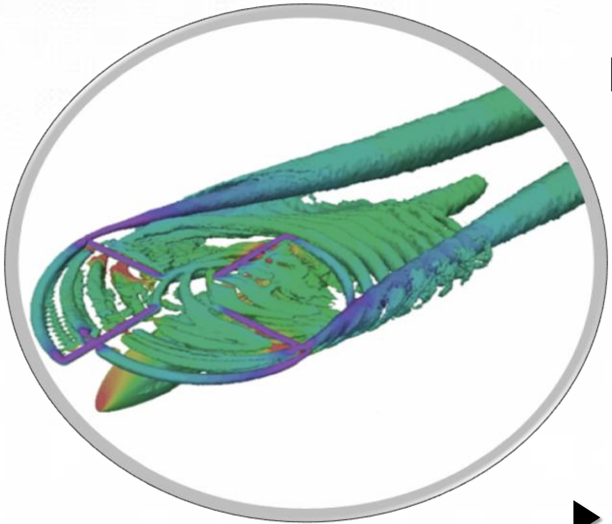
공기역학 해석을 위한 OpenFOAM 활용

Pusan National University
Dept. of Aerospace Engineering
Applied Aerodynamics and Design Lab.
오세종

Applied Aerodynamics & Design Laboratory

Contents



AADL 소개

- Members
- Research fields

Introduction of AADL

Professor

- Professor Sejong oh
  - High Lift Devices
  - Grid formation approach
  - Numerical analysis method
  - Helicopter aerodynamics

Students

- Student member
  - 1 post-doctoral researcher
  - 3 Ph.D Candidates
  - 2 master course students

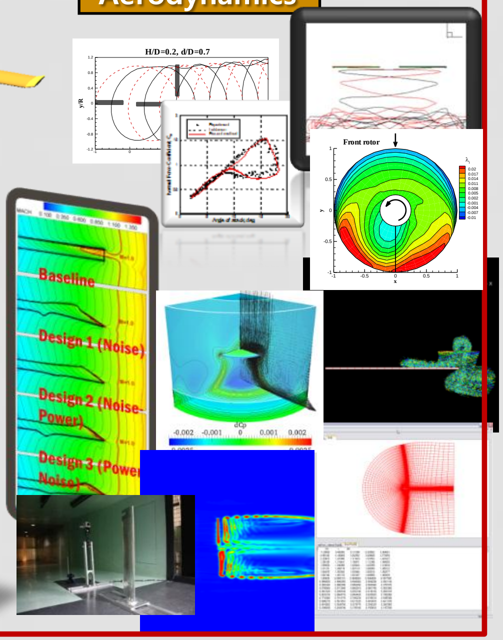
Introduction of AADL

Research fields

Research Interests

MDAO

Applied Aerodynamics



Aviation Safety

2차원 익형의 착빙에 따른 공력 성능 해석

- 연구 목적 및 방법
- 연구 결과

Introduction

Aircraft Icing

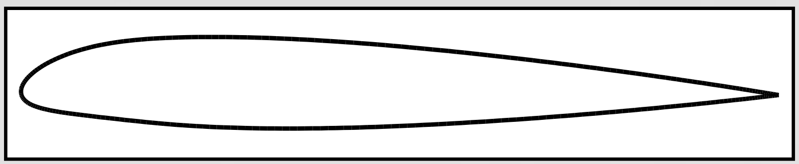
- Aircraft icing : Super-cooled liquid water droplets impact and freeze on the aircraft surface
  - Aircraft, helicopter, wind turbine blade, ship, and power line
  - Major cause of aircraft accidents*
  - Aircraft Owners and Pilots Association(AOPA) report : 1990～2000, 3230 accidents are concerns with weather conditions
  - 388 accidents(12%) are related to aircraft icing phenomenon
  - Accumulated ice changes surface roughness, and deforms the wing shapes
  - Degradation of left, drag and moment coefficient

Applied Aerodynamics & Design Laboratory

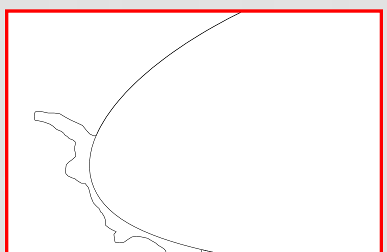
Introduction

Aircraft Icing

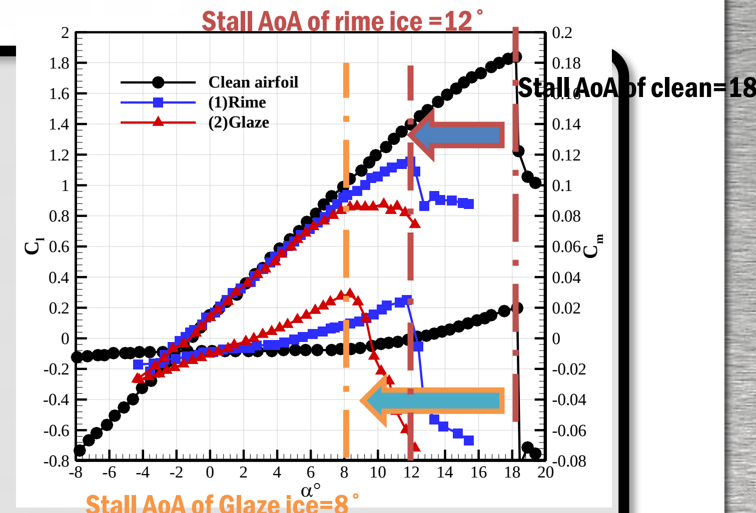
- Main wing icing can reduce aircraft performance and safety
  - Maximum lift, stall margin are reduced, and the drag is dramatically increased
- For the flight safety, researches on related icing phenomenon is now in progress

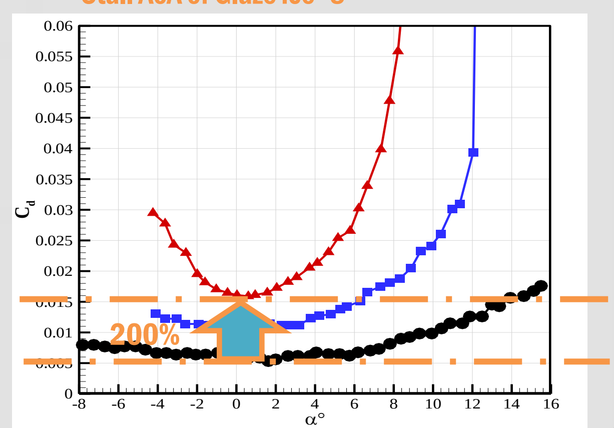


V_∞=77m/s, T_∞=-25.3℃, LWC=0.55g/m³, MVD=30μm, t=10m, α=2°



V_∞=90m/s, T_∞=-6.2℃, LWC=0.85g/m³, MVD=20μm, t=11.3m α=5°

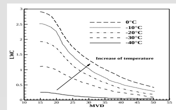




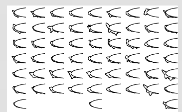
Methods

Scope of this study

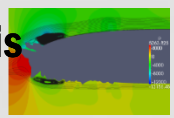
Selection of icing conditions
(FAR Part 25 Appendix c)



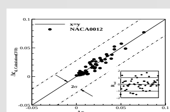
Acquiring ice accretion shapes
(NASA icing wind tunnel)



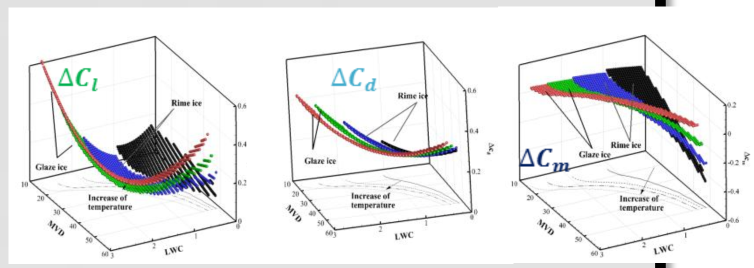
Aerodynamic performance analysis
(OpenFOAM)



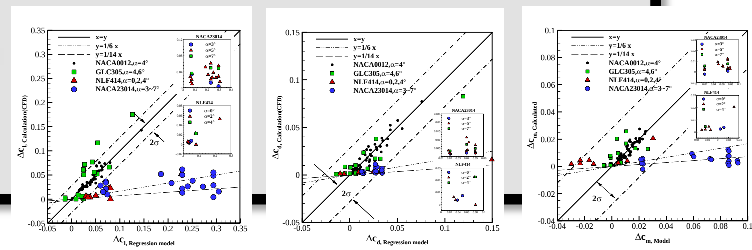
Response Surface Methodology
(2^{nd} order polynomial equations)



Relations between meteorological parameters and degradation of aerodynamic performance



Analyze the effects of the airfoil shape on degradation of aerodynamic performance

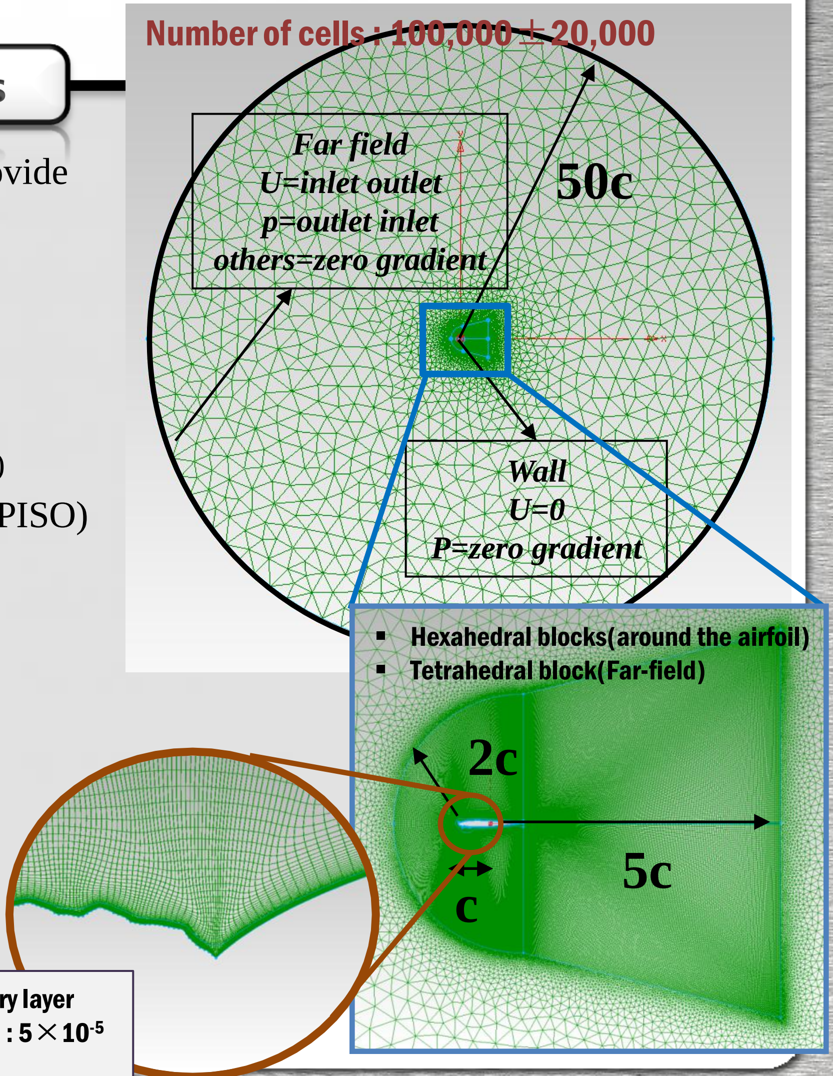


Applied Aerodynamics & Design Laboratory

Methods

Aerodynamic Performance Analysis

- Numerical approach : Icing wind tunnel does not provide aerodynamic performance of iced airfoil
- OpenFOAM : Navier-Stokes equation based aerodynamic solver
  - Flow separation and reattachment due to ice horn
- Unstructured grid
  - To handle the complex geometries(ice accretion shape)
- pisoFoam* : Pressure Implicit Splitting of Operator(PISO)
  - Incompressible, Turbulent flow
  - M_{∞}<0.33, Re>2 X 10^6
- Turbulent : SA, and k-ω SST are compared
  - SA model yields better results
- Steady state assumption
  - Low angle of attack
  - Not massive separation condition

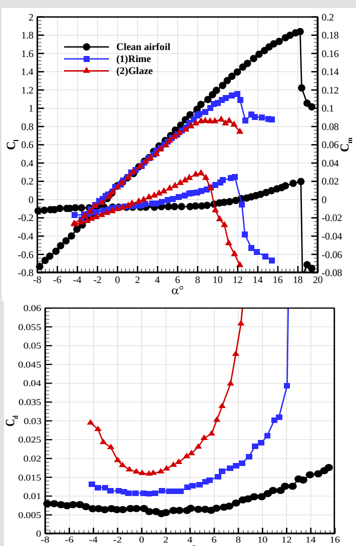


- Capturing boundary layer
- Height of first grid : 5 × 10^{-5}
- y+<2.2
- Growth ratio : 1.1

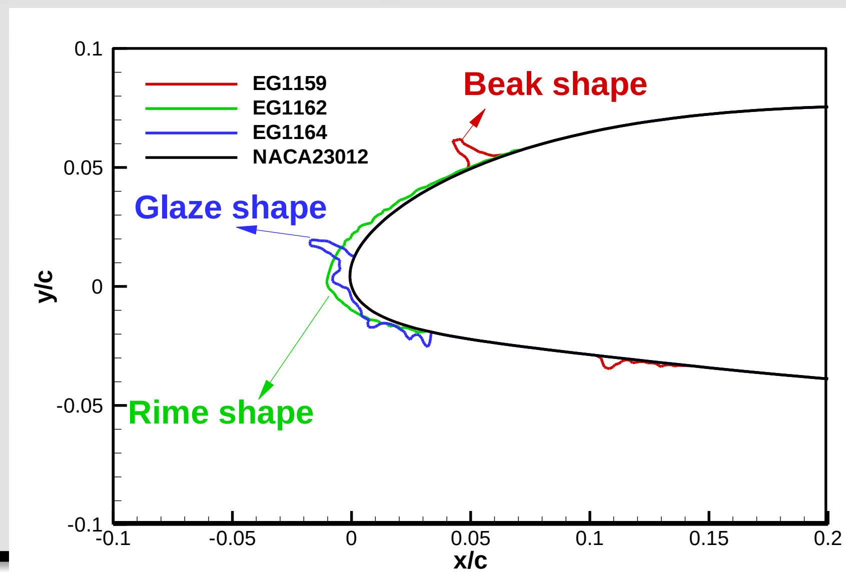
Methods

Validation of aerodynamic solver

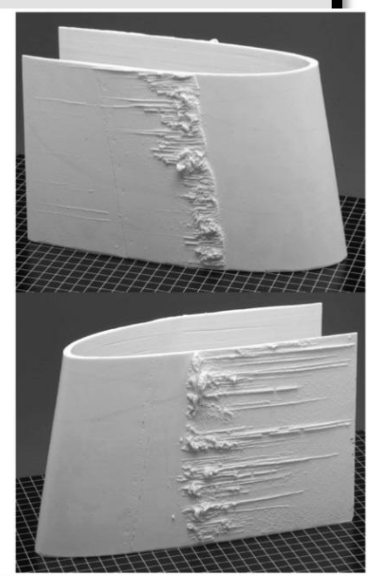
- The results of aerodynamic solver are compared with experimental data*
  - Experimental condition : $M_{\infty} = 0.2$, $Re = 1.6 \times 10^6$
- Ice accretion shaped is tested in the dry wind tunnel
  - NASA IRT ice shapes → Casting model → Dry wind tunnel test
- NACA23012(clean), rime, and glaze ice shape



[Lift, drag, pitching moment coefficients]



[Ice accretion shapes]

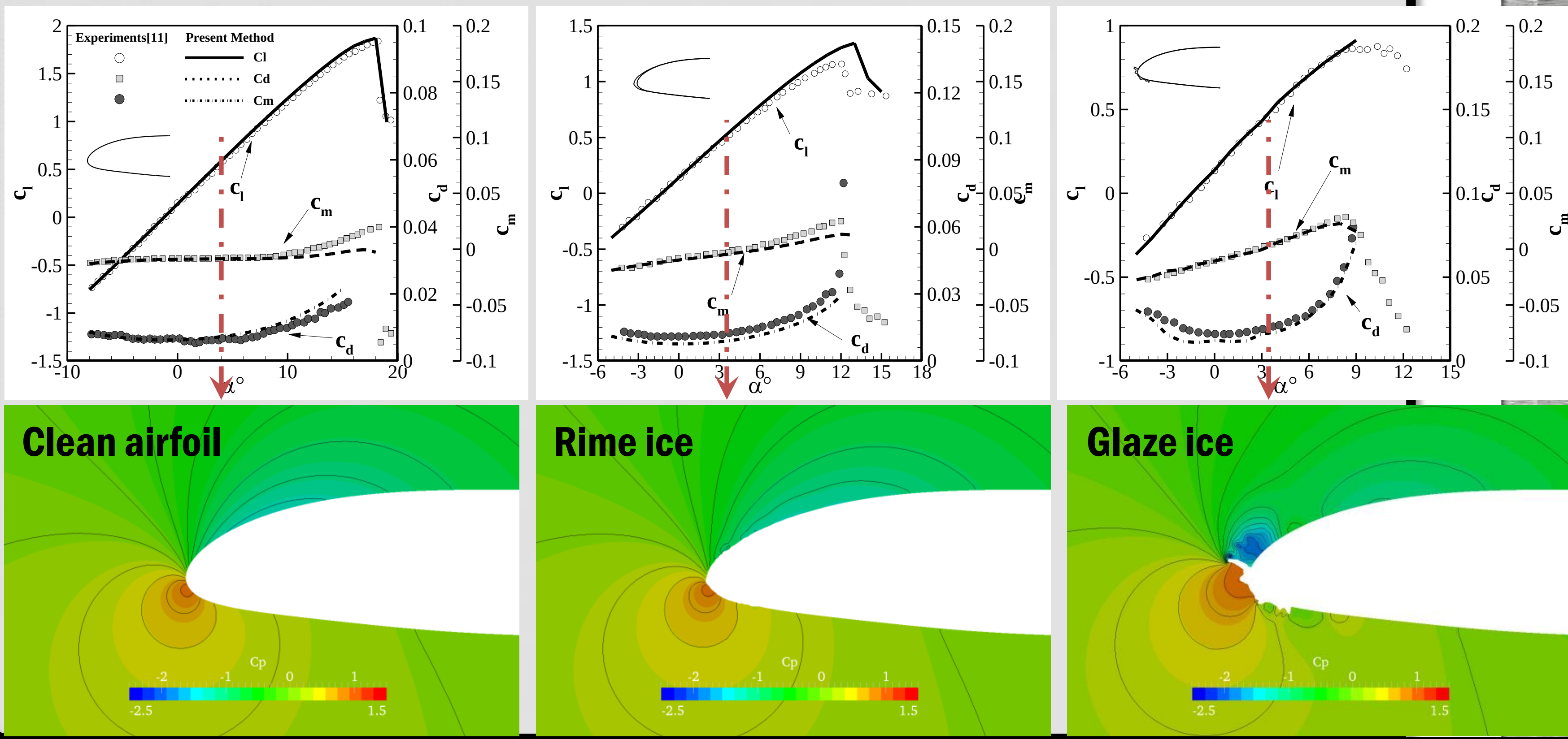


[Casting Model]

Applied Aerodynamics & Design Laboratory

Methods

Validation of aerodynamic solver



- *Flow conditon :M∞=2.0,Re = 15.9 × 10⁶,α=4°*

- *Icing condition : α=2° V∞=77.2m/s, T∞=-22.2° C, LWC=0.55g/m³, MVD=30μm, time=10min*
- *Flow condition : M_∞=2.0,Re = 15.9 × 10⁶,α=4°*

- *Icing condition : α=5° V_∞=90m/s, T_∞=-2.2° C, LWC=0.85g/m³, MVD=20μm, time=11.3min.*
- *Flow condition : M_∞=2.0,Re = 15.9 × 10⁶,α=4°*

Error at α=4°
c_l : 5%, c_d : 8%, c_m : 2.5%

Error at α=4°
c_l : 5%, c_d : 10%, c_m : 2.5%

Methods

RSM

- The present study employs RSM to efficiently analyze the correlation with obtained meteorological parameters and aerodynamic performance without ice shape parameters
- A 2nd-order polynomial regression model is constructed

$$y_i = \beta_0 + \beta_1 x_1 + \beta_2 x_2 + \cdots + \beta_{11} {x_1}^2 + \beta_{22} {x_2}^2 + \cdots + \beta_{12} x_1 x_2 + \cdots$$

ΔC_l, ΔC_d, ΔC_m

Aerodynamic penalties

$$\Delta c_l = c_{l,clean} - c_{l,ice}$$
$$\Delta c_d = c_{d,ice} - c_{d,clean}$$
$$\Delta c_m = c_{m,clean} - c_{m,ice}$$

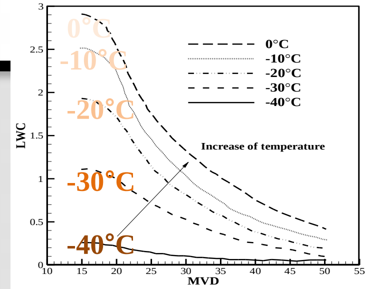
x_1	x_2	x_3	x_4	x_5
V_∞	T_∞	LWC	MVD	Time

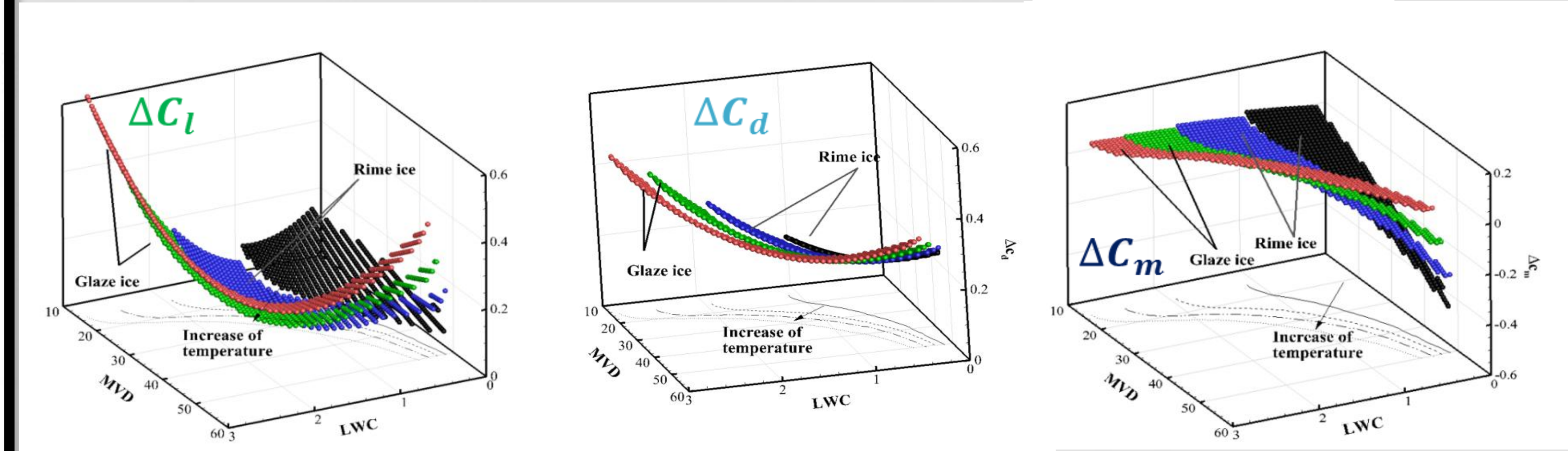
- From the icing parameters(V, T, LWC, MVD, Time), RSM model can predict the aerodynamic penalties
- RSM is composed single airfoil(NACA0012) with various icing conditions
- Various icing conditions(57 cases) including rime and glaze icing conditions are employed

Results

Application results

- RSM model is applied to FAR Part 25, Appendix C condition
- Velocity and icing time are selected as an accident condition*
  - *V=67m/s, Time=6min*



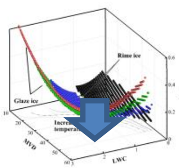


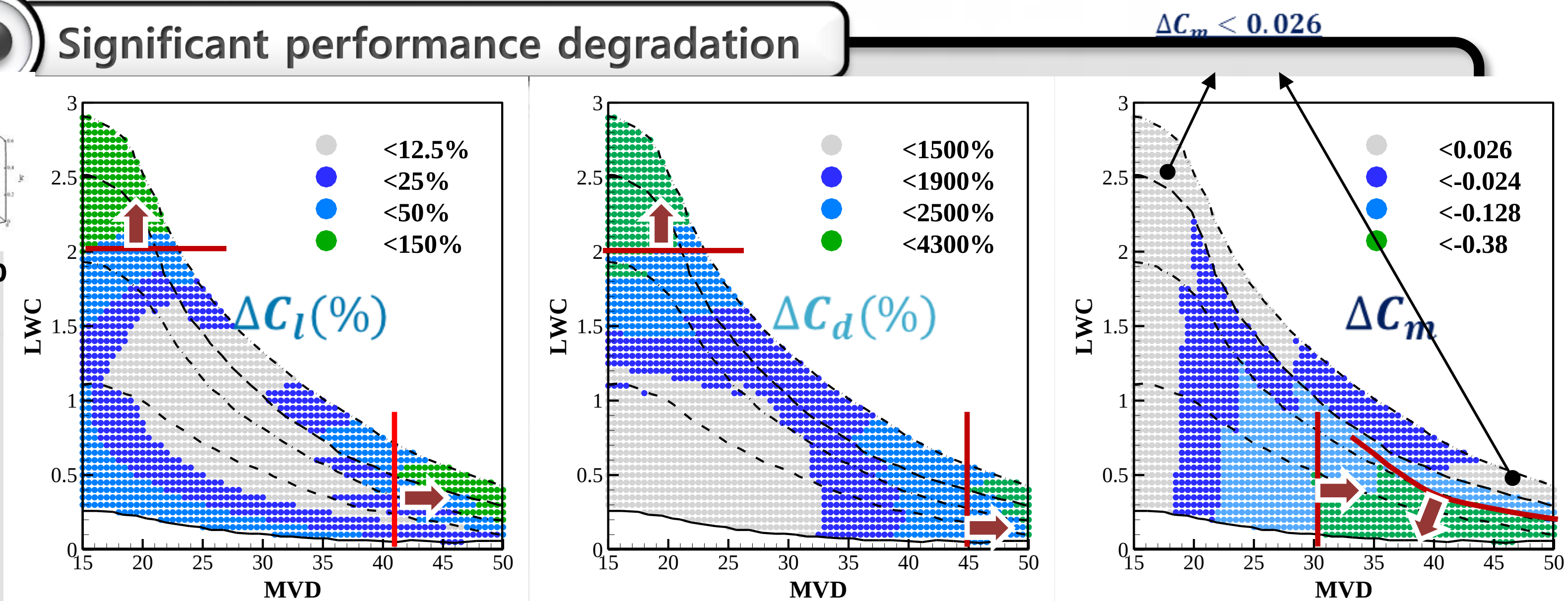
- Drag and lift penalties increased in the glaze ice conditions
  - High LWC, MVD, temperature region
- Moment penalties increased in the rime ice conditions
  - High MVD, and low temperature region irrespective of LWC

Applied Aerodynamics & Design Laboratory

Results

Significant performance degradation


Results of 2D projection



- The regions are divided into 4 such that same level of degradation of lift
- The lift and drag overlap in some areas
  - The drag increases over 2500% in areas very similar to where the lift decreases more than 50%, compared to a clean airfoil
- Moment coefficient is under -0.128 : stall condition of clean airfoil
  - When we use de-icing devices, the moment turn from negative to positive

고정익 날개의 끝단 와류 저감에 관한 연구

- 연구 목적 및 방법
- 연구 결과

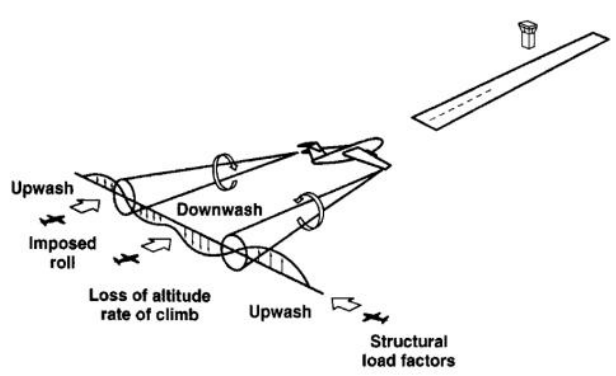
Introduction

Motivation

- Aviation distribution is increasing all over the world
  - capacity of an airport has posed a serious limitation
  - methods to increase the efficiency of airport operations are needed

- The separation time interval
  - It is effective to reduce the separation time intervals between leading aircraft and following aircraft
  - Separation time intervals are limited by intensity and range of the wing tip vortex shed by leading aircraft
    - ✓ Vortex wakes of leading aircraft are persistent and can be hazardous
    - ✓ Following aircraft must delay their arrival until the vortex wakes have decayed to a harmless level
    - ✓ Typically it takes more than 3 minutes*

- Wingtip vortex attenuation study is needed



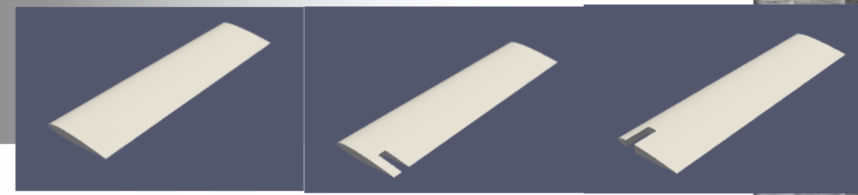
▲ Possible encounters with lift-generated wake by a following aircraft*

Based on standard separation, constant airspeed of 120 knots (Small), 140 knots (757/Large) & 160 knots (Heavy).

Following Aircraft	Leading aircraft			
	Heavy	B-757	Large	Small
Heavy	90 -	106 -	72 -	94 -
	90	90	56	56
Large	129 -	103 -	64 -	86 -
	145	103	64	64
Small	150 -	150 -	90 -	75 -
	188	171	120	75

▲ Approximate Separation time intervals *

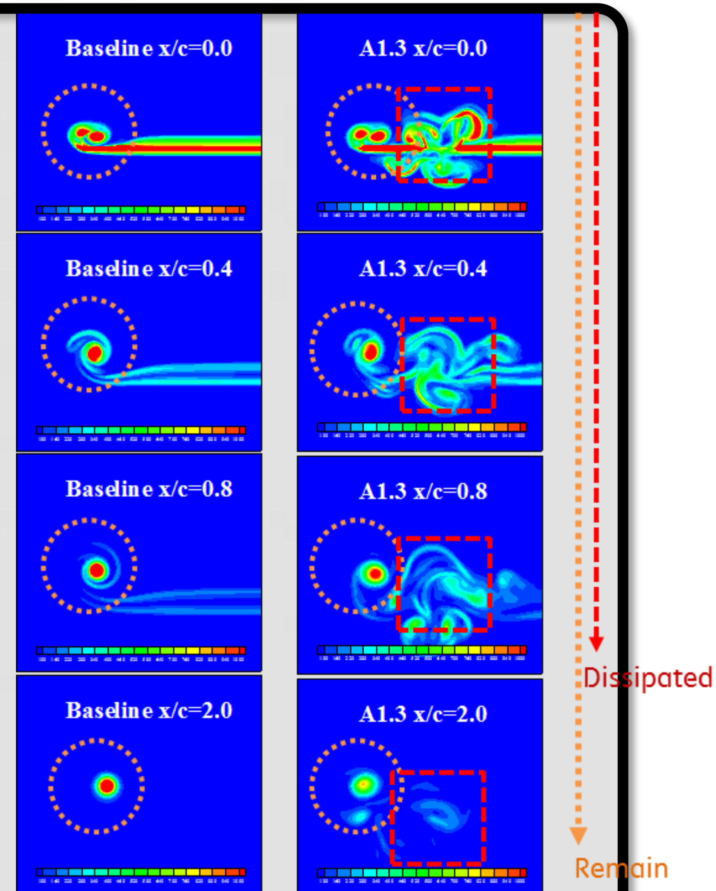
Introduction



▲ Baseline, TE chipped, TIP chipped wings

Objective

- TE Chipped wing
  - Vortex occurred at chip and wing tip begin to mix at end of wing
  - Once wing tip vortex is fully developed, it's hard to dissipate
- TIP Chipped wing
  - Making a chip at tip to prevent occurring wing tip vortex
  - Wing tip vortex cuts during the process of generation
- Goal of this study
  - Comparing vortex attenuation effect and aerodynamics performance between TE chipped wing and TIP chipped wing
  - Confirming vortex attenuation effect according to shape of TIP chipped wing
  - Suggesting the optimal model for fixed wing and rotating wing



Methods

Computational grid

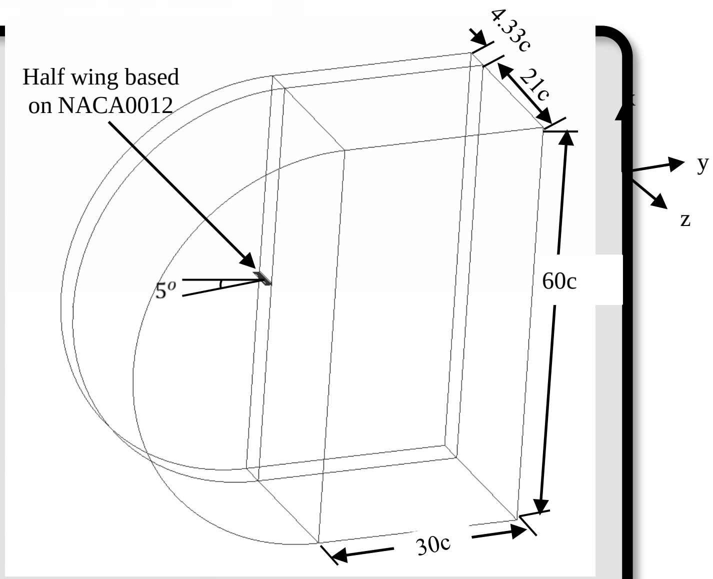
- Wing Geometry
  - Rectangular wing with truncated wingtip
  - Wing profile : NACA0012
  - Chord length : 0.203m
  - Span : 0.897m
  - Incidence angle : 5°

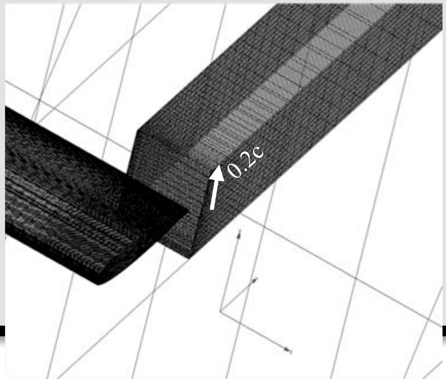
- Computational Grid
  - Spanwise symmetry
  - Topology : C-H type
  - Domain extension : $-30 \leq \frac{x}{c} \leq 30$

    $-30 \leq \frac{y}{c} \leq 30$

    $0 \leq \frac{z}{c} \leq 4 \times \frac{b}{2}$

- Clustering with equal spacing fine grid
  - rear field of wingtip region, $\pm 0.2c$



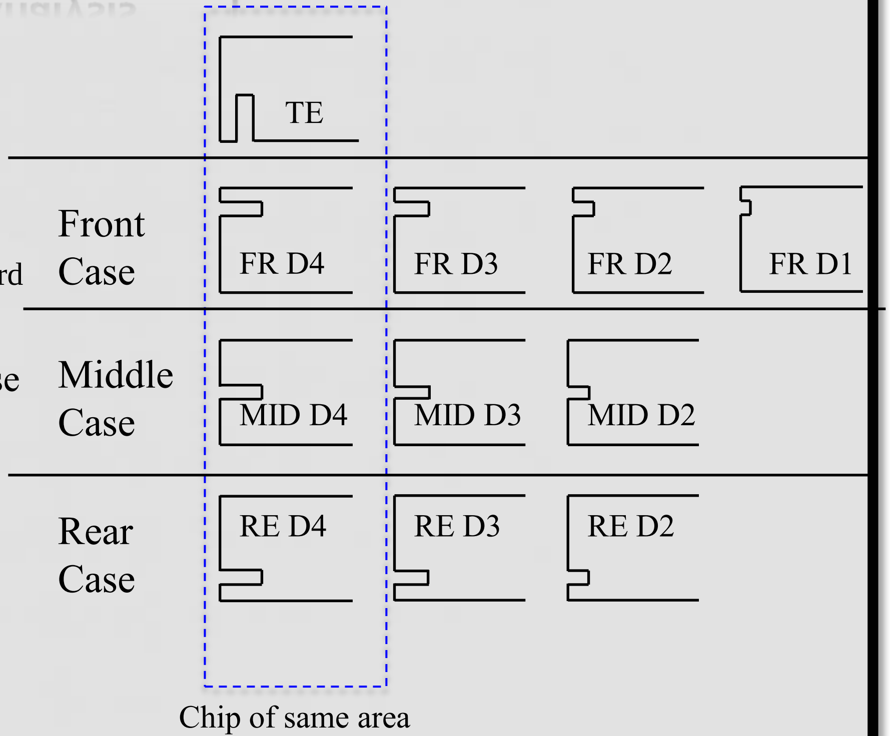


Applied Aerodynamics & Design Laboratory

Introduction

Definition of shape for Analysis

- Location Variation
  - Front, Middle, Rear
- Depth Variation
  - D1, D2, D3, D4
  - Ex) D1 means the depth of chip is 0.1chord
- D4 Case's Chip area is same with TE Case
- 11 parameters was analyzed



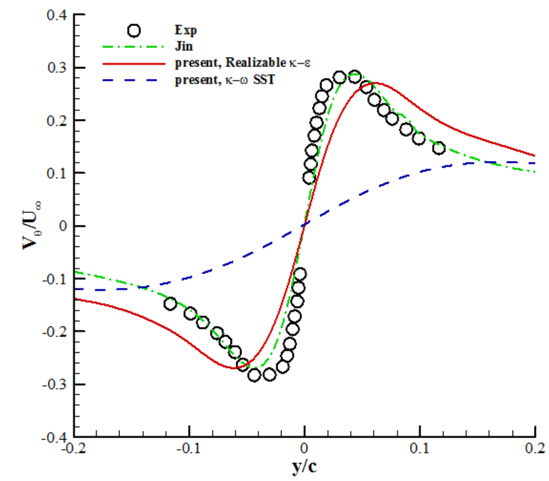
Results

Numerical Validation

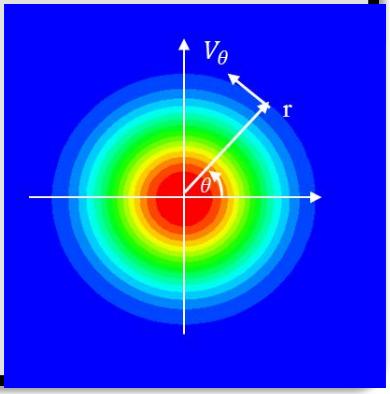
- Swirl velocity distribution
  - V_θ of core radius
  - To obtain more accurate results, using the average value of two data which obtained by extracting
    - ✓ 0°, 90° with maximum point of vorticity magnitude as the center

- Comparison with experimental result and Numerical result
  - Experimental result : Devonport*
  - Numerical result : Jin** - Navier-Stokes, RSTM
  - OpenFOAM's turbulence model : Relizable κ-ε, κ- ω SST

- Results
  - 20% error from the experimental value
    - ✓ reliable results than the 32% error of Jesse result
    - ✓ These value of error is insufficient for accurate result, but enough for confirming the tendency of flow
  - Improved result is obtained by adopting the betterment grid



▲ Swirl velocity distribution for validation at x/c=10

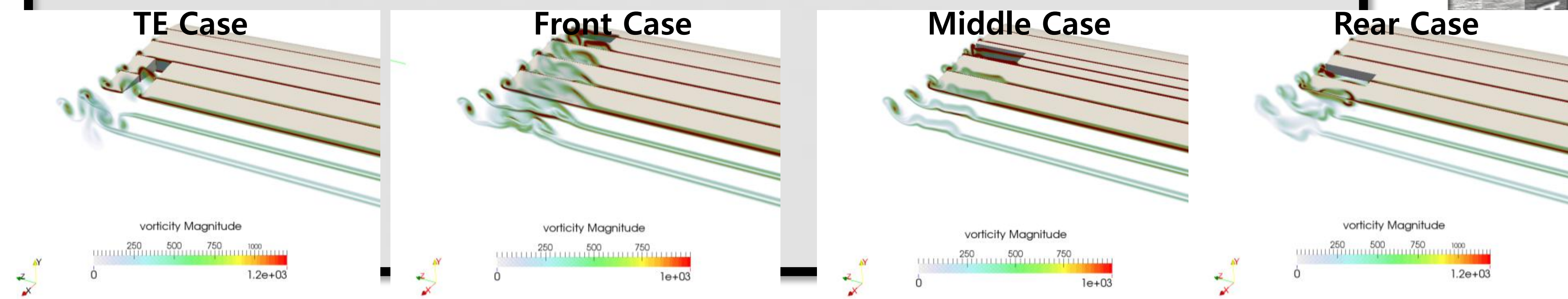


▲ Vorticity magnitude contour

Results

Effect of the Chip Location

- TIP Chipped wing, D4 Case & TE Chipped wing
  - Chipped area and chip AR are same
- TE Chipped wing
  - Vortices occurred at chip and tip begin to mix at end of wing
    - ✓ The vortices grow along the freestream, without anymore disturbance
  - Vortices of two different directions generated by the chip
- Tip Chipped wing
  - Wing tip vortex cuts during the process of generation
    - ✓ Vortex generated by the chip inflates temporarily because flow faces end of chip
  - Vortices of two different directions generated by the chip

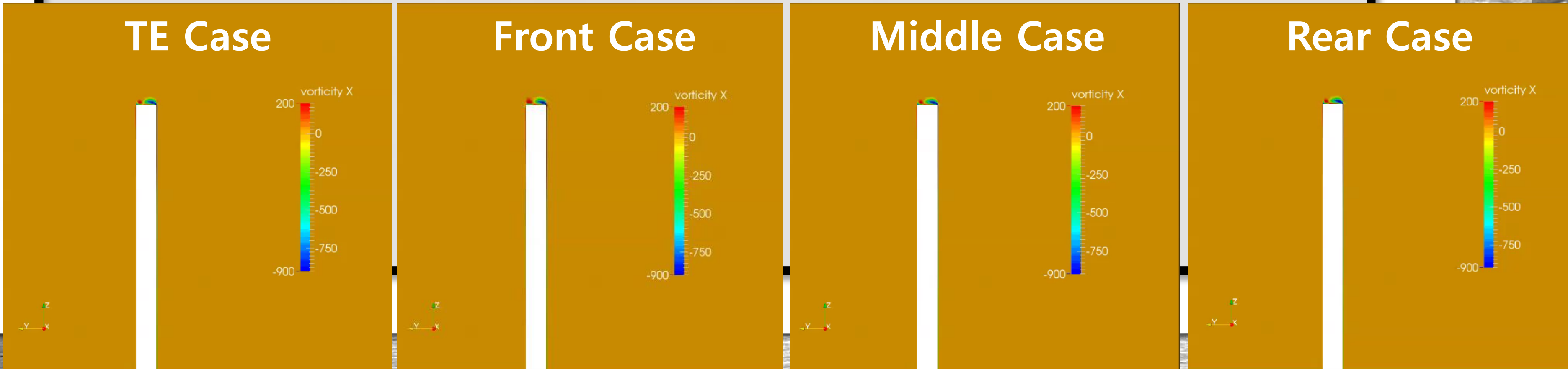


▲ Axial vorticity contour for TE Chipped wing and Tip Chipped wing at naer-field of wing

Results

Effect of the Chip Location

- TIP Chipped wing, D4 Case & TE Chipped wing
  - Chipped area and chip AR are same
- TE Chipped wing
  - Vortices occurred at chip and tip begin to mix at end of wing
    - ✓ The vortices grow along the freestream, without anymore disturbance
  - Vortices of two different directions generated by the chip
- Tip Chipped wing
  - Wing tip vortex cuts during the process of generation
    - ✓ Vortex generated by the chip inflates temporarily because flow faces end of chip
  - Vortices of two different directions generated by the chip



▲ Axial vorticity contour for TE Chipped wing and Tip Chipped wing at near-field of wing

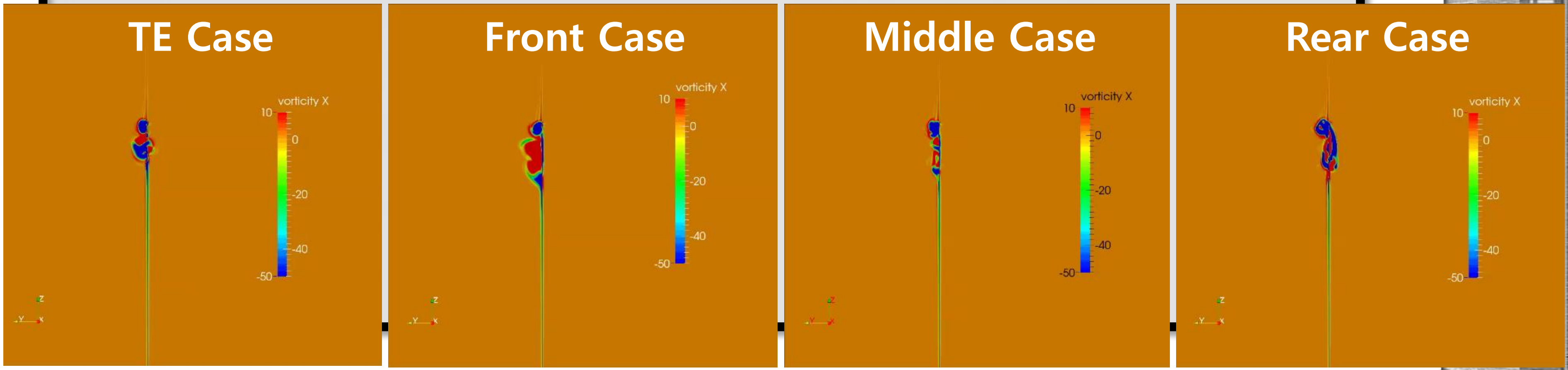
pplied Aerodynamics & Design Laboratory

Results

Effect of the Chip Location

- TE Chipped wing
  - The two sub vortices generate relatively close and they are mixed rapidly
  - After sub vortices become one, it integrates into main vortex occurred at tip
    - ✓ Vortices become one about 10 chord behind

- TIP Chipped wing
  - The two sub vortices generate relatively far and they are mixed slowly
  - After sub vortices become one, it integrates into main vortex occurred at tip
    - ✓ If chip is getting closer to leading edge, sub vortices are mixed at far downstream
    - ✓ In the Front Case, even at 20chord downstream, still remain in a state of two vortex

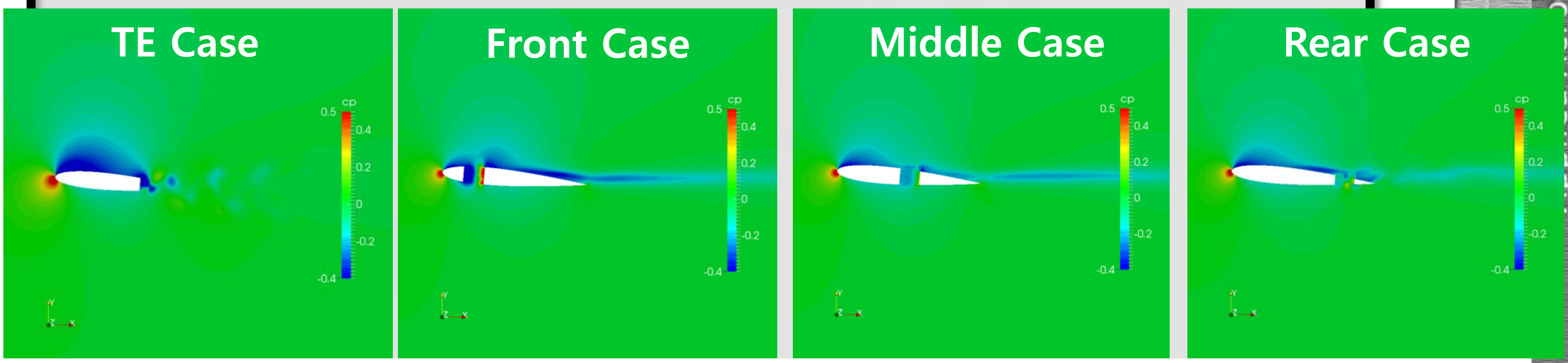


▲ Axial vorticity contour for TE Chipped wing and Tip Chipped wing at far-field of wing

Results

Effect of the Chip Location

- Tip chipped wing
  - If chip is getting closer to leading edge, more strong vortices occur
  - If chip is located front side of the wing
    - ✓ Chip is located at the section which has high pressure difference
    - ✓ Strong vortex occurs because of high pressure difference

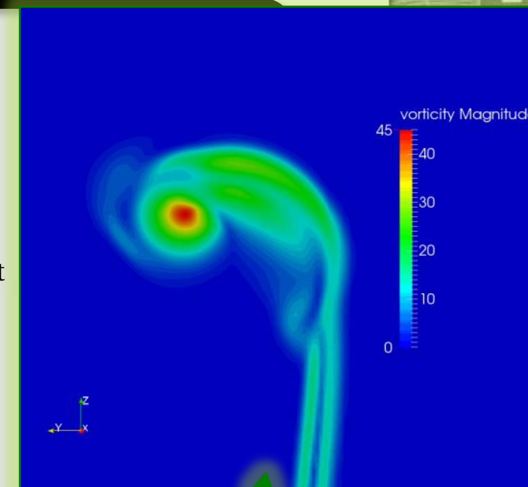


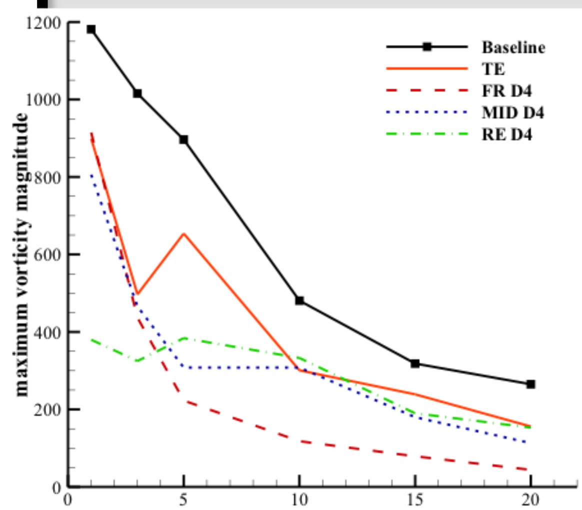
▲ Pressure coefficient contour for TE Chipped wing and Tip Chipped wing

Results

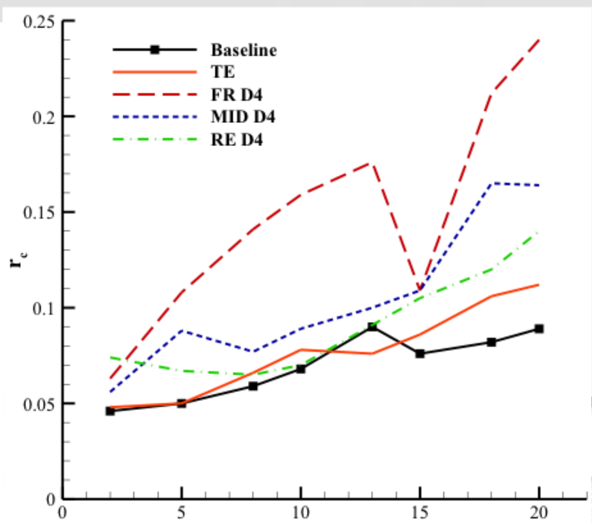
Effect of the Chip Location

- Maximum vorticity magnitude & core radius
  - TE Chipped wing
    - ✓ Similar with dissipation rate of vortex attenuation of Rear case which has weak result
  - TIP Chipped wing
    - ✓ If chip is located front side of the wing, the maximum vorticity magnitude is the smallest
- Swirl velocity distribution
  - Front Case shows 56.5% vortex alleviation rate
  - In front case, vortices wiggle because they still remain respectively at 20chord behind
    - ✓ It could be expected to make more dissipation

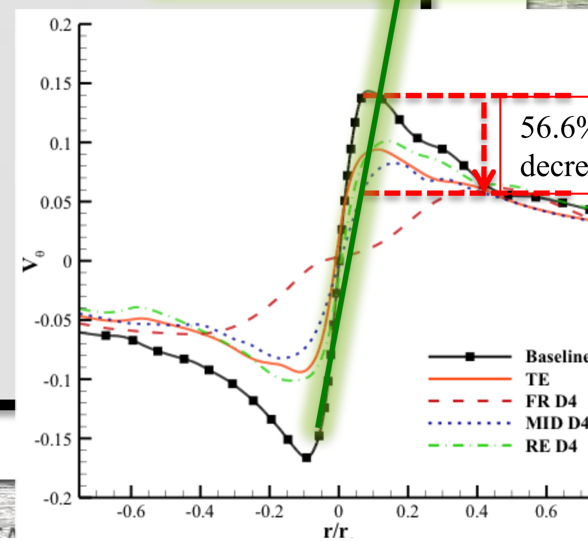




▲ Maximum vorticity magnitude



▲ core radius

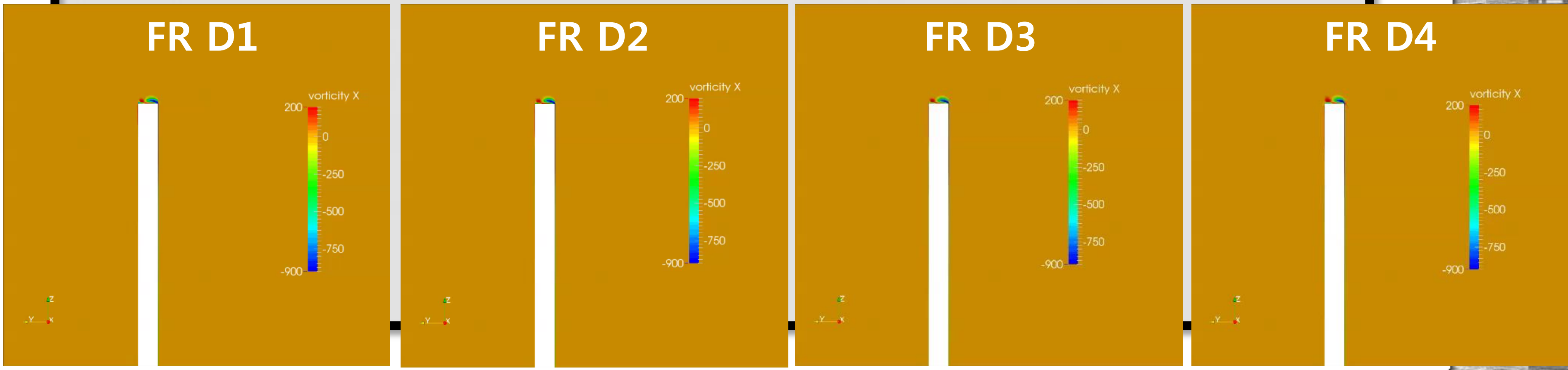


▲ Swirl velocity distribution at x/c=20

Results

Effect of the Chip Depth

- Front D1, D2, D3 D4 Case
  - The depth of the chip is changed from 0.1 chord to 0.4 chord
- Number of sub vortex and vortex strength differ depending on chip depth
  - The number of Sub vortex
    - ✓ Only 1st sub vortex occurs at FR D1 and D2
  - Strength of Sub vortex
    - ✓ FR D1 case generates weaker 1st vortex
    - ✓ D2 ~ D4 Case occur almost same strength's 1st sub vortex
    - ✓ The deeper depth of chip, 2nd sub vortex occurs on a wide range

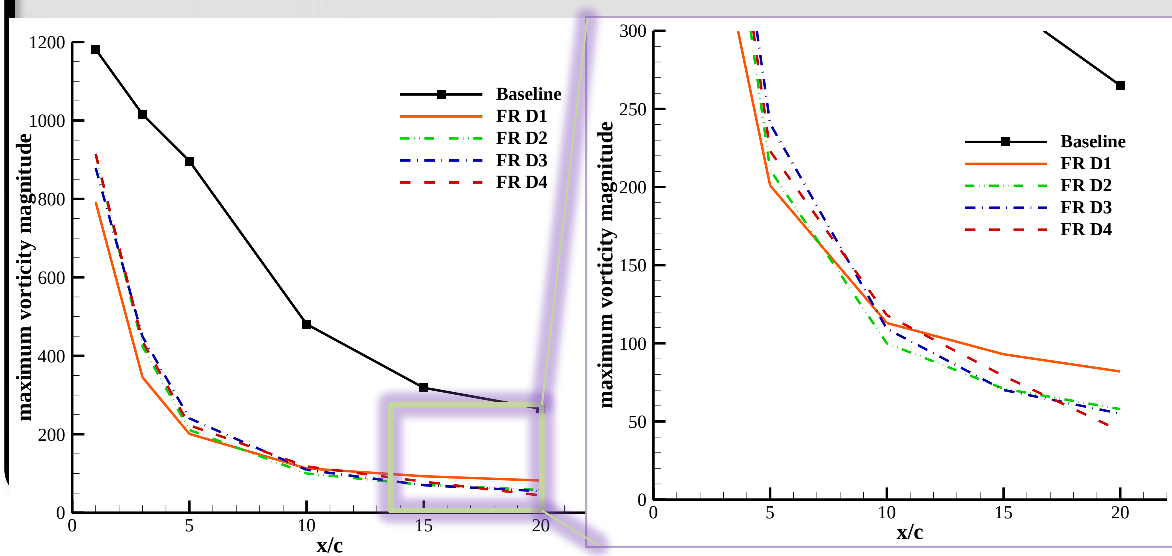


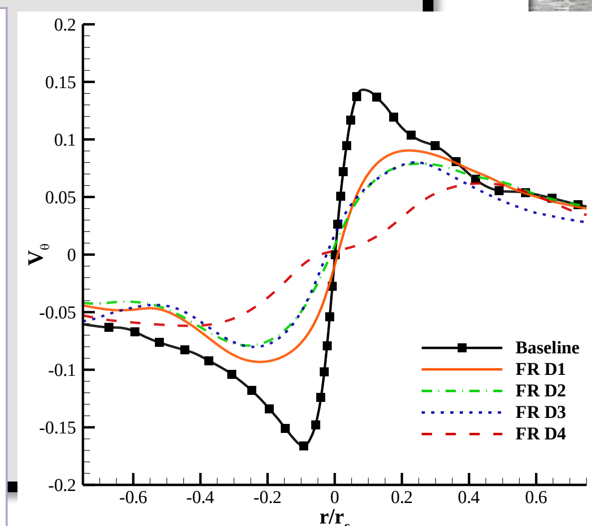
ied Aerodynamics & Design Laboratory

Results

Effect of the Chip Depth

- Maximum vorticity magnitude
  - Effect of depth is less critical than effect of location
  - The case which has deeper chip shows the higher vortex dissipation rate.
- Swirl velocity distribution
  - Vortices become one at 20chord behind except D4 Case
  - Dissipation rate : from minimum 36% to maximum 56.5%





Results

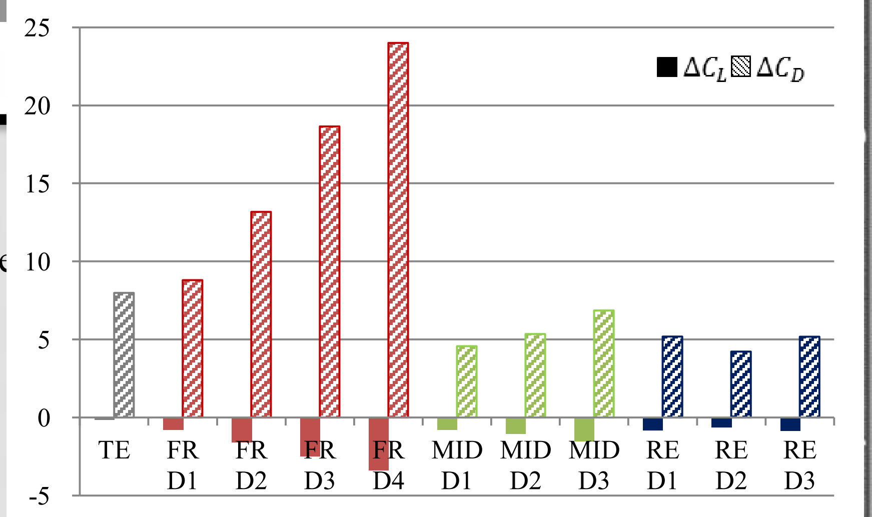
Effect of the Chip Depth

- TIP Chipped wing, D4 Case & TE Chipped wing
  - Chipped area and chip AR are same
- TE Chipped wing
  - Vortices occurred at chip and tip begin to mix at end of wing
    - The vortices grow along the freestream, without anymore disturbance
  - Vortices of two different directions generated by the chip
- Tip Chipped wing
  - Wing tip vortex cuts during the process of generation
    - Vortex generated by the chip inflates temporarily because flow faces end of chip
  - Vortices of two different directions generated by the chip

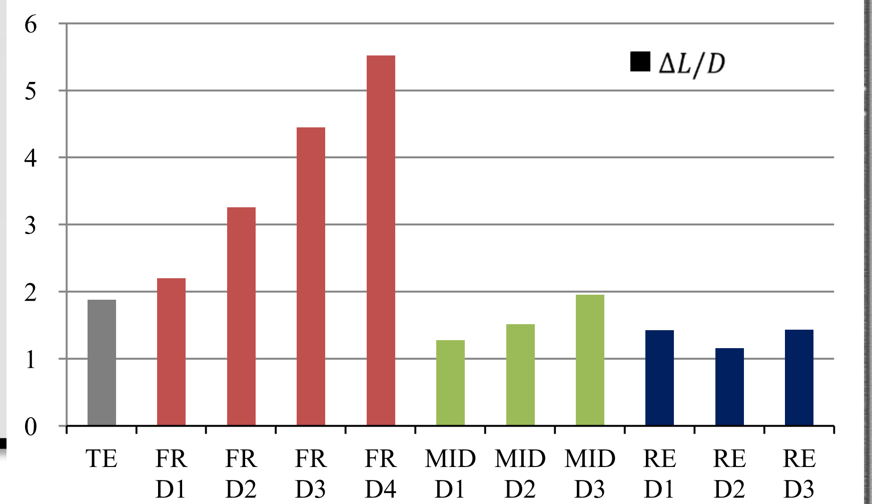
Results

Aerodynamic performance

- Aerodynamic coefficient
  - All of chipped wing case's aerodynamic performance is declined
  - Lift coefficient change
    - Less than 3% in all of chipped wing case
  - Drag coefficient change
    - Changed from 4% to 25%
  - Lift - drag ratio change
    - Increase as value of depth getting bigger
    - Increase as location of chip getting closer to leading edge
    - Changed from 1.1% to 5.5%



▲ Amount of change in the drag coefficient and lift coefficient



▲ Amount of change in the L/D

항공기 Nacelle/Pylon 위치에 따른 Shock-Buffet 현상의 수치적 연구

- 연구 목적 및 방법
- 연구 결과

Applied Aerodynamics & Design Laboratory

Introduction

Motivation

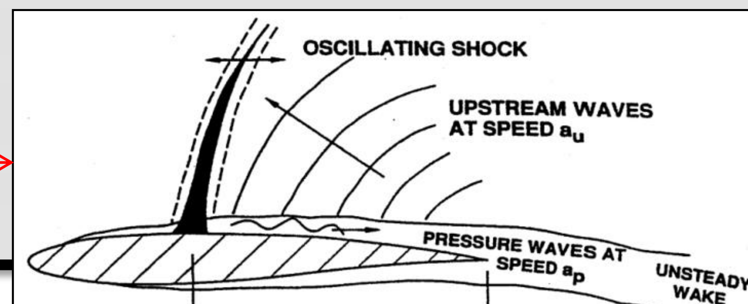
▼ Under the wing

▼ At the rear fuselage

Aerodynamic issues of engine installation

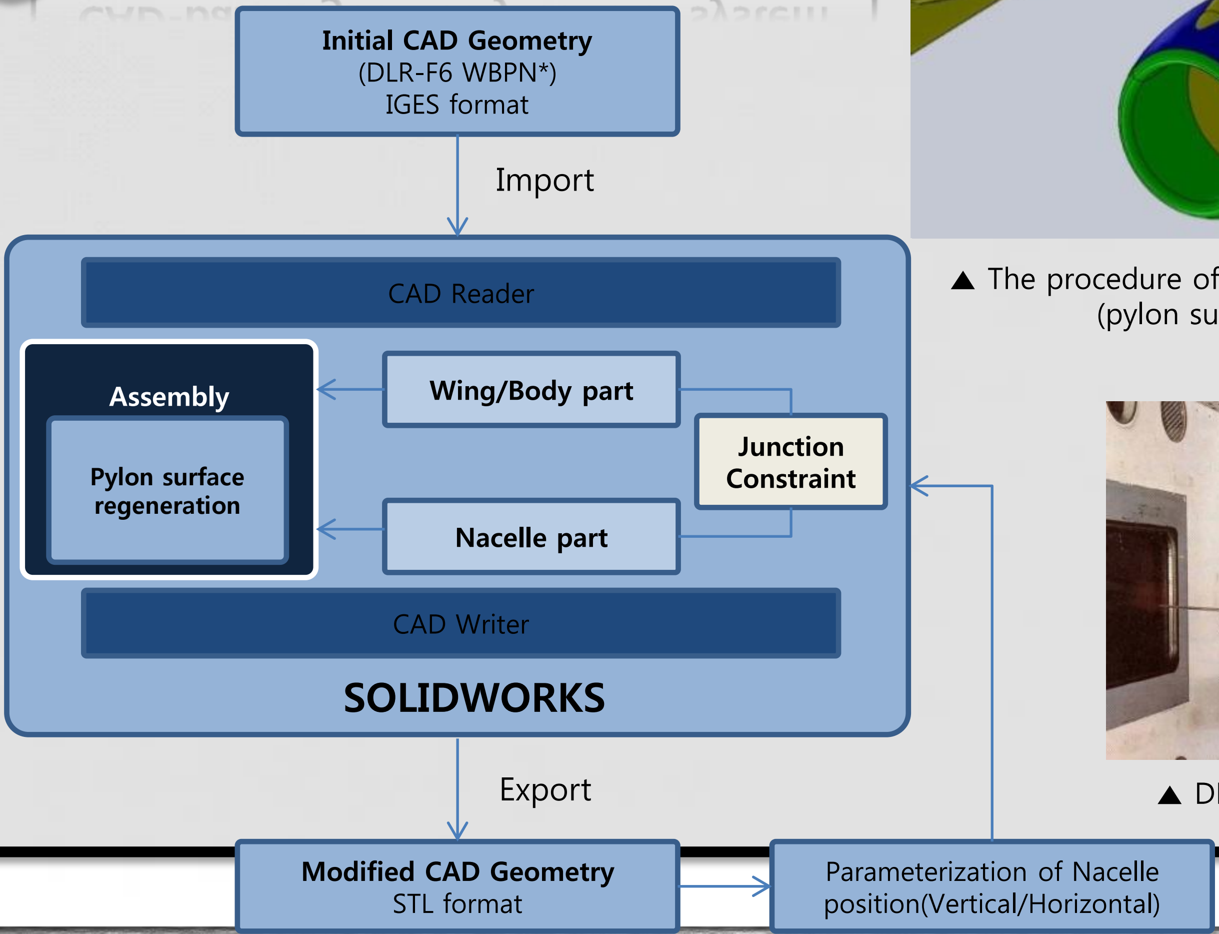
- 날개 하부 유동 가속화 ⇒ **충격파의 발생**
- 강도가 강해질 경우 **천음속 버펫(Buffet)** 발생
- 충격파의 진동, 압력 섭동 현상 수반
- 비행 포위선도(Flight envelop) 제한

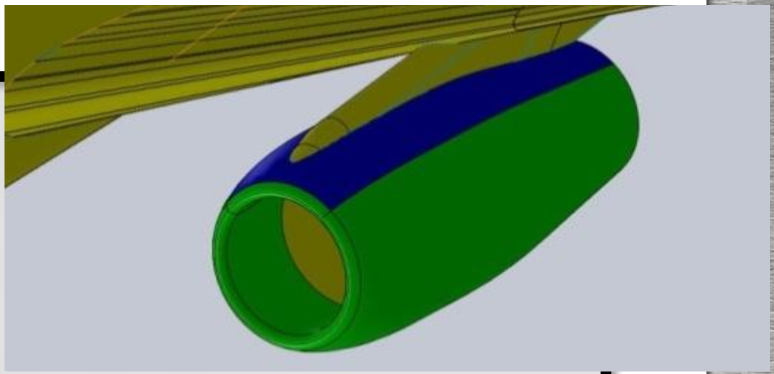
➡ **버펫을 고려한 엔진 위치 설계 요구**



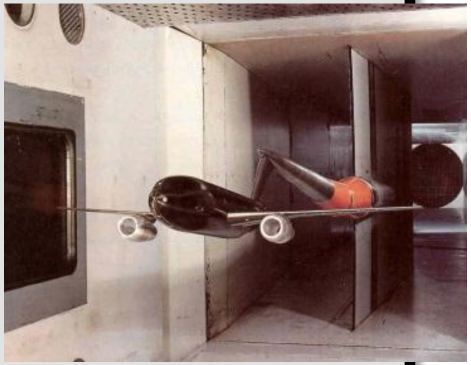
Methods

CAD-based geometry control system





▲ The procedure of CAD-based geometry control (pylon surface regeneration)

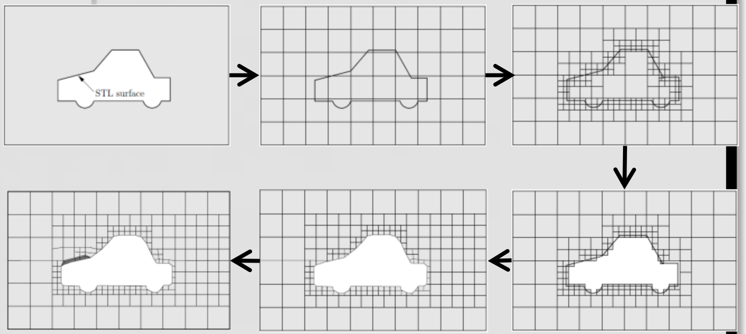


▲ DLR-F6 configuration

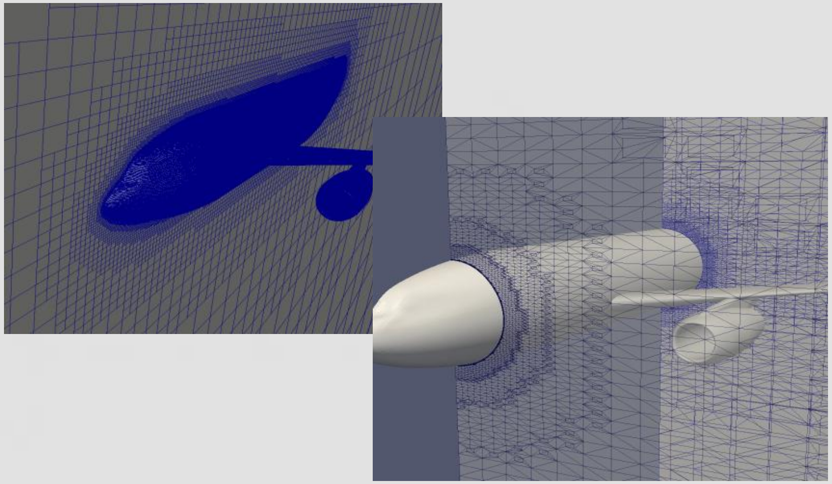
Methods

Automatic grid generation

- Automatic grid generation using snappyHexMesh
  - 나셀의 위치 변화에 따라 항공기 외부형상 변화
  - 외부격자 자동생성 요구
  - OpenFOAM에서 제공하는 자동격자생성 유틸리티인 snappyHexMesh 이용



- Application of snappyHexMesh
  - Euler 해석을 수행하므로 경계층 격자를 생성하는 add-layer off
  - 1 case당 격자 생성 시간 : 6 processors / 약 40 분 소요
  - 격자 수 : 3,950,000개



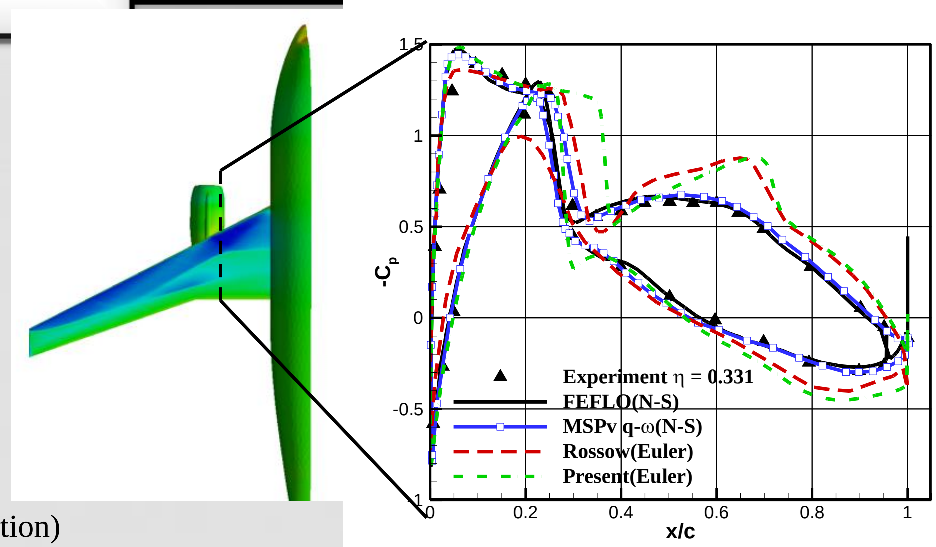
Methods

Solver setting & Validation

- Solver setting(ISAAC)
  - Density-based turbo 기반 내재적 압축성 솔버 코드*
  - Time integral : LU-SGS(Implicit Scheme)
  - Flux Scheme : Roe Scheme
  - 경계조건 : Riemann Condition

- Validation
  - Validation case : 2nd AIAA drag prediction workshop
  - 유동조건 : M_∞=0.75, α=0.98deg (Cruise condition)
  - 파일론 근방 날개표면 C_p distribution(y/b=0.331)
    - ✓ 실험치 및 기존의 N-S 수치해석 결과와 비교
    - ✓ 충격파의 강도와 위치를 비교적 정확하게 예측
  - Aerodynamic Force(Lift) 비교
    - ✓ 간섭현상에 의한 양력의 손실(약 10%)을 동일하게 예측

➢ <u>Euler 해석으로 항공기 간섭현상을 적절히 예측</u>



▲ C_p distribution at y/b=0.331

Experiment*	C_L	Lift Loss
Wing/Body	0.557468	10.88%
Wing/Body/Pylon/Nacelle	0.49679	
OpenFOAM_Euler	C_L	**Lift Loss**
Wing/Body	0.818806	10.00%
Wing/Body/Pylon/Nacelle	0.737	

Methods

Flow condition & Kinkology

- 유동조건
  - 버펫은 대부분 임계 운용조건(하강 또는 선회)에서 발생
  - 하강조건(M_∞=0.74, α=−2.9deg)으로 유동조건 적용

- <u>공력 특성 꺾임 분석법을 이용한 버펫 발단(Buffet Onset) 탐색</u>

날개 뒷전 박리 (버펫 발생)

⇩

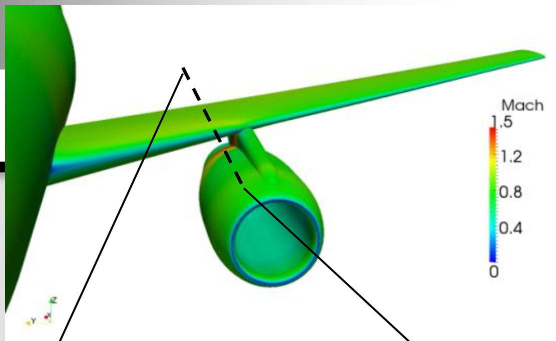
공력하중(Mean aerodynamic load) 변화

⇩

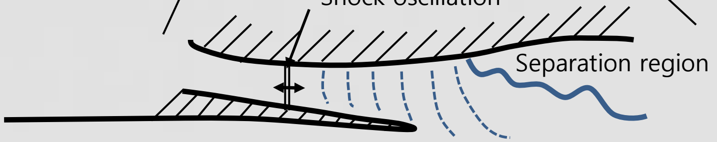
공력특성 불연속 지점 및 기울기의 급격한 변화 부분 판단

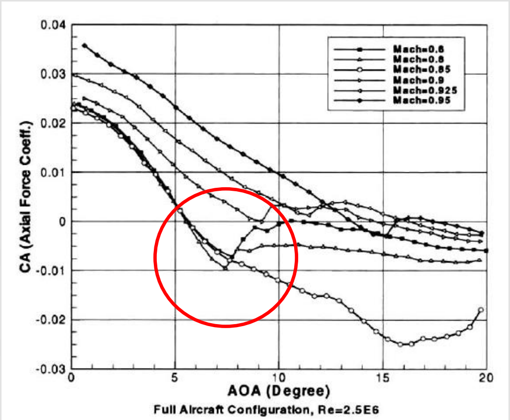
⇩

버펫 발단(Buffet Onset) 예측



▲ Mach contour at the descent condition
($M_\infty = 0.74, \alpha = -2.9\text{deg}$)



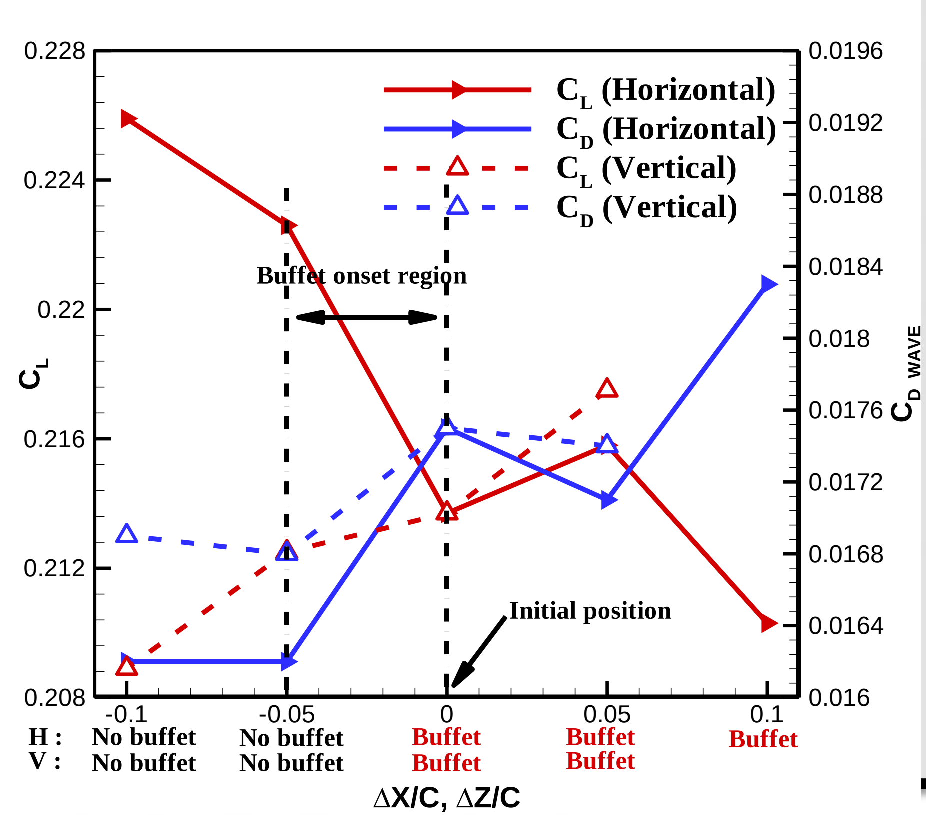


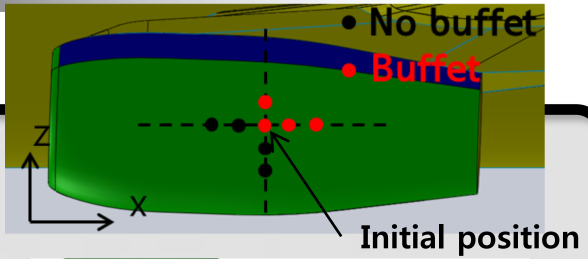
▲ Axial Force Curves*

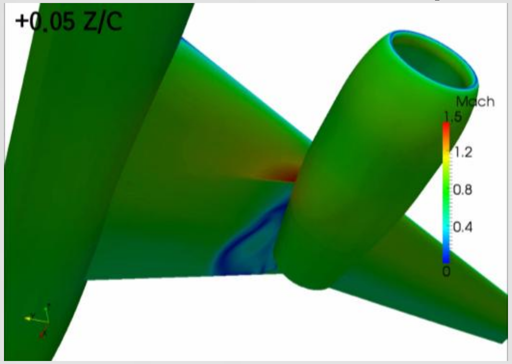
Results

Parametric study

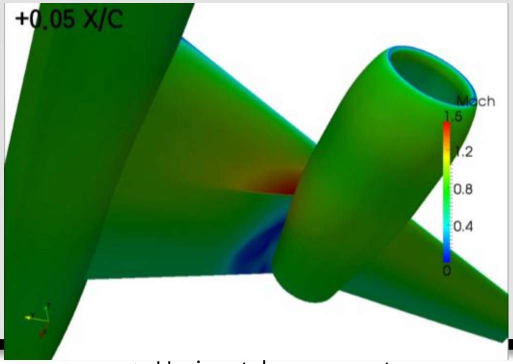
- Effects of positioning nacelle on aerodynamic performance







▲ Vertical movement



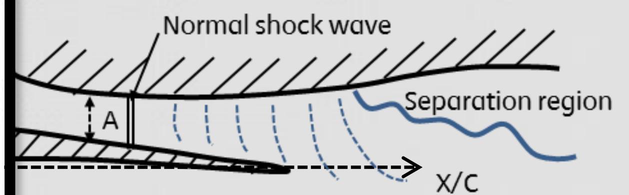
▲ Horizontal movement

Applied Aerodynamics & Design Laboratory

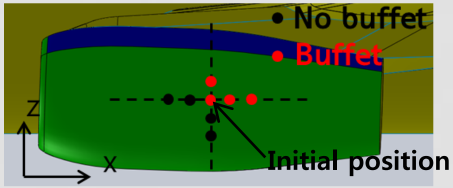
Results

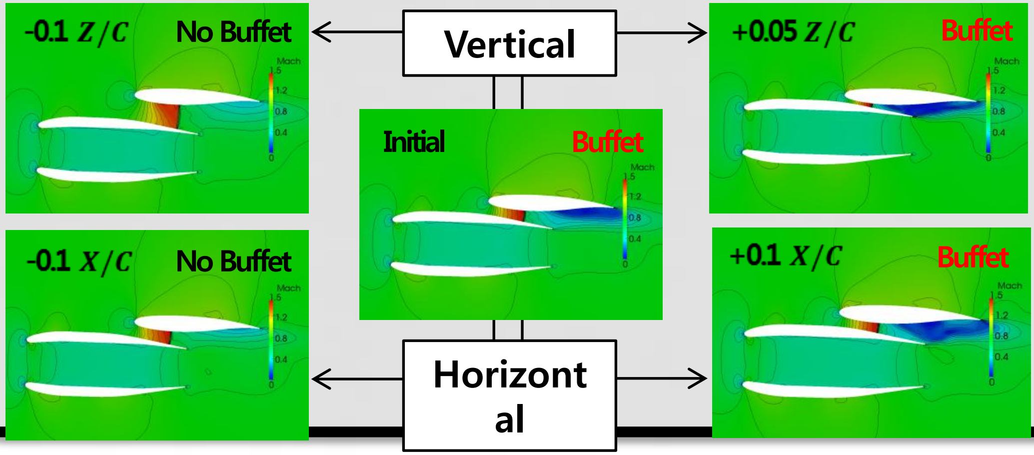
Area ratio VS Shock-buffet

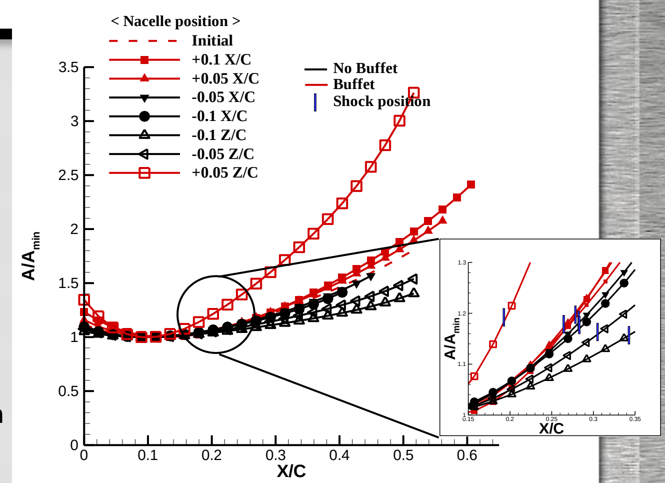
- 날개 뒷전 박리 원인 : 확산영역에서 발생하는 압력 증가(역압력 구배)
- 수축확산 노즐형상구간의 각도가 6°이상을 가지는 CASE ⇒ Buffet 발생
- Divergence angle이 6°를 넘길 경우 역압력 구배에 의한 박리 발생*

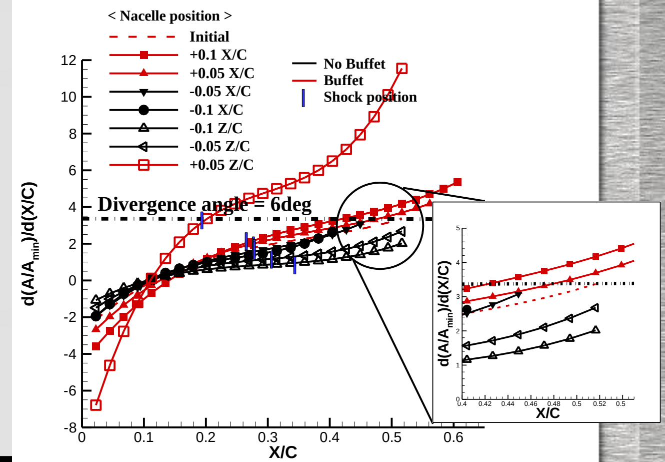


▲ Definition of area in Convergence-divergence nozzle









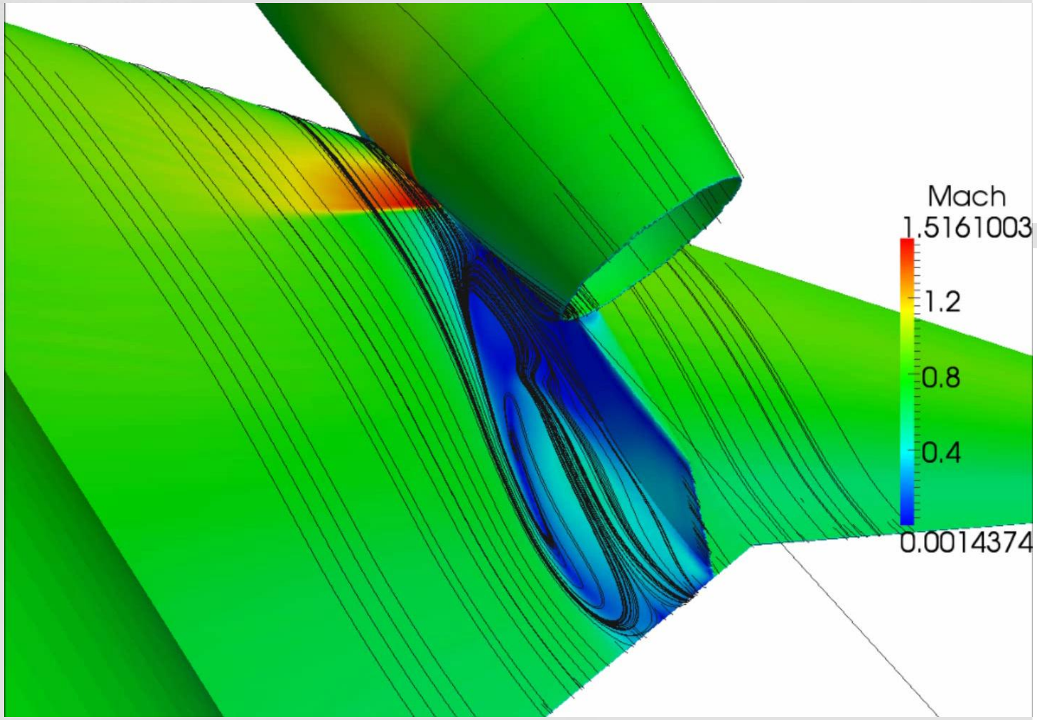
*2009, D. S., Miller, "Internal flow system," Miller Inovation.

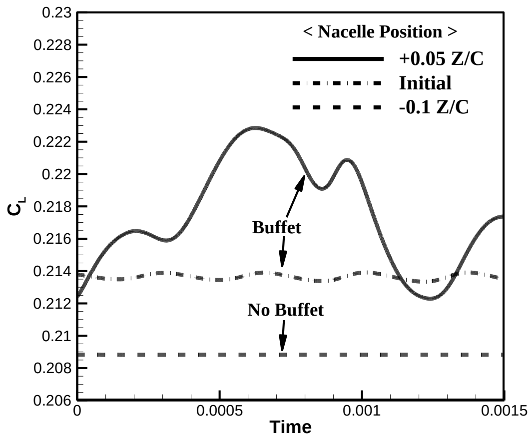
Applied Aerodynamics & Design Laboratory

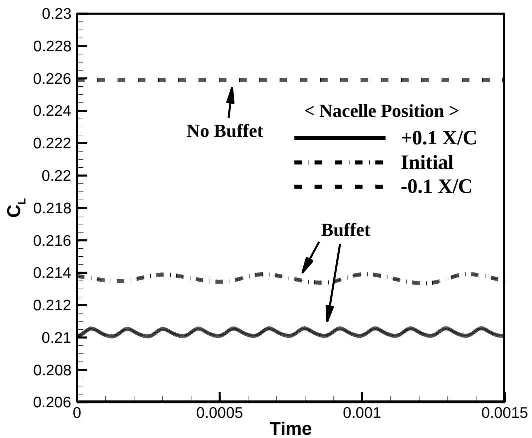
Results

Unsteady effect of Buffet

- 노즐 형상 구간이 버펫의 발생 및 강도에 영향







Applied Aerodynamics & Design Laboratory

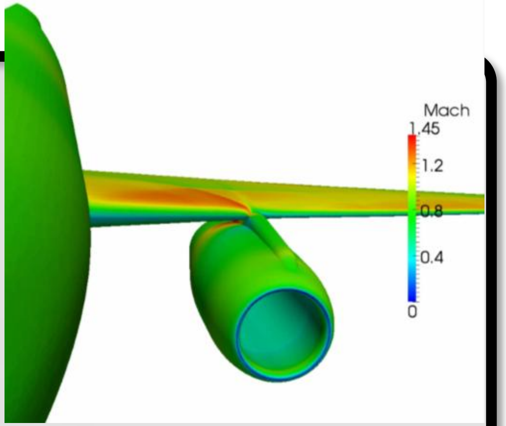
Results

Area ratio VS Shock-buffet

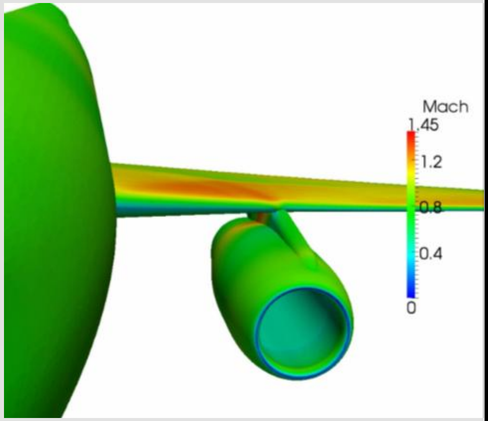
- 순항조건*(M_∞=0.74, α=0.98deg)에서 나셀의 위치 변화가 항공기 성능에 미치는 영향 분석
- 임계운용조건(하강조건)에서의 버펫을 고려한 나셀 위치 이동이 순항조건에서는 날개 상하부에서 발생하는 충격파 완화
- 항공기 성능(양항비)에는 크게 영향을 주지 않음

▼ Aerodynamic performance at cruise condition

	Initial	-0.1 Z/C	-0.1 X/C
C_L	0.753	0.735	0.752
ΔC_L(%)	-	-2.39	-0.13
C_{D_WQVE}	0.0364	0.0354	0.0359
ΔC_{D_WQVE} (%)	-	-2.75	-1.37
$\Delta L/D_{\_wave}$(%)	-	**0.37**	**1.26**



▲ Vertical movement



▲ Horizontal movement

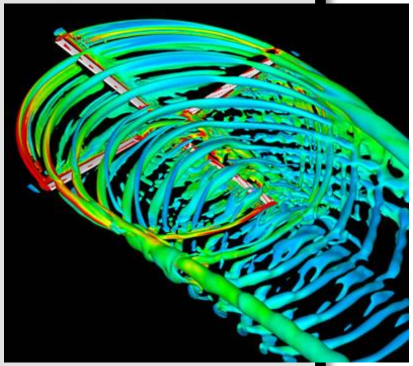
로터 성능 해석용 IASM 모델 개발

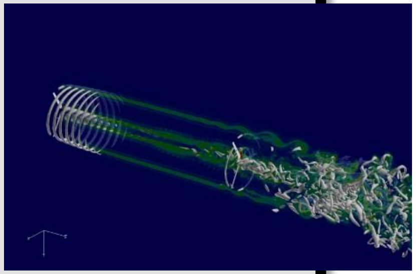
- 연구 목적 및 방법
- 연구 결과

Introduction

Motivation

- Numerical analysis of flow fields around rotor
  - Typical multidisciplinary research area
  - Various numerical methods exist for the rotor analysis
  - **Full CFD simulation** requires huge computational resource to complete an aerodynamic load analysis due to wake instability and complex flow around rotors
  - **Momentum theory** can be useful at preliminary and conceptual design stage, but generally not used for performance analysis
  - **Medium fidelity method** such as blade element theory yields reasonable results for rotor performance analysis in relatively shorter time than full CFD, but that requires vortex wake or inflow model such as uniform or dynamic inflow model

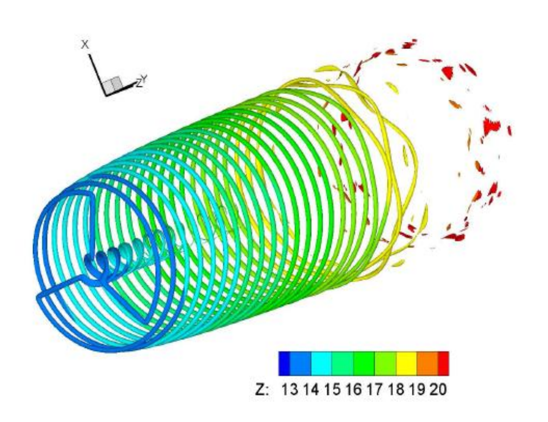




Introduction

Motivation

- Actuator model method for rotor analysis
  - Actuator Surface/Line Model
    - ✓ In ALM and ASM, the aerodynamic effects of a rotor blade are imposed on the cells in the computational domain which lie in the exact location of the rotor blade at given azimuth angle
    - ✓ This method is applied by several researches at the wind turbine[Troldborg(2008); Masson(2008)] and the helicopter[Kim et al.(2009)]
    - ✓ As can be found in the previous studies, one of **the most ambiguous steps in ASM/ALM is to set the location of the reference line** which will be used for measuring induced velocities
    - ✓ When the reference line is too far away from the blade, it tends to yield relatively low induced velocities. On the other hand, when too short, it is difficult to separate the induced velocity due to the tip vortex from that of bound circulation on the blade
    - ✓ Accordingly, **the performance analysis capability of ASM/ALM is considered less reliable than full CFD method**



Methods for rotor analysis

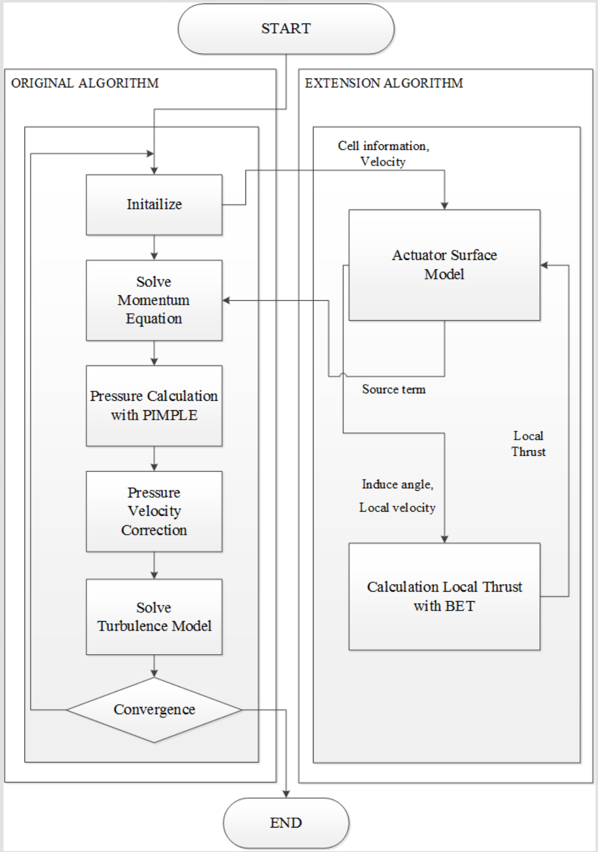
Rotor Performance Analysis Solver

- RANS solvers
  - **pimpleFoam** is **PISO+SIMPLE** algorithm-based unsteady Navier-Stokes solver for the incompressible and turbulence flow in OpenFOAM
  - These solvers are able to use various RANS-based turbulence model in OpenFOAM
- Modification of RANS solvers in order to include actuator model
  - In order to implement actuator surface model to pimpleFoam solvers, a source term from BET is added to the calculating region cells
  - **BET(Blade Element Theory)** method is employed for considering rotor effect
  - **The velocity field** required in BET was obtained from **CFD calculation results** and the force vector converted from source term is included in the **momentum equation**

Modified OpenFOAM algorithm

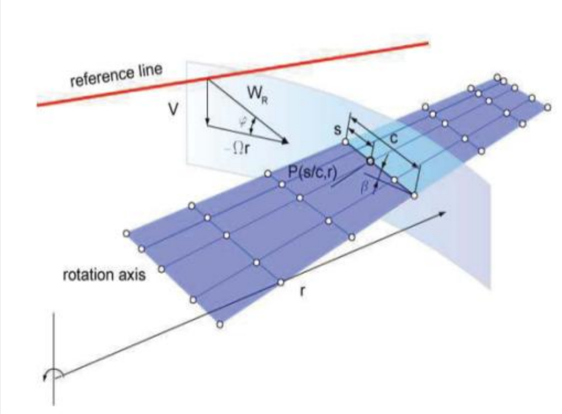
Modified OpenFOAM solver

- Modified RANS solvers algorithm
  - The modified RANS solvers algorithm is as shown in the right
  - The source code to include rotor effect based on the actuator surface method is implemented in the solvers package
  - The source code for calculating source term about rotor effect obtains parameters as the velocity field and the cell geometry information from the original algorithm
  - The original part of the solvers follows the OpenFOAM standard algorithms, and the extension part of solvers includes the actuator model, BET and flapping solution



Actuator Surface Model

- Actuator Surface Model
  - The way of adding source term to momentum equation in the Actuator Surface Model is similar to the Actuator Disk Model
  - The relative velocity used in BET is obtained by the reference line as shown right
  - Displacement effect of each blades are considered for exact induced velocity
  - Tip loss correction was considered by tip loss function or displacement effect
  - The calculation of local thrust in this method is in consideration of the rigid blade positions at calculating time as shown right figure and following equation



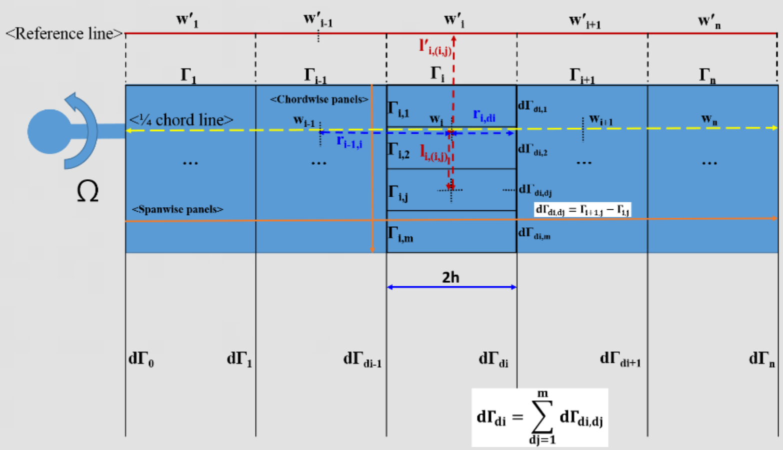
$$dT = d\psi(dL cos\alpha_i - dD sin\alpha_i) = \frac{dA}{2}\rho V^2(c_l cos\alpha_i - c_d sin\alpha_i)$$

Applied Aerodynamics & Design Laboratory

Theory and Numerical methodology

Actuator Surface Model

- The actuator surface model with displacement effect of reference line
  - The induced velocity values from CFD at reference line equal the velocity values are induced by the Γs distributed on the blade along the spanwise and chordwise panels
  - Therefore, the exact induced velocities for BET are obtained by correcting the effect of Γs on CFD velocities at the reference line
  - For the exact induced velocity calculating, the displacement correction and bound circulation considering due to reference line position is needed
  - The displacement correction is derived by lifting line theory



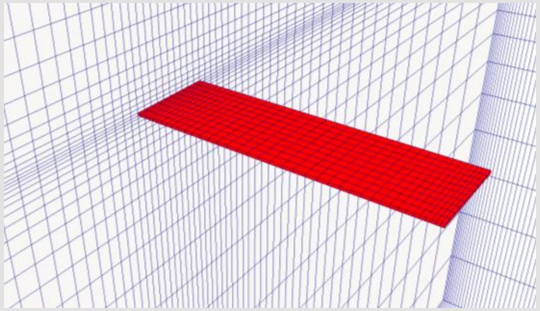
[Vortex filaments of ASM]

Results of Applications

N0015 fixed wing case

- Parametric study about reference line and chordwise panel numbers
  - For the validation of displacement correction on the rotating, the parametric study about reference line position and number of chordwise panel is needed
  - N0015 fixed wing case is selected because this has the experimental results (McAlister,1992) according to the angle of attack
  - Calculated angle of attack : 4º, 8º, 12º
  - Re : $1.5*10^6$
  - Parametric study cases (AoA : 8º)

	Chordwise panel No.(cdn_)	Reference line positions(rp)
Reference line positions study	6	0.5, 1.0, 1.5, 2.0, 3.0, 5.0 avg. chord in front of 1/4 chord line
Chordwise panels study	2, 4, 6, 8	1.5 avg. chord in front of 1/4 chord line

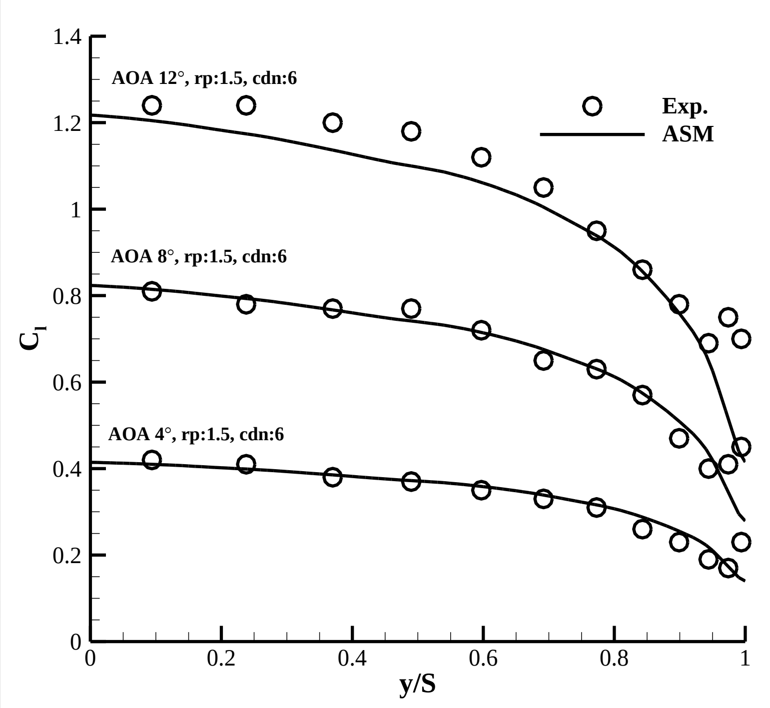


[Analysis Mesh]

Results of Applications

N0015 fixed wing case

- Calculation results
  - Right figure shows the results of C_l distribution along the wing span
  - In all experiment data, there is a peculiar distortion in C_l value along the outermost 3% of the span by the tip vortex that forms on the suction side of the wing tip
  - The calculation results by ASM are in good agreement with the experiment data along the almost of the wing span for all cases

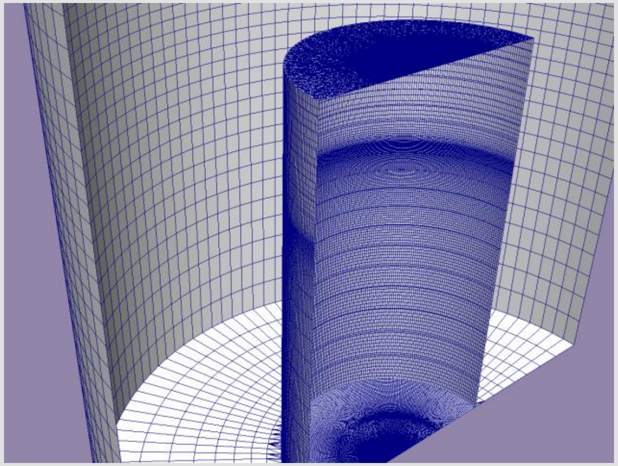


Results of Applications

Hovering flight case

- Onera7A rotor

	7A case
Radius (m)	2.1
Root cut	0.2 R
Chord	0.0667 R
Aspect Ratio	15
Blade planform	Rectangular
Blade number	4
Rotor tip speed (Mach No.)	0.6612
Twist angle (°)	Linear twist
Collective pitch angle at 0.7R (°)	7.5
C_T	0.00679

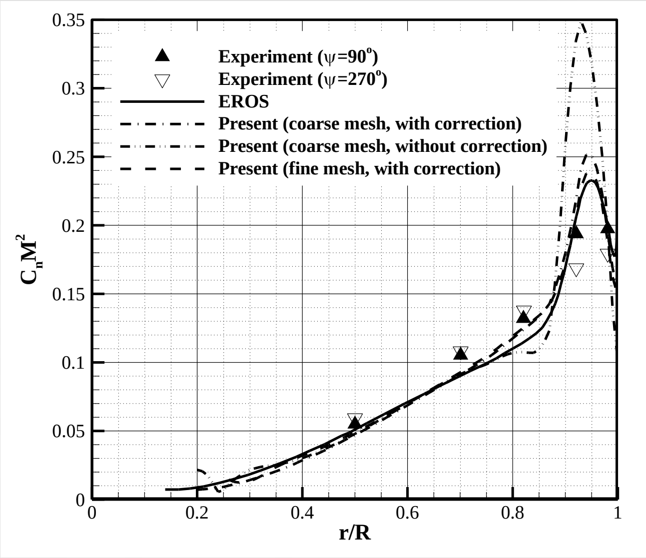


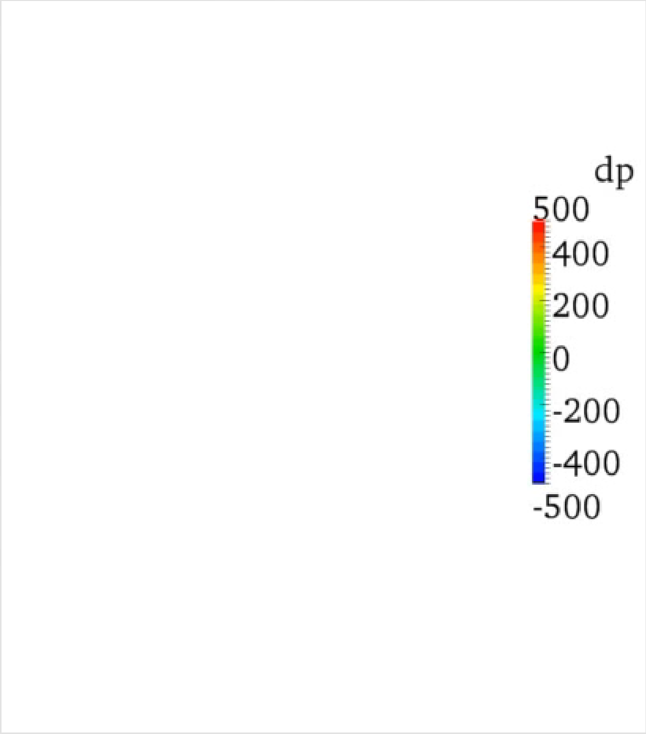
[Analysis Mesh, 6,667,992 cells]

Results of Applications

Hovering flight case

- Caradonna Case
  - Following figure is the comparison resutls the experiment test and analysis results
  - Right animation ise the velocity contour and Q criteria of the tip vortex





Results of Applications

Forward flight case

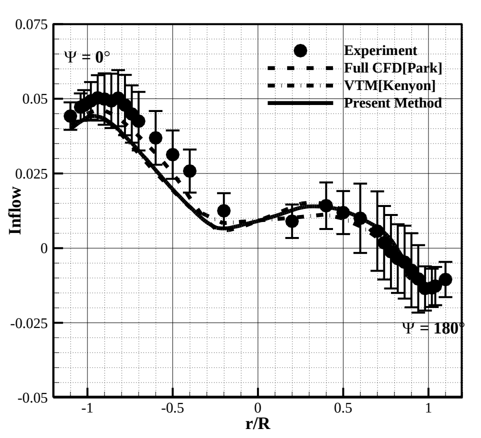
- Elliott experiment(1988) conditions

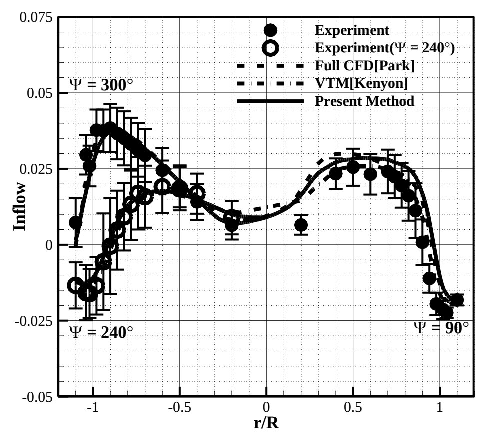
	Experiment	ASM
Blade type	Rectangular	
Radius(m), 1R	0.8605	
Root cut	0.253R	
RPM	2100	
C_T	0.0063	0.00624 (0.006255)
Solidity	0.0977	
Shaft angle(°)	-3	
Linear twist(°)	-8	
Advanced Ratio	0.15	
θ_0(°)	9.37	7.328
θ_{1c}(°)	-1.11	-1.985
θ_{1s}(°)	3.23	2.681

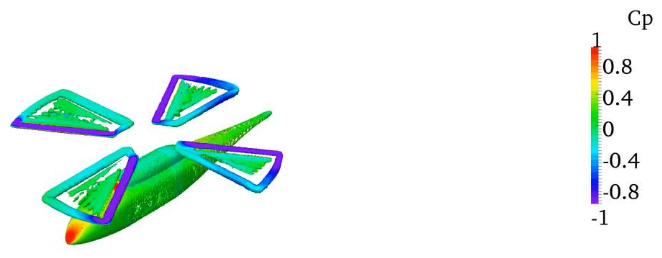
Results of Applications

Forward flight case

- Inflow results
  - Inflow distributions show good correlation with experimental data and other numerical analysis
  - In the Q criteria results were gotten by the solver based on actuator surface model, the blade tip wake strength due to the tip vortex contraction shows clearly







[Iso Q criteria and velocity magnitude]

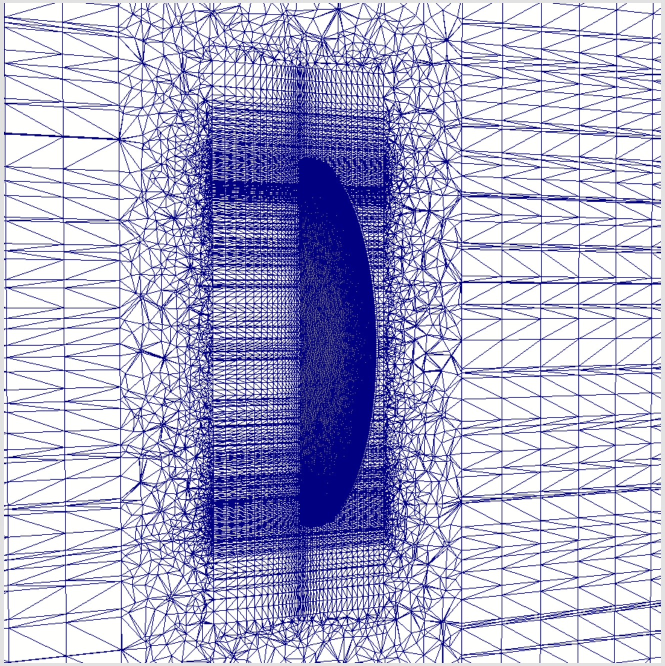
Applied Aerodynamics & Design Laboratory

Results of Applications

5MW RWT Analysis

- Calculation conditions

Blade No.	3
Radius(m), 1R	63
Root cut	0.0456R
RPM	9.2
Wind speed (m/s)	8
Mesh type	Hybrid mesh
Blade information positions	17
Mesh No.	3,284,022 cells
With hub	X

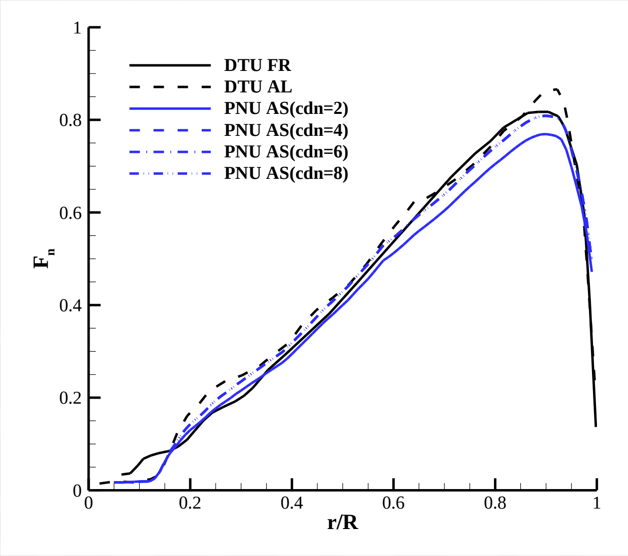


[Analysis Mesh]

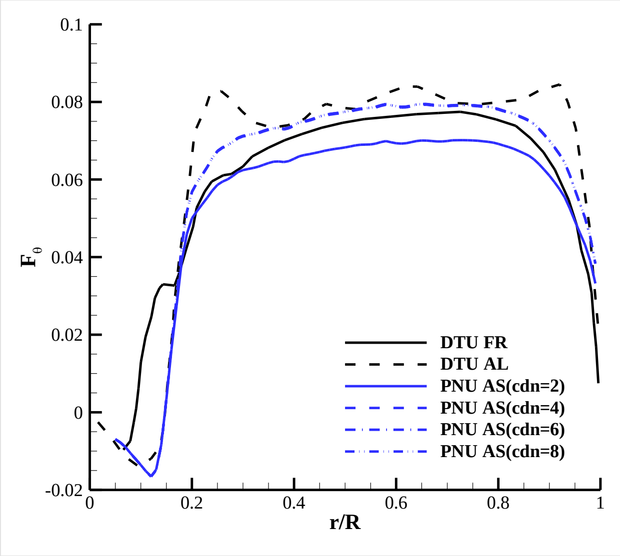
Results of Applications

5MW RWT Analysis

- Calculation results
  - Sectional F_n, F_θ results according to chordwise panel number
  - The case results that has over 4 panel number were almost same regardless of number of panels and were seen to be in good agreement with FR results



[Sectional Normal Force]

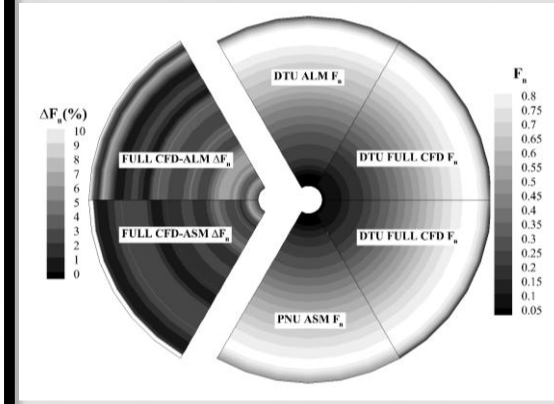


[Sectional Tangential Force]

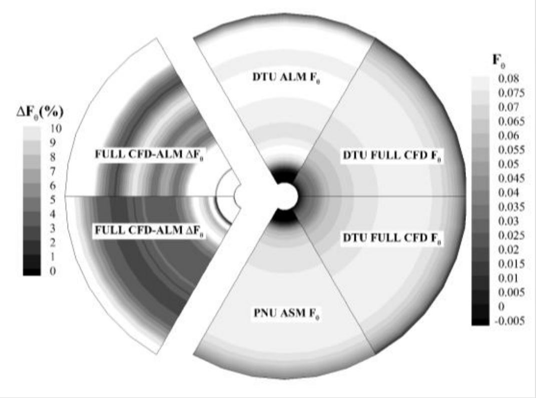
Results of Applications

5MW RWT Analysis

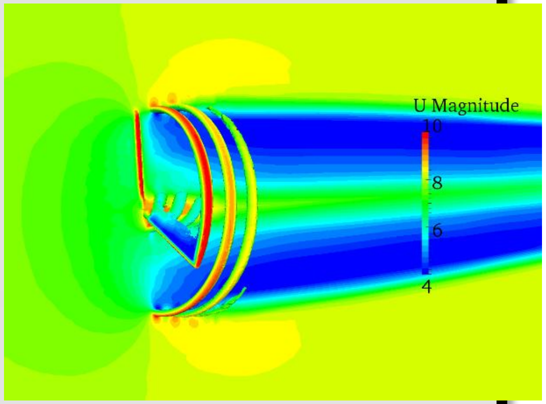
- Calculation results
  - The comparison of normal and tangential force coefficients from the results of DTU ALM, ASM of present method and full CFD of DTU
  - ΔF_n and ΔF_θ denote the difference of values between full CFD and ALM or ASM per the maximum value of full CFD
  - It is observed that the present results show more similar behavior to full CFD than to ALM
  - The over-prediction of tangential forces at the tip and root region is not observed in the present results



[Comparison of Sectional Normal Force]



[Comparison of Sectional Tangential Force]



[Iso Q criteria and velocity magnitude]

Concluding remark

- Conclusions

Concluding remarks

Conclusions

- 공력 해석을 위한 오픈폼의 활용
  - 기 개발된 해석자의 사용이 가능한 유동 영역 해석의 경우,
    - ✓ 해석 케이스의 경우는 비교적 셋업이 용이
    - ✓ 외부 유동 해석에서의 경계 조건 설정은 압축성, 비압축성 영역에 대한 적절한 예측을 수행하여 경계 조건 설정 필요
    - ✓ 난류 모델에 경우에는, 사전에 검증 과정을 통해 해석 조건에 적절한 난류 모델을 선정하여여 함
  - 기 개발된 해석자에 추가적인 수정이 필요한 경우,
    - ✓ 수정에 사용할 기존 해석자의 선택에 유의 : 정상, 비정상 해석자 유무, 열해석 필요의 유무등
    - ✓ 기존 해석자에 해석 항을 추가하는 변경의 경우, 기존 해석자의 골격을 유지하면서 추가항의 처리 부분을 확장하는 방식으로 수정
    - ✓ 새로운 형식으로 해석자를 개발하는 경우(ISSAC 등), 오픈폼 내부에서 사용이 가능한 라이브러리와 타 연구자가 공개한 프로그램을 기반으로 개발하는 것이 시간 및 버그 수정 시간을 절약하는 수 있는 방안임

Thank you for your attention

- Q & A