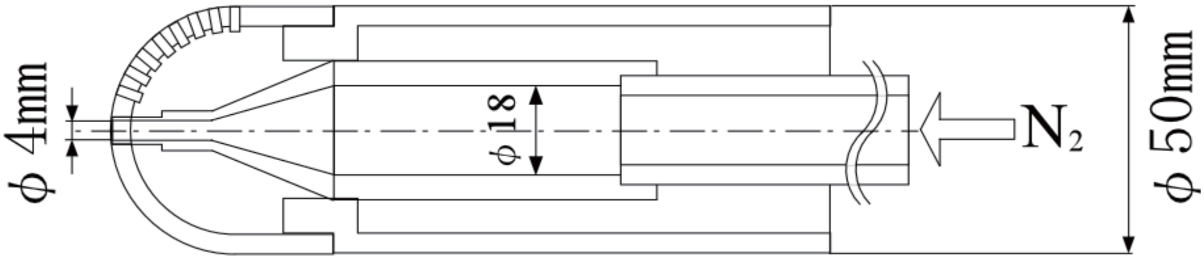
Free stream (average on exp.)

Mach Number	3.96
Stagnation Pressure	1.37 MPa
Stagnation Temperature	397 K, 497 K
Reynolds Number	$2.1\mathrm{x}10^6$

Secondary flow

Mach Number	1.0
Total Pressure Ratio, PR	0.0 - 0.8
Stagnation Temperature	300 K

Experimental model : blunt body



- The diameter of the model : 50 mm
- The diameter of the jet orifice : 4 mm
- Mach number at the jet orifice : 1.0

- The calorimeter gauges are attached at 20 to 90 degrees (every 10 degrees)
- The model is installed into the free stream after the flow becomes steady flow.

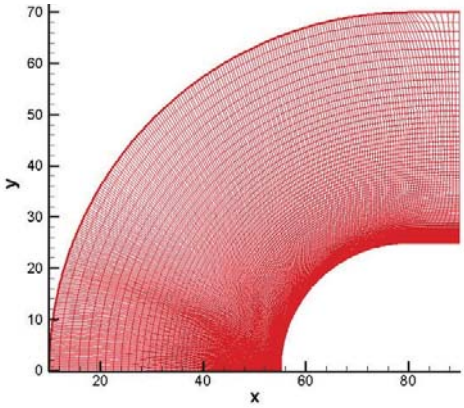
Numerical Analysis Method

- Governing Equation: Reynolds averaged axisymmetric Navier-Stokes equation (RANS)
- Time Integration : LU-ADI method
- Convection Term: AUSM-DV scheme with MUSCL interpolation
- Viscous Term: 2nd-order central difference scheme
- Turbulence model : k-ω two equation model
  - C_μ term is introduced in order to prevent the excessive generation of *k* in collision region. Craft et.al(1996).

Flow Conditions and Grid for supersonic flow

<Main flow>	
Mach number	3.96
Total pressure [MPa]	1.37
Total temperature [K]	397
<Secondary flow>	
Mach number	1.0
Pressure ratio	0, 0.4, 0.6, 0.8
Total temperature [K]	300
<Wall condition>	
Wall temperature [K]	295

- Grid (242 × 160)

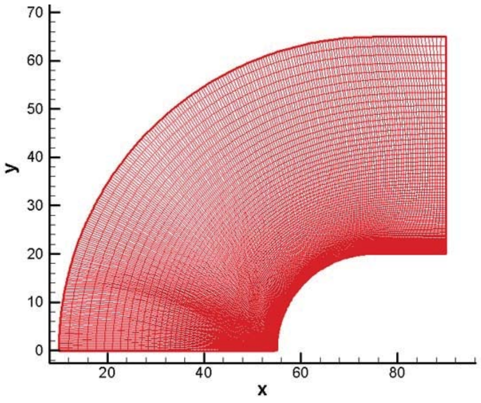


Diameter of body [mm]	50
Diameter of jet orifice [mm]	4

Flow Conditions and Grid for hypersonic flow

<Main flow>	
Mach number	8.0
Total pressure [MPa]	4.5
Total temperature [K]	800
<Secondary flow>	
Mach number	1.5
Pressure ratio	0.0251~0.0859
Total temperature [K]	300
<Wall condition>	
Wall temperature [K]	300

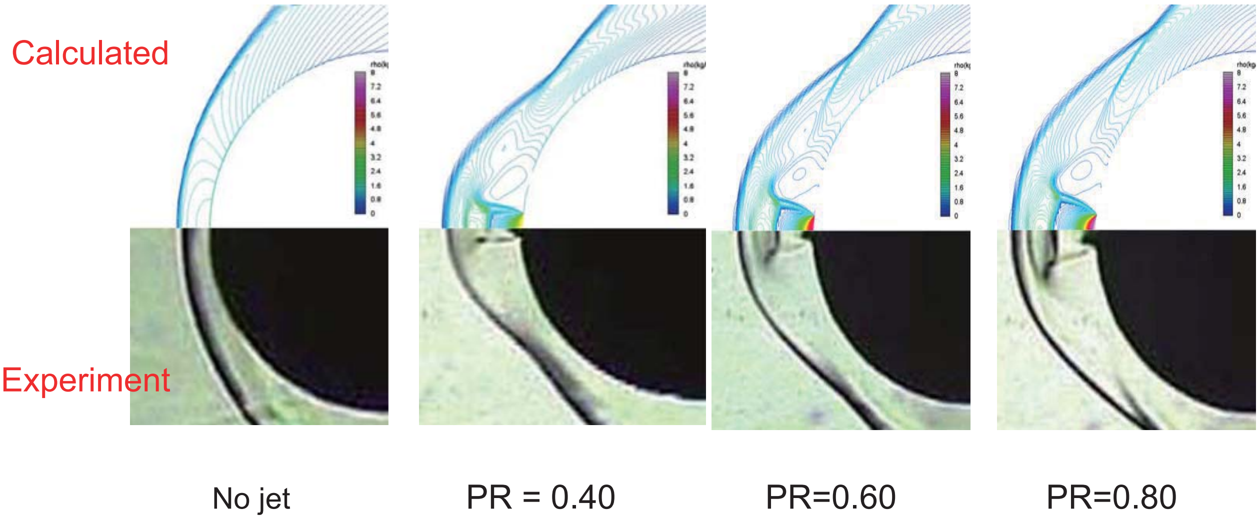
- Grid (240 × 160)



Diameter of body [mm]	40
Diameter of jet orifice [mm]	4.34

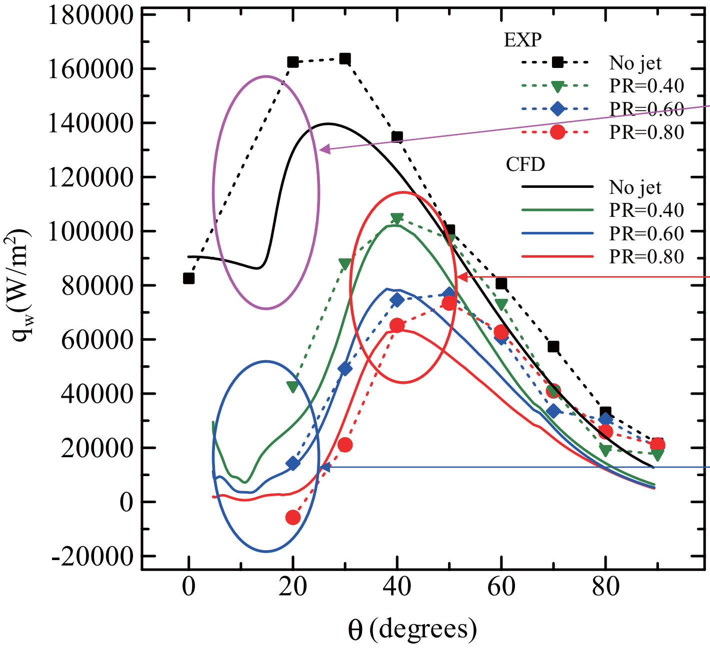
- This flow conditions and grid configuration are based on the experiment conducted by Tokyo Univ. in 1975.

Flow field at each PR (Mach 3.96)



Space Transportation Systems Engineering Laboratory, Kyushu University

Comparison of Heat Flux (Mach 3.96)



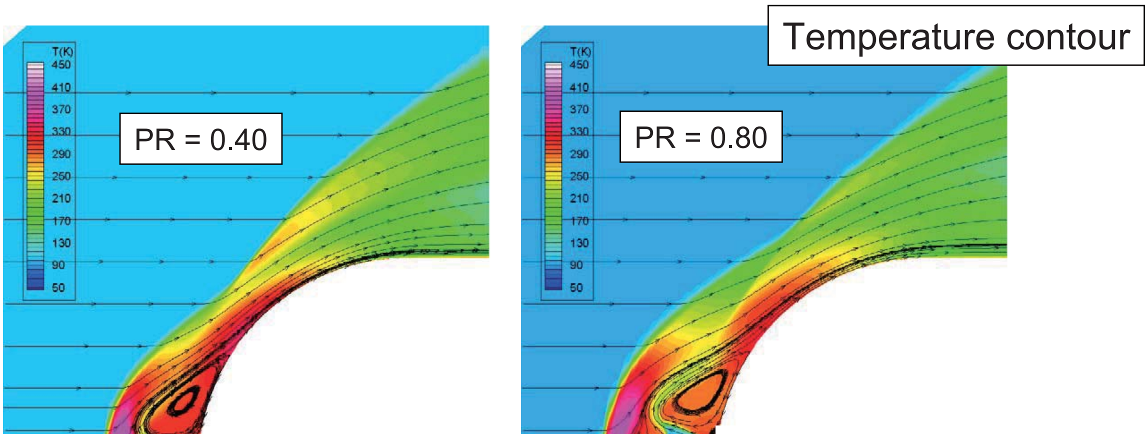
Boundary layer transition occurs

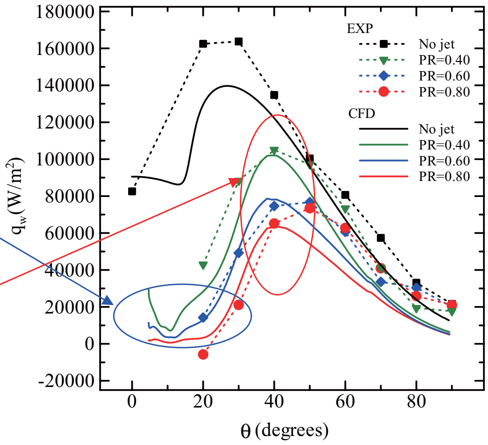
Heat flux increases due to recompressed shock

Heat flux decreases in recirculation region

Space Transportation Systems Engineering Laboratory, Kyushu University

Flow field change by PR (Mach 3.96)

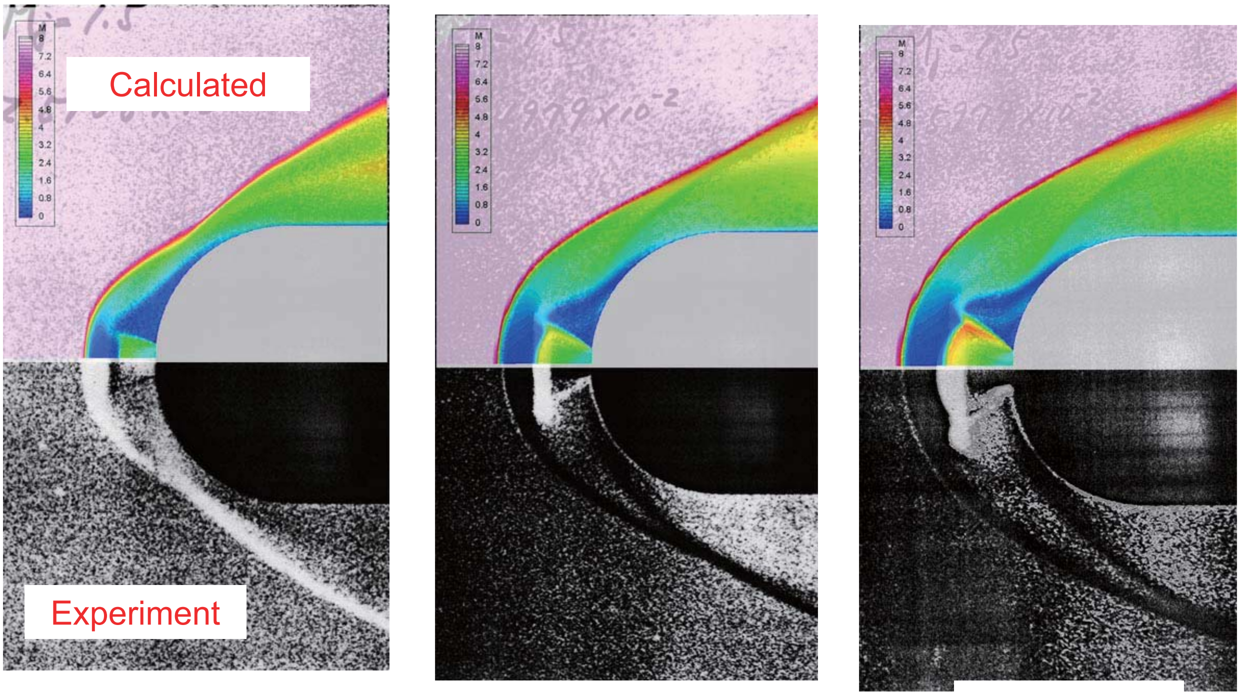




Heat flux decreases
in recirculation region

Heat flux increases
due to recompressed shock

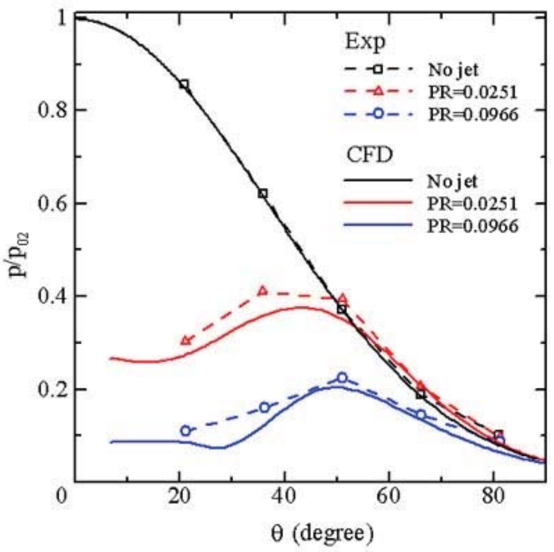
Flow Field at each PR (Mach 8)



PR=0.0251 PR=0.0549 PR=0.0859

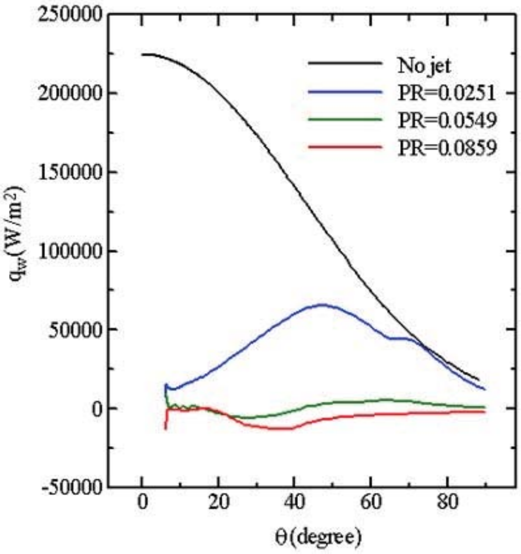
Pressure Distributions (Mach 8)

Pressure Distributions



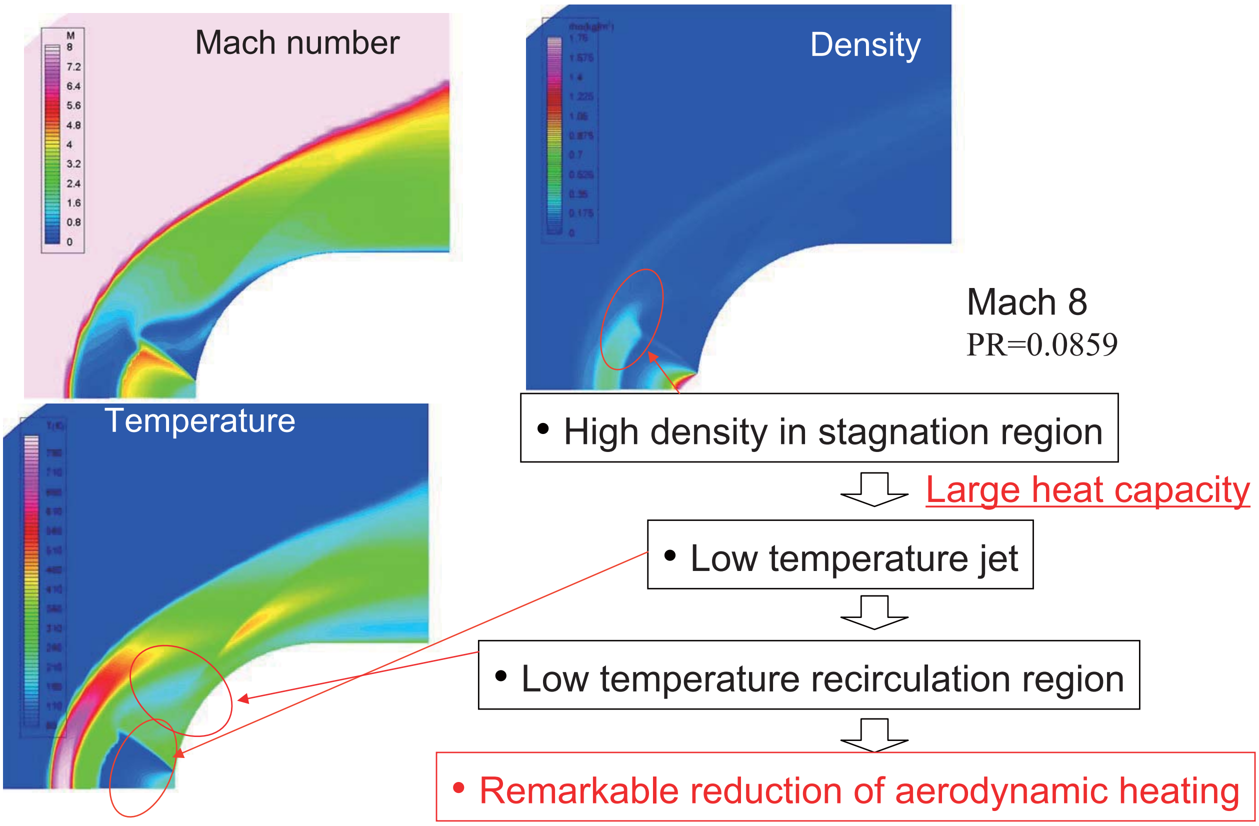
- CFD pressure distribution shows good agreement with experimental measurement.

Heat Flux Distributions (CFD)



- Heat flux decreased more considerably than the case for supersonic.
- Heat flux at each angles decreased as PR increases.

Understanding the flow field



Mach 8
PR=0.0859

- High density in stagnation region

Large heat capacity

- Low temperature jet
- Low temperature recirculation region
- Remarkable reduction of aerodynamic heating

Consideration

- The opposing jet is useful to reduce aerodynamic heating in supersonic and hypersonic flow.
- To understand the mechanism of reducing aerodynamic heating by the opposing jet, detailed flow field should be clarified.
- CFD is very powerful tool to understand the flow field, but has to be validated.

Background .. Once again

Development of scram-jet engine is now in progress as a propulsion system of hypersonic transports and space planes.

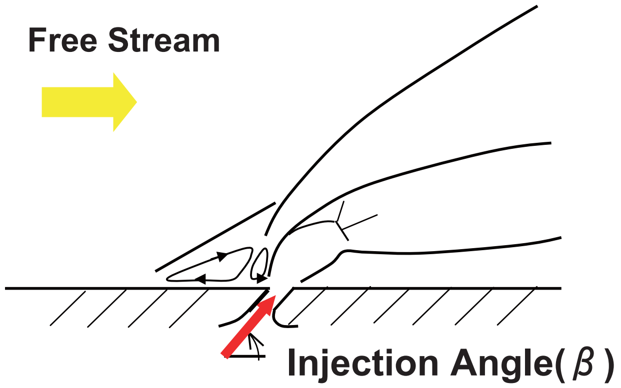
⇩

･In scram-jet engine, the speed of air is very high. Hence rapid mixing and combustion of air and fuel is required. (Air residence time in combustor is 10^{-3}～10^{-4} sec)

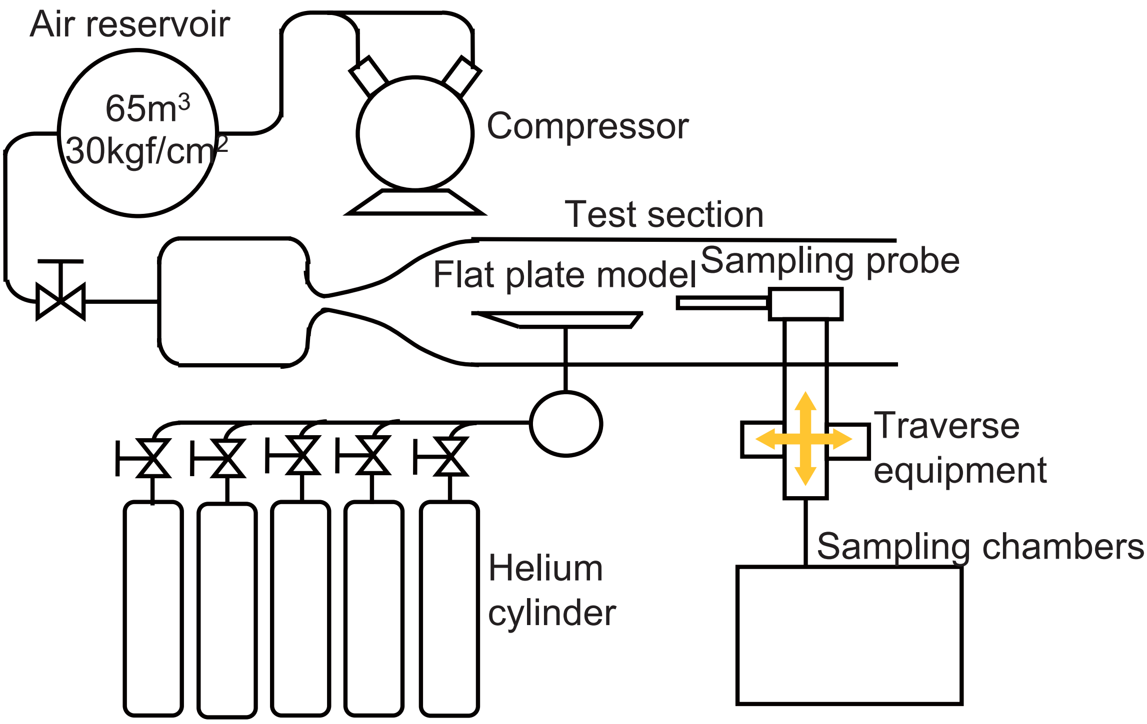
･At high Mach number, suppression of development of shear layer occurs and it makes mixing of air and fuel difficult .

Objective of the present study

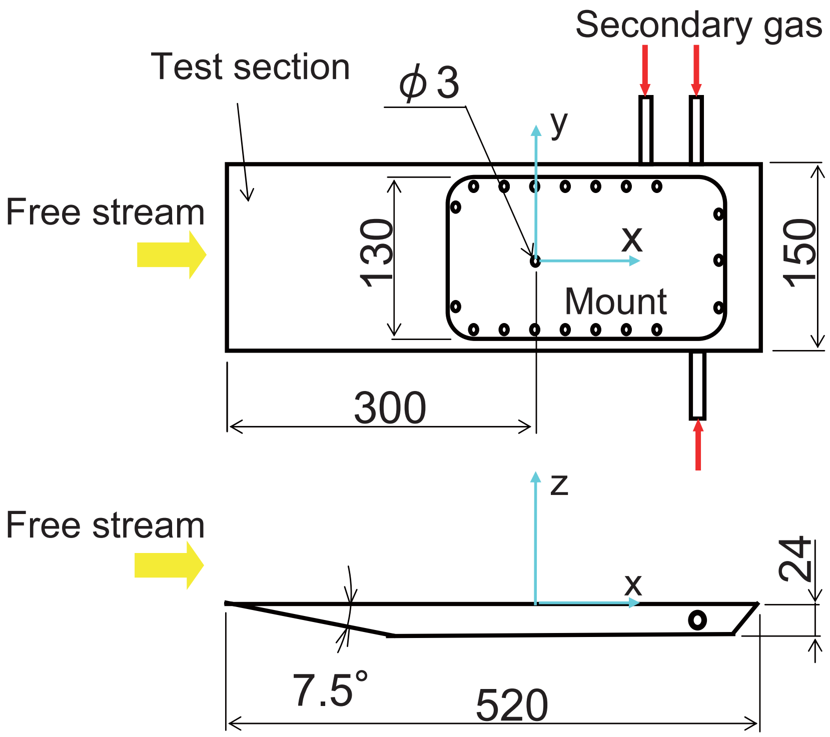
Investigation of the effect of the injection angle β of three-dimensional circular nozzle on supersonic mixing flow



Schematic of experimental facility



Schematic of the experimental model



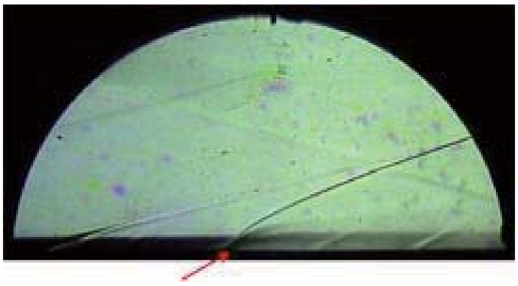
Space Transportation Systems Engineering Laboratory, Kyushu University

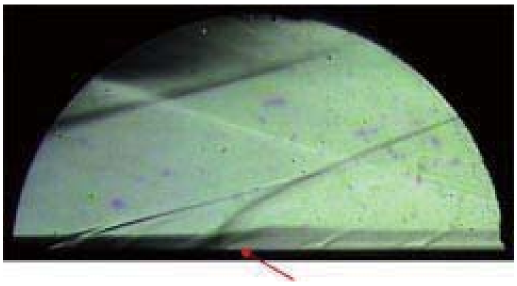
Experimental conditions

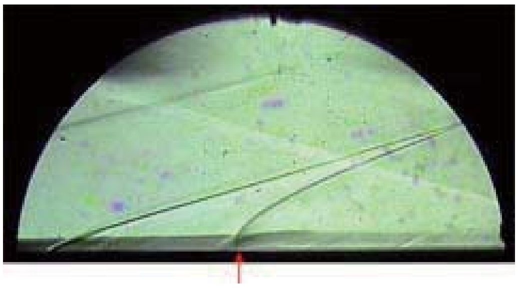
Free stream	Gas	Air
	Mach number	3.76
	Total pressure	1. 12 MPa
	Total temperature	286.9 K
Secondary gas	Gas	Helium
	Mach number	1.0
	Total pressure	0.40 MPa
	Total temperature	286.9 K

Space Transportation Systems Engineering Laboratory, Kyushu University

Flow visualization by Schlieren method


(a) $\beta = 30^\circ$


(c) $\beta = 150^\circ$

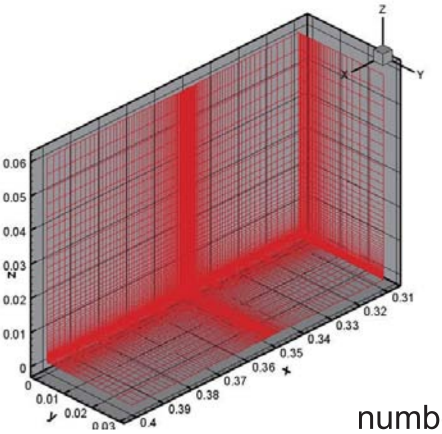

(b) $\beta = 90^\circ$

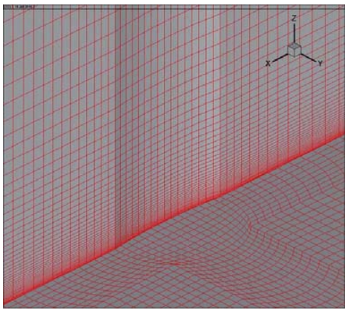
- separation shock wave
- bow shock wave

As injection angle β becomes large, separation region becomes wider and bow shock wave becomes stronger.

Numerical method

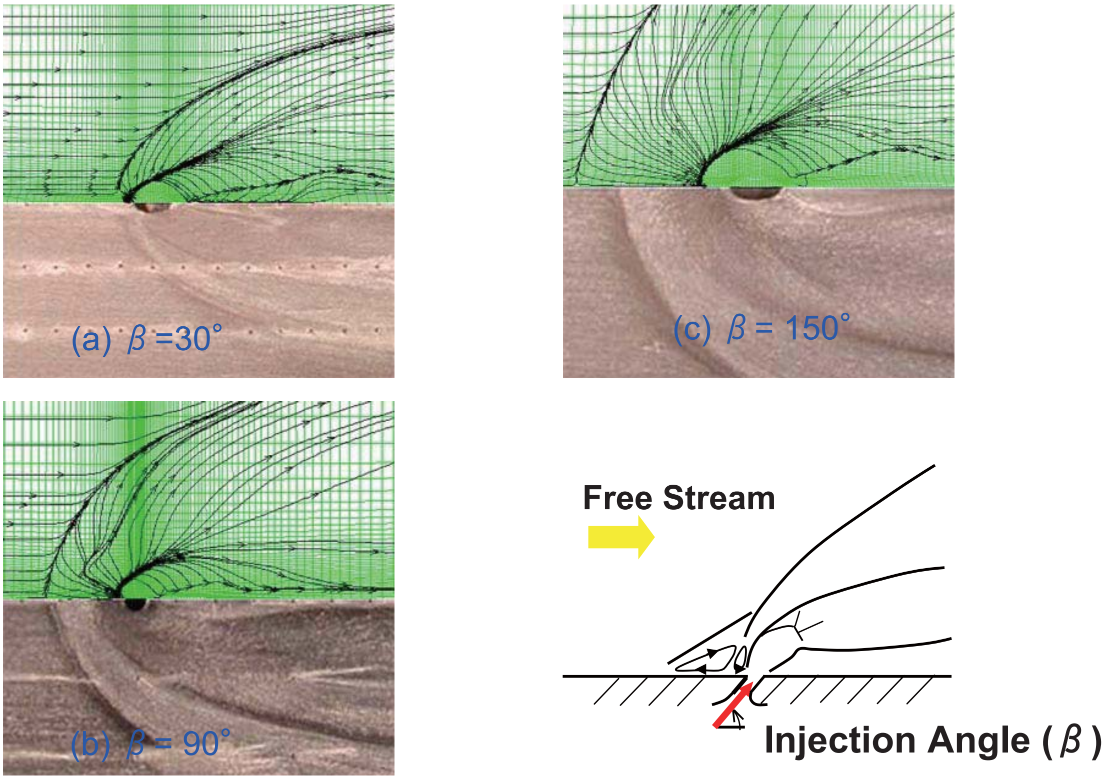
Governing equations :	Reynolds averaged 3D full N-S
Convective terms :	AUSM-Plus scheme
Viscous terms :	2nd order central difference
Time integration :	LU-ADI method
Turbulence model :	k-ω two equation model with Low Reynolds number effect



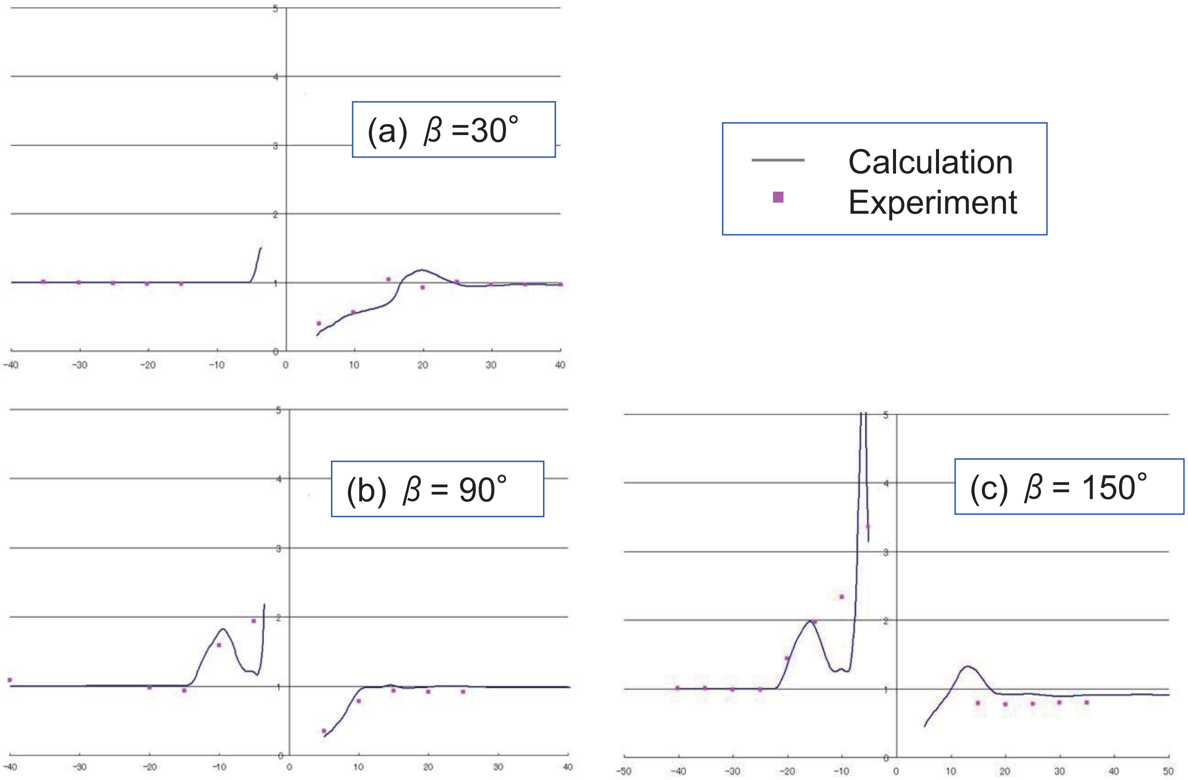


number of grid point : 116×42×75

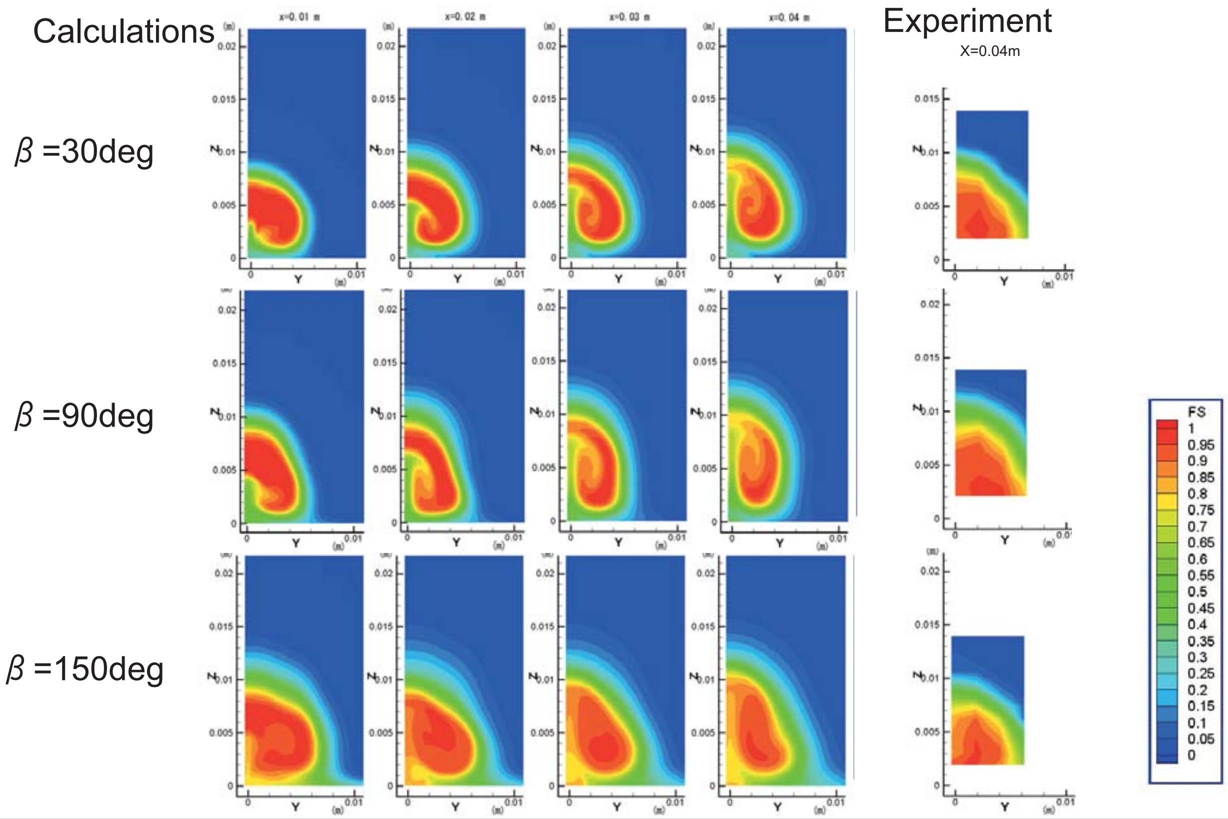
Stream line on surface



Wall pressure distribution at y=0

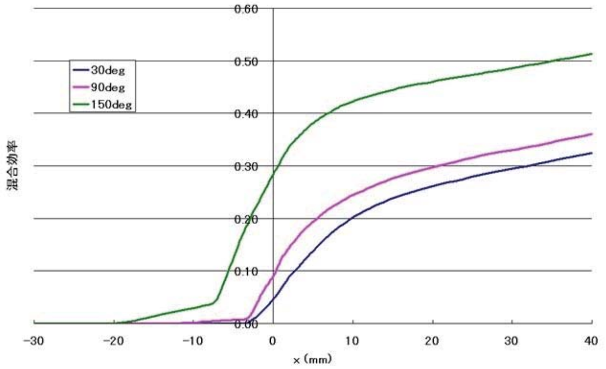


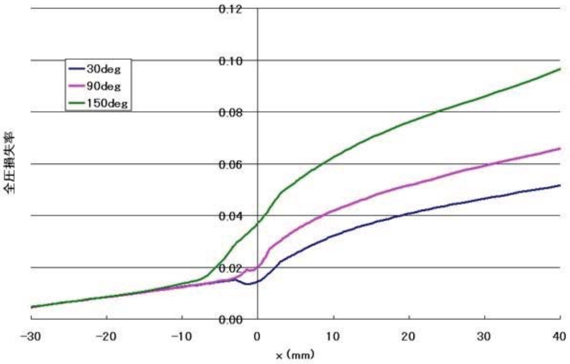
Volume fraction distribution



Space Transportation Systems Engineering Laboratory, Kyushu University

Mixing efficiency and Total pressure loss





$$\eta(x) = \frac{\int_A \frac{\rho u f_{H_2}}{\Phi} dA}{\int_A \rho u f_{H_2} dA}$$

$$\Phi = \begin{cases} 1 & (\text{if } \phi(x,y,z) \le 1) \\ \phi(x,y,z) & (\text{if } \phi(x,y,z) > 1) \end{cases}$$

$$\phi(x,y,z) = 35 \times \frac{f_{H_2}}{1 - f_{H_2}}$$

Stoichiometric mass ratio ... Hydrogen : Air = 1 : 35

A : Cross section of test section
f_{H_2} : Mass fraction of hydrogen
ϕ : Local equivalent ratio in the cross section dA

$$\pi(x) = 1 - \frac{\int_{A(x)} \rho u p_t dA}{\int_{A_i} \rho u p_t dA + \int_{A_j} \rho u p_t dA}$$

p_t : Local total pressure
A : Cross sectional area at x
A_i : Cross sectional area at the entrance of the test section
A_j : Cross sectional area of the injector

Space Transportation Systems Engineering Laboratory, Kyushu University

Considerations

1) Supersonic mixing phenomena can be fairly simulated not only in separated region but also in mixing.
2) Flow characteristics for injection angles shows good agreements between CFD and experiments.
3) Detailed measurement of flow field and reliability of CFD should be improved.

Conclusions

EFD and CFD in a university,

Research

Education