

Universiti Teknologi Malaysia

BORANG PENGESAHAN STATUS TESIS*JUDUL: WING-EXTERNAL STORE AERODYNAMIC INTERFERENCES OF A SUBSONIC AIRCRAFTSESI PENGAJIAN: 1999/2001Saya HUONG YU SAINT
(HURUF BESAR)

mengaku membenarkan tesis (PSM/Sarjana/Doktor-Falsafah)* ini disimpan di perpustakaan Universiti Teknologi Malaysia dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hakmilik Universiti Teknologi Malaysia.
2. Perpustakaan Universiti Teknologi Malaysia dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (✓)

SULIT

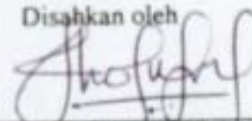

(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia sepertimana yang termaktub di dalam AKTA RAHSIA RASMI 1972)

TERHAD

(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)

TIDAK TERHAD

Disahkan oleh



(TANDATANGAN PENULIS)


(TANDATANGAN PENYELIA)

Alamat Tetap:

No. 24, USJ 13/2USJ Subang JayaSubang Jaya Selangor Darul Ehsan.Tarikh: 13/12/01Dr. Tholudin Bin Hj Mat Lazim

Nama Penyelia

Tarikh: 13/12/01

- CATATAN:
- * Potong yang tidak berkenaan.
 - ** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh tesis ini perlu dikelaskan sebagai SULIT atau TERHAD.
 - * Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secara penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

**WING-EXTERNAL STORE AERODYNAMIC INTERFERENCES OF A
SUBSONIC AIRCRAFT**

HUONG YU SAINT

**A thesis submitted in fulfillment of the requirements for the award
of the Degree of Master of Engineering (Mechanical)**

**Faculty Of Mechanical Engineering
Universiti Teknologi Malaysia**

AUGUST 2001



BAHAGIAN A - Pengesahan Keputusan

Apakah keputusan bahawa projek penyelidikan ini layak untuk dipertimbangkan untuk

berdasarkan semua _____

" I declared that I read this thesis and in my point of view this thesis is qualified in term of scope and quality for the purpose of awarding the Degree of Master of Engineering (Mechanical)."

Nama

Jawatan

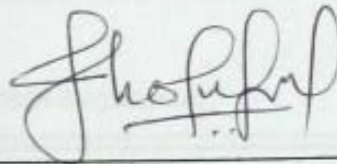
(Geng/Unit)

* / Sila letakkan tandatangan anda

Signature

Supervisor Name

Date



DR. THOLUDIN BIN HJ MAT LAZIM

13/12/01

Nama dan Alamat

Pemeriksa I

Nama dan Alamat

Pemeriksa II

Nama dan Alamat

Pemeriksa III

(Geng/Unit)

Disahkan oleh Pengarah Akademik, Pengarah Penyelidikan & S&T

Tandatangan

Nama

DR. IR. ABDUL HUSNUS



13/12/01

BAHAGIAN A – Pengesahan Kerjasama*

Adalah disahkan bahawa projek penyelidikan tesis ini telah dilaksanakan melalui kerjasama antara _____ dengan _____

Disahkan oleh:

Tandatangan : Tarikh :

Nama :

Jawatan :

(Cop rasmi)

* Jika penyediaan tesis/projek melibatkan kerjasama.

BAHAGIAN B – Untuk Kegunaan Pejabat Sekolah Pengajian Siswazah

Tesis ini telah diperiksa dan diakui oleh:

Nama dan Alamat
Pemeriksa Luar : **Dr. Ir. Bambang Basuno**
School of Aeronautic Engineering
Universiti Sains Malaysia
Kampus Kejuruteraan Transkrian
14300 Nibong Tebal
Seberang Perai Selatan
Penang

Nama dan Alamat
Pemeriksa Dalam I : **Dr. Mohd Nazri bin Mohd Ja'afar**
Fakulti Kejuruteraan Mekanikal
UTM, Skudai

Pemeriksa Dalam II :


Nama Penyelia lain :
(jika ada)

Disahkan oleh Pengurus Akademik (Penyelidikan) di SPS:

Tandatangan : Ibrahim Busu Tarikh : 29/12/01

Nama : **DR. IBRAHIM BIN BUSU**

“ I declared that this thesis is the result of my own work except the ideas and summaries which I have clarified their sources. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any degree.”

Signature : 
Writer's Name : HUONG YU SAINT
Date : 13/12/01

ACKNOWLEDGEMENT

I would like to express my gratitude to my supervisor, Dr Tholudin Bin Hj Mat Lazim for his guidance, trust and assistant throughout the project.

I would like to thanks the personnel at UTM production laboratory and assistant rendered by SMEa Sdn Bhd in completing the wind tunnel models.

My special appreciation to my wife and my children for their patience during the course.

I would like to thank my colleagues and friends for their moral support and assistant specially, Shaharudin Bin Ahmad, Endra and Darwis.

Finally, I would like to thank the Ministry of Defense for sponsoring my study and UTM for providing grant for the research.

ABSTRACT

Modern fighter aircraft are mostly designed to carry its store externally. Installing store to an aircraft wing externally would have much engineering implication especially through the change in the aerodynamic characteristic. This research was carried out to study the application of the Computational Fluid Dynamics (CFD) method along with experimental methods in predicting the aerodynamic interference caused by these installations. Commercial CFD code, Fluent 5.3 had been validated using experimental results reported in the literature for two dimensional, subsonic and transonic flow over the NACA 0012 and the RAE 2822 airfoil. Subsequently, low speed wind tunnel experiments were carried out over a wing model installed with an external store. The wing model was fabricated based on a digitized BAe Hawk 208 fighter wing. For further CFD code validation, the wind tunnel configurations were again simulated using the CFD method and its results were validated with the experimental results. Finally, a simplified full scale Hawk 208 aircraft model carrying an external store was simulated at various attitudes and flow speeds. In the two-dimensional subsonic flow, pressure distribution predicted by CFD was in good agreement and comparable to the experimental results. For the transonic two-dimensional flow validation, pressure distribution predicted by various flow models were slightly different from the experimental results (9% to -22.2% in term of C_L). For wind tunnel configuration, an average of about 12% deviation in pressure distribution between the results predicted by the CFD method and measured in the wind tunnel. The results of the full scale Hawk 208 simulation show that the aerodynamic interference caused by the store installation were mostly evidence on the lower wing surface and negligible on the upper surface at low angle of attack. This trend was reversed as the angle of attack was increased. The area of influence on the wing surface by store interference increased in line with the increased in airspeed.

ABSTRAK

Kebanyakan pesawat pejuang modern adalah direka untuk membawa "store" di bahagian luar seperti di bawah sayap. Memasang "store" di bawah sayap pesawat mempunyai banyak implikasi terutamanya yang disebabkan oleh perubahan dalam ciri-ciri aerodinamik. Projek kajian ini mengkaji penggunaan kaedah CFD dalam meramal gangguan aerodinamik seiring dengan ujikaji terowong angin. Pengaturcaraan CFD komersial iaitu Fluent 5.3 telah digunakan untuk meramal aliran dua dimensi ke atas aerofoil NACA 0012 dan RAE 2822 dalam lingkungan aliran subsonik dan juga transonik. Ini diikuti dengan satu ujikaji terowong angin ke atas satu model yang terdiri daripada sebahagian sayap pesawat yang dipasang dengan pelancar roket, ia merupakan model yang diringkaskan daripada pesawat pejuang Hawk 208. Keputusan ujikaji terowong angin akan digunakan untuk menilai ketepatan ramalan oleh kaedah CFD dalam aliran 3 dimensi dan memantau perubahan aerodinamik yang disebabkan oleh Pelancar Roket. Akhir sekali, kaedah CFD digunakan untuk meramal aliran angin dan gangguan angin yang disebabkan oleh pemasangan Pelancar Roket ke atas satu model pesawat Hawk 208 bersaiz penuh pada pelbagai kelajuan dan sudut tuju. Keputusan simulasi dengan kaedah CFD untuk aliran dua dimensi, mendapati ramalan tekanan bagi aliran subsonik adalah sangat baik jika dibandingkan dengan keputusan ujikaji yang dilaporkan. Walaubagaimanapun ramalan CFD bagi aliran transonik memperolehi perbezaan antara 9% ke -22.2% dalam C_L dibanding dengan keputusan ujikaji. Perbezaan sebanyak 12% dalam tekanan udara didapati antara ramalan CFD dengan keputusan ujikaji terowong angin yang dijalankan dalam kajian ini. Dalam kes simulasi pesawat berukuran penuh, ramalan CFD menunjukkan bahawa, gangguan aliran angin tertumpu pada bahagian bawah sayap dan hampir tiada kesan pada bahagian atas pada sudut serang yang rendah, trend ini terbalik apabila sudut serang meningkat. Keluasan kawasan yang dipengaruhi oleh gangguan Pelancar roket meningkat seiring dengan kelajuan pesawat.

	2.2.1 Fundamental Fluid Dynamic and Governing Equations	11
	2.2.2 Approximation Level and CFD	22
	2.3 Spoke Distribution (Mesh Generation)	23
	2.4 Numerical Solution (Solving Eq Discretization)	28
	2.5 Numerical Resolution (Iteration and Acceleration Techniques)	30
CHAPTER	SUBJECT	PAGE
	2.6 Fundamental Air Properties	
	2.7 Approximations and its Validity	
	2.7 Summary	30
	ABSTRACT	v
	CONTENT	Vii
CHAPTER III	LIST OF TABLE	Xiii
	LIST OF FIGURES	Xiv
	NOMENCLATURES	xvii
	3.2 Research Methodology Overview	37
	3.3 Software Description	39
CHAPTER I	INTRODUCTION	1
	1.1 Introduction	1
	1.2 Research Objective	2
	1.3 Research Scope	3
	1.4 Research Methodology	3
	1.5 Expected Results	4
	1.6 Outline of Thesis	5
CHAPTER IV	TWO DIMENSIONAL SUBSONIC FLOW OVER A CABLE AIRFOIL - CASE	45
CHAPTER II	LITERATURE REVIEW	6
	2.1 Computational Fluid Dynamics	8
	2.2 Selection of Mathematical Model	11
	4.1 Simulation Methodology	49

	2.2.1	Fundamental Fluid Dynamic	11
	2.2.2	and Governing Equations	13
	2.2.2	Approximation Levels and CFD	22
2.3		Space Discretization (Mesh Generation)	25
2.4		Numerical Scheme (Governing Eq	28
		Discretization)	28
2.5		Numerical Resolution (Iteration and	30
		Acceleration Techniques)	30
2.6		Fundamental Air Properties	31
		Approximations and Its Validity	35
2.7		Summary	35
CHAPTER III	METHODOLOGY		36
	3.1	Introduction	36
	3.2	Research Methodology Overview	37
	3.3	Software Description	39
	3.3.1	Photomodeler Pro 3.0	39
	3.3.2	Gambit Preprocessor	40
	3.3.3	Fluent 5.3	40
	3.4	CFD Credibility and Implementation	41
	3.5	Aircraft Wing External Geometry	42
		Digitization	42
	3.6	Wind Tunnel Facilities	46
	3.6.1	Pressure Distribution	47
	3.6.2	Aerodynamic Forces	48
CHAPTER IV	TWO DIMENSIONAL SUBSONIC FLOW		48
		OVER NACA 0012 AIRFOIL - CASE	
		VALIDATION 1	
	4.1	Introduction	48
	4.2	Experimental Parameters	49
	4.3	Simulation Methodology	49

CHAPTER IV	4.4	Space Discretization	52
	4.5	Numerical Treatment:	53
	4.5.1	Wall Treatment /Near Wall Meshes	54
	4.5.2	Convergence Judgement	55
	4.6	Results and Discussion	59
	4.6.1	Pressure Distribution Prediction & Discussion	62
	4.6.2	Aerodynamic Forces	63
	4.6.3	Wall Interference Assessment	65
	4.6.4	Flow Over NACA 0012 At Mach Number of 0.4 and 0.067, Zero Incidence	66
	4.7	Airfoil Critical Number	68
	4.8	Stall Angle	69
	5.4	Results	76
	5.4.1	Experimental Results	76
CHAPTER V		TRANSONIC FLOW OVER RAE 2822 AIRFOIL – VALIDATION CASE 2	70
	5.1	Introduction	70
	5.2	Experimental Flow Conditions	71
	5.3	Simulation Strategy and Numerical Treatments	71
	5.4	Results and Discussion	76
	5.4.1	Pressure Distribution	77
	5.4.2	Aerodynamic Forces	81
	5.5	Summary	82
	6.3.4	Difference in Simulated and Experimental Results	106
	6.5	Transf Wall Interference	107
	6.7	Convergence and Grid Generation	108

**CHAPTER VI LOW SPEED FLOW OVER “WING
SECTION-STORE MODEL” –
EXPERIMENTAL AND SIMULATION**

6.1	Introduction	84
6.2	Wind Tunnel Experiment	85
6.2.1	Wing Model	85
6.2.2	Model Machining and Construction	87
6.2.3	Interconnection for Wing, Pylon and Store	90
6.2.4	Wind Tunnel Testing	91
6.3	CFD Simulation of Wind Tunnel Configuration	93
6.3.1	Space Discretization	93
6.3.2	Simulation Methodology	95
6.4	Results	98
6.4.1	Experimental Results	98
6.4.2	Simulation Results	100
6.4.2.1	Aerodynamic Forces	101
6.4.3	Correlation Between Experimental and Simulation	102
6.5	Results Analysis and Discussion	103
6.5.1	External Store Aerodynamic Interference	103
6.5.2	Simulation and Experimental Results Validation	104
6.5.3	Deficiency in Experimental Model	105
6.5.3	Deficiency in Simulation	105
6.5.3.1	Modeling	105
6.5.4	Different in Simulated and Experimental Results	106
6.6	Tunnel Wall Interference.	107
6.7	Convergence and Grid Generation	108

6.8	Summary	109
-----	---------	-----

CHAPTER VII HAWK 208 AIRCRAFT SIMULATION

7.1	Introduction	110
7.2	Hawk 208 and Computational Model	111
7.3	Computational Mesh and Domain	112
7.4	Numerical Simulation and Treatments	114
7.6	Results and Discussion	116
	7.6.1 External Rocket Launcher	116
	Interference and Effects at Mach	
	0.6 and Zero Incidence	
7.7	Effect of Aircraft Speed at $\alpha=0$ on	121
	Aerodynamic Interference.	
7.8	Effect of Aircraft Incidence on The	124
	Aerodynamic Interference	
7.9	The Effect of External Store on Flow	128
	Around The Tail and Stabilizer	
7.10	Aerodynamic Forces	131
7.11	Convergence Rate	132
7.12	Summary	133

CHAPTER VIII CONCLUSION AND RECOMMENDATION

8.1	Summary	134
8.2	Conclusion	136
	8.2.1 Aerodynamic Interference	136
	8.2.2 CFD Validation	137
8.3	Recommendation	138

REFERENCES	140
------------	-----

LIST OF TABLES

TABLE NO.	TITLE	PAGE
2.1	Summary of approximation levels	24
4.1	Simulations parameters and setting	58
4.2	Simulated lift and drag coefficients	64
5.1	Comparison of predicted C_L and C_D between experimental and various flow model.	82
6.1	Simulated aerodynamic forces	101
7.1	Comparison of lift and drag forces between clean and wing-store configuration	131
3.5	Schematic layout of wind tunnel utilized for current study.	47
4.1	Flow chart for NACA 0012 validation.	55
4.2	Computational domain and grid distribution.	57
4.3	Comparison of pressure distribution between fine and coarse mesh.	59
4.4	Convergence trend monitoring.	60
4.5	Pressure distribution predicted by compressible inviscid, viscous and RANS-RSM turbulence model.	62
4.6	Pressure distribution predicted by various incompressible RANS model.	63

LIST OF FIGURES

FIGURE NO.	TITLE	PAGE
2.1	Numerical simulation scheme	10
2.2	Various approximations and general physical modeling process.	23
2.3	Dynamic approximation levels	24
2.4	Progress in model complexity level simulated in CFD	25
3.1	Flow chart for overall study	38
3.2	Photographs of marked Hawk 208 wing surface	43
3.3	Hawk 208 geometry vertices produced with Photomodeller 3.0	44
3.4	Hawk 208 wing geometry after smoothens with AutoCAD 2000 based on vertices generated from Photomodeler.	45
3.5	Schematic layout of wind tunnel utilized for current study.	47
4.1	Flow chart for NACA 0012 validation	51
4.2	Computational domain and grid distribution	52
4.3	Comparison of pressure distribution between fine and coarse mesh.	54
4.4	Convergence trend monitoring	56
4.5	Pressure distribution predicted by compressible inviscid, laminar and RANS-RK ϵ turbulence model.	59
4.6	Pressure distribution predicted by various compressible RANS model	60

4.7	Pressure distribution predicted by compressible RANS-RK ϵ turbulence model.	61
4.8	Comparison of pressure distribution on NACA 0012 between airfoil in wind tunnel and airfoil in open air.	65
4.9	Predicted pressure distribution over NACA 0012 at Mach No 0.4	67
4.10	Predicted pressure distribution over NACA 0012 at Mach No 0.067	67
4.11	Shock capturing by Fluent 5.3. over NACA 0012 airfoil	68
4.12	Velocity contour and particles tracking released from 15% chord for NACA 0012	69
5.1	Computational domain and grid near the airfoil region	74
5.2	Simulation flow chart for RAE 2822 case study	75
5.3	Distribution of pressure coefficient over RAE 2822	76
5.4	Turbulence kinetic energy at leading and trailing edge.	79
6.1	Conceptual design of experiment wing model	86
6.2	Wing model fabrication	88
6.3	Completed wing section, pylon and rocket launcher model	89
6.4	Connection mechanism between the wing/pylon and store.	90
6.5	Model of clean wing and wing/store configuration positioned in the wind tunnel test section.	92
6.6	Wing meshing	94
6.7	Pressure distribution at mid span for top and lower wing surface	95
6.8	Flow chart for wind tunnel simulation	97
6.9	Graphs of pressure distribution for various spanwise station surrounding the store	99
6.10	Interference effect on pressure distribution at various stations	100

6.11	External store interference on pressure distribution in wing leeward direction.	101
6.12	Comparison of pressure distribution on wing surface between corrected experiment results and the simulated results	102
6.13	Wall interference effect	108
7.1	Mesh for Hawk 208 modified – clean configuration	113
7.2	Mesh for Hawk 208 installed with LAU 5003 rocket launcher at the inboard station	113
7.3	Schematic diagram showing the simulation flow	115
7.4	Top wing surface iso-bar contour	117
7.5	Lower wing surface iso-bar contour	118
7.6	Velocity profile at station 25cm from store center	118
7.7	Pressure distribution at different span-wise location	120
7.8	Effect of velocity on span-wise aerodynamic interference	121
7.9	Effect of velocity on chord-wise aerodynamic interference	123
7.10	Effect of incidence on chord-wise aerodynamic interference.	125
7.11	Effect of incidence on span-wise aerodynamic interference at quarter chord	126
7.12	Effect of incidence on span-wise aerodynamic interference at three quarter chord	127
7.13	Comparisons of tangential velocity at the tail area between clean and store configurations.	128
7.14	Comparison of tangential velocity at the tail area between clean and store configuration at 5 degrees incidences	129
7.15	Particles flow over the wing, released from front of store, leading edge and wingtip	130

ρ	Density
τ	Time
U	Velocity
u	Velocity component in x direction
v	Velocity component in y direction
w	Velocity component in z direction
γ	Specific heat ratio
μ	Viscosity coefficient
ν	Kinematics viscosity
ρ	Density

NOMENCLATURE

E	Energy
i	Internal energy
K	Kinetic energy
M	Mach number
m	Molar mass
p	Pressure
R	Universal gas constant
s	Entropy
T	Temperature
t	Time
U	Velocity vector
u	Velocity component in x direction
v	Velocity component in y direction
w	Velocity component in z direction
C _v	Constant volume
C _p	Constant pressure
Re	Reynold number
Pr	Prandtl number
S _{u,v,w}	Source in x, y and z direction
div	Divergence
grad	Gradient
u'	Fluctuating velocity in x direction
v'	Fluctuating velocity in y direction
w'	Fluctuating velocity in z direction
T ₀	Total temperature
α	Incidence angle
ε	Dissipation
φ	Scalar
Φ	Mean scalar
φ	Fluctuating scalar
φ'	Time varying scalar
γ	Specific heat ratio
ψ	Stream function
μ	Viscosity coefficient
ν	Kinematics viscosity
ρ	Density

ρ_0	Stagnation density
τ	Stress
δ	Del
ω	Vorticity
l	Characteristic length
λ	Second viscosity

CHAPTER I

INTRODUCTION

1.1 Introduction

Modern fighter aircraft are heavily designed to carry stores which are stored in a bay or are mounted externally, for example under the wing. The flowfield around such a store is affected by the surrounding components such as the wing, the engine and the control surfaces, and changes considerably. The flow phenomena that may be initiated or introduced around a store include local shock waves, flow separation and turbulence. These phenomena may extend downstream and affect other aircraft components such as the horizontal and vertical stabilizer and hence the controllability and the stability of the aircraft.

Before a store can be carried on an aircraft carriage, a store clearance program needs to be conducted. The aim of the store clearance program is to ensure that the store is safe to be carried and/or released without causing stability and controllability difficulties to the aircraft and the store, and to achieve its objective of release.

Store clearance studies may include many areas such as aerodynamics, structural, flight, physical integration, trajectory and release, aircraft performance and stability analysis. The overall scope is wide and involves multiple disciplines.

CHAPTER I

INTRODUCTION

1.1 Introduction

Modern fighter aircraft are mostly designed to carry store externally. When a store is installed externally, for example under the wing, the flowfield on its surrounding components such as the wing, the engine and the control surfaces will change considerably. The flow phenomena that may be induced or introduced could include local shock waves, flow separation and turbulence. These phenomena may extend downstream and affect other aircraft components such as the horizontal and vertical stabilizer and hence the controllability and the stability of the aircraft.

Before a store can be certified for aircraft carriage, a comprehensive store clearance program needs to be conducted. The aim of the store clearance program is to ensure that the store is safe to be carried and/or released without causing any stability and controllability difficulties to the aircraft and the store besides achieving its objective of release.

Store clearance studies may include many areas such as aerodynamic, structure, flutter, physical integration, trajectory prediction, aircraft performance and stability analysis. The overall scope is wide and involves multiple engineering

disciplines. Ability to identify the changes in aerodynamic characteristics are perhaps the most critical and important. It is a prerequisite for other analysis, i.e. the aerodynamic loads data are required for subsequent aircraft structural, stability and performance analysis and the store trajectory prediction.

Aerodynamic change investigation in the external store clearance studies usually involved complex geometry (multi components with mutual interference) and complex flow field (two or more dominant flow phenomenon in a single flow). Traditionally, flow of such nature was investigated through wind tunnel testing beside empirical methods.

In recent year, Computational Fluid Dynamics (CFD) simulation has come into practice in various aerodynamics study including store clearance. CFD is basically a theoretical method using computational procedure to solve the universal conservation laws those govern the fluid flow. Latest development in this field had seen the CFD been integrated with computational structural analysis code for interdisciplinary analysis [1]. References [2,3] presented methods of integrating CFD and six degrees of freedom (6 DOF) dynamics simulation in store trajectory analysis.

1.2 Research Objective

1.2.1 Research Methodology

- (i) To identify the aerodynamic interference effect to the present of external store using theoretical method, which are suitable in environment where the wind tunnel facilities and empirical data are not available.
- (ii) To investigate the feasibility of using commercially available general purpose CFD code to identify the aerodynamic forces and characteristics as a result of the mutual aerodynamics interference between the external store and wing.

1.3 Research Scopes

- (i) To carry out literature review on methods in determining aircraft wing and external store aerodynamics interference.
- (ii) To determine the suitability of applying a commercial CFD code for predicting aerodynamic interference involving store-wing configuration at subsonic flight.
- (iii) To explore the use of a 3 Dimensional measurement software to extract measurement from photographs, i.e. Photomodeler for Hawk 208 wing geometry digitization.
- (iv) To construct a pressure model of an aircraft wing installed with external store and pylon and carried out wind tunnel testing.
- (v) To identify the aerodynamic interference effect when a Rocket Launcher pod LAU 5003, installed externally to a simplified HAWK 208-support fighter model.

1.4 Research Methodology

1.4.1 Outline of Thesis

This research includes comprehensive literature review on progresses in CFD application especially in complex aerodynamic interference study. Followed with validation on a commercial CFD code, i.e. Fluent 5.3 by Fluent Inc. The validation begins with simulations of a simple two dimensional, subsonic and transonic flow over NACA 0012 and RAE 2822 airfoil, respectively. Simulated results were validated with the reported data in literature.

Subsequently, wind tunnel experiments on a pressure model built based on a section of Hawk 208, support fighter's wing installed with external store were carried out. These experimental configurations were then numerically simulated using the CFD method. The results obtained from experiment and simulation were compared and formed second stage of CFD code validation.

Finally, a simplified full-scale Hawk 208 installed with LAU 5003 Rocket Launcher Pod was simulated at different flight condition. This would help to identify the effect of the external store aerodynamic interference on flowfield around the wing specifically and overall system generally.

1.5 Expected Results

It is expected that, the simulation results will be satisfactory for preliminary investigation involving aerodynamic interference at subsonic flight. The simulation results, supplemented with the experimental results, will reveal some facts on the nature of aerodynamic interference for wing-store configuration under present study.

1.6 Outline of Thesis

Chapter one generally introduces the outline of the research. Followed by literature review in chapter two, review was emphasized on the nature of problem, application of CFD method in aerodynamic interference study and the development in CFD especially in aerodynamic application.

Chapter three outlined the general research methodology and the Hawk 208 wing geometry digitization processes utilizing computer program.

Chapter four presents a validation case in which a subsonic flow over NACA 0012 airfoil was simulated. The simulation results were validated with the experimental results reported in literature.

Chapter five extends the two dimensional validation into the transonic flow regime where a transonic flow over RAE 2822 airfoil was simulated. The simulation results were again validated with the experimental results.

Chapter six outlines wind tunnel experiment of low speed flow over a wing-store configuration model. The model was built based on digitized Hawk wing section complete with external store. These wing-store configurations were then simulated using CFD method.

Chapter seven presents the final simulation study in which simplified full scales Hawk 208 aircraft flies at various angle of attack and speed were simulated. Aerodynamic interference arise from external store installation were analyzed extensively. The thesis ends with conclusion and recommendation in chapter eight.

The importance of a proper study of the aerodynamic interference with external store carriage is as explained by Yip and Norman [3]. "Besides the change of "load factor" on store released from aircraft, the proper understanding and design may minimize or even reverse the position of L/D which was associated with the intrusion of spanwise flow and effective angle of attack of the store arrangement".

There are a lot of theoretical and mathematical models with various approximations available for describing various nature of flow phenomenon. When it comes to the problem of complex flow over complex geometry such as wing-store

7. E.A.T. Christopher and C.T. Snow (1981). "The Use of Multipole for Calculating the Aerodynamic Interference Between Bodies Of Revolution". *Journal of Aircraft*, Vol. 18, No 9, 671-679.

8. A. Cecko and E.N. Tuozzo and R.D. Dyer and J. Dejongh (1982). "Pan Air Applications to Weapon Configuration Database". *Journal of Aircraft*, Vol. 19, 128-134.

REFERENCES

9. Alexander, E.N., Thomas and J. Tumanyan (1984). "PAN AIR application to...

1. Manoj K. Bhardwaj, Rakesh K. Kapania, Eric Reichenbach and Guru P. Guruswamy (1998). "Computational Fluid Dynamics/Computational Structural Dynamics Interaction Methodology For Aircraft Wings". *AIAA Journal*, Vol. 36, No 12, 2179-2185.

2. Robert F. Tomaro, Frank C. Witzeman and William Z. Strang (2000). "Simulation Of Store Separation For The F/A-18c Using Cobalt". *Journal of Aircraft*, Vol. 37, No 3, 361-367.

3. Nathan C. Prewitt, Davy M. Belk and Raymond C. Maple (1999). "Multiple-Body Trajectory Calculations Using the Beggar Code". *Journal of Aircraft*, Vol. 36, No 5, 802-808.

4. James M. Brock, Jr and Bruce A. Jolly (1988). "Application of Computational Fluid Dynamics at Eglin Air Force Base". SAE 985500.

5. Vijaya Shanker and Norman Malmuth (1981). "Computational And Simplified Analytical Treatment Of Transonic Wing/Fuselage/Pylon/Store Interaction". *Journal of Aircraft* vol. 18, No 8, 631-637.

6. John T. Bahna, David A. seidal, Samuel R. Bland and Robert M. Bennett (1989). "Unsteady Transonic Flow Calculation for Realistic Aircraft Configurations". *Journal of Aircraft*. Vol. 26, No 1, 21-28.

7. P.A.T Christopher and C.T Shaw (1981). "The Use of Multipole for Calculating the Aerodynamic Interference Between Bodies Of Revolution". *Journal of Aircraft*, Vol. 21, No 9, 673-679.
8. A.Cenko and E.N Tincoco and R.D.Dyer and J. Dejongh (1982). "Pan Air Applications to Weapons Carriage and Separation". *Journal of Aircraft*, Vol. 18, 128-134.
9. Alex cenko, E.N. Tincoco and, J. Tustaniwskyj (1984). "PAN AIR application to Mutual Interference Effects". *Journal of Aircraft*. Vol. 21, No 10.
10. N. Singh (1990). "Incompressible Potential Flow about 3D Configuration". *Journal of Aeronautical Society of India*. Vol. 42, No 2, 129-133.
11. Alex Cenko (1983). "Pan Air Application to Complex Configurations". *Journal of Aircraft*, Vol. 20, No 10. 887-892.
12. Lillianne P. Troeger and Gregory V. Selby (1998). "Computation of the Aerodynamic Characteristics of A Subsonic Transport". *Journal of Aircraft*, Vol. 35, NO 2. 183-190.
13. Jie Li, Fengwei Li And Qin E (2000). "Numerical Simulation of A Transonic Flow over Wing-Mounted Twin-Engine Transport Aircraft". *Journal of Aircraft*, Vol. 37, No 3. 469-478.
14. Paresh Parikh, Shahyar Pirzadeh and Neal T. Frink (1994). "Unstructured grid solutions to a wing/pylon/store configuration". *Journal of Aircraft*. Vol. 31, No 6, 1291-1296.
15. C.C Rossow, J.L Godard, H. Hoheisel and V. Schmitt (1994). "Investigation of propulsion integration interference effects on a transport aircraft configuration". *Journal of Aircraft*. Vol. 31, No 5, 1022-1030.

16. Lawrence E. Lijewski and Norman E. suhs (1994). "Time-accurate CFD approach to transonic store separation trajectory prediction". *Journal of Aircraft*. Vol. 31, No 4, 886-891.
17. Tsze C. Tai (2000). "Effect Of External Components On V-22 Aircraft Forward Flight Aerodynamics". *Journal Of Aircraft*, Vol 37, No 2. 201-207.
18. Ismail H. Tuncer and Max F. Platzer (1998). "Computational Study of Subsonic Flow over A Delta Canard-Wing-Body Configuration". *Journal of Aircraft*, Vol. 35, No 4. 554-561.
19. Susumu Takanashi and Masami Takemoto (1994). "The Latest Progress In He Computational Aerodynamic System For Practical Aircraft Configuration". SAE 940061.
20. Eugene I. Tu (1992). "Navier-Stokes simulation of a closes-coupled Canard-Wing-Body configuration". *Journal of Aircraft*. Vol. 29, No 5, 830-837.
21. Rusell M. Cummings, Yehia M.Rizk, Lewis B. Schiff and Neal m. Chaderjian (1992). "Navier-Stokes prediction for F18 wing and fuselage at large angle of attack". *Journal of Aircraft*. Vol 29, No 4, 565-574.
22. Hirsch (1988). "Numerical Computation of Internal and External Flows". Volume 1. Brussels: Wiley.
23. E. Rathakrishnan. (1995). "Gas dynamics". Prentice -hall of India private limited New Delhi-110001.
24. Shih-I Pai and shijun luo (19). "Theoretical And Computational Dynamics Of A Compressible Flow". Van Nostrand Reinhold & Science Press, Beijing.
25. John D. Anderson (1990). "Fundamentals of Aerodynamics". McGraw Hill, Inc, New York.

26. H.K Versteeg & W. Malalasekera (1995). "An introduction to Computational Fluid Dynamic - Finite Volume method". London: Longman.
27. G.Barakos and D. Drikakis (2000). "Investigation of Nonlinear Eddy-Viscosity Turbulence Models in Shock/Boundary-Layer Interaction". AIAA Journal, Vol. 38, No 3, 461-469.
28. Javier Jimenez and Robert D. Moser (2000). "Large-Eddy Simulations: Where Are We And What Can We Expect?" AIAA Journal. Vol. 37, No 4, 605-612.
29. Sandip Ghosal (1999). "Mathematical and Physical Constraints on Large-Eddy Simulation of Turbulence". AIAA Journal, Vol. 37, No 4. 425-433.
30. S.L Woodruff, J.M Seiner and M.Y. Hussaini (2000). "Grid Size Dependence in the Large-Eddy Simulation of Kolmogorov Flow". AIAA journal, vol. 38, No 4, 600-604.
31. F.R. Menter (1994). "Two Equation Eddy-Viscosity Turbulence Models For Engineering Applications". AIAA Journal, Vol. 32, No 8, 1598-1603.
32. S. Duranti and F. Pittaluga (2000). "Navier-Stokes Prediction of Internal Flows with A Three-Equation Turbulence Model". AIAA Journal, Vol 38, No 6, 1098-1102.
33. Shia-Hui Peng and Lars Davidson (2000). "New Two-Equation Eddy Viscosity Transport Model For Turbulent Flow Computation". AIAA Journal, Vol. 38, No 7, 1196-1205.
34. David C. Wilcox (1994). "Simulation of Transition with A Two-Equation Turbulence Model". AIAA Journal, Vol. 32, No 2, 247-255.
35. Jack R. Edwards and Suresh Chandra (1996). "Comparison of Eddy Viscosity-Transport Turbulence Models for Three-Dimensional, Shock-Separated Flowfields". AIAA Journal, Vol. 34, No 4, 756-763.

36. Arthur Rizzi and Jan Vox (1998). "Toward Establishing Credibility in Computational Fluid Dynamics Simulations". AIAA Journal, Vol. 36, No 5, 668-675.
37. Antony Jameson (1999). "Re-Engineering the Design Process through Computation". Journal of Aircraft, Vol. 36, No 1.
38. Lawrence W. Spradley, Rainald Lohner and T.J. Chung (1996). "Generalized Meshing Environment for Computational Mechanics". AIAA Journal, Vol. 36, No 9. 1735-1737.
39. Dimitri J. Mavriplis (2000). "Viscous Flow Analysis Using A Parallel Unstructured Multigrid Solver". AIAA Journal, Vol. 38, No 11, 2067-2075.
40. Yannis Kallinderis, Aly Khawaja and Harlen McMorris (1996). "Hybrid Prismatic/Tetrahedral Grid Generation for Viscous Flows around Complex Geometries". AIAA Journal, Vol 34, No 2. 291-298.
41. R.P. Koomullil and B.K. Soni (1999). "Flow Simulation Using Generalized Static and Dynamic Grids". AIAA Journal, Vol. 37, No 12, 1551-1557.
42. P.A.Henne (1990). Applied Computational Aerodynamics, Progress In Astronautics And Aeronautics" vol. 125. 91-130, American Institute Of Aeronautics and Astronautics Inc. Washington, DC.
43. W.G. Habashi, J. Dompierre, Y. Bourgault, M. Fortin and M.G. Vallet (1998). "Certifiable Computational Fluid Dynamics through Mesh Optimization". AIAA Journal, Vol 36, No 5, 703-711.
44. Richard J. Smith and Leslie J. Johnston (1996). "Automatic and Flow Solution for Complex Geometries". AIAA Journal, Vol 34, No 6. 1120-1124.
45. Patrick J. Roache (1998). "Verification of Codes and Calculations". AIAA Journal, Vol. 36, No 5, 696-701.

46. Jack R. Edwards and James L. Thomas (2000). "Development of a (Nm2) Preconditioned Multigrid Solvers for Euler and Navier-Stokes Equations". AIAA Journal, Vol 38, No 4, 717-720.
47. Heng Lin and Ching-Chang Chieng (1999). "Computations Of Transonic Turbulent Flow Past Airfoils Using Multigrid/Bi-Cgstab Algorithm". AIAA Journal, Vol. 37, No 1, 118-124.
48. John J. Bertin and Micheal L. Smith (1998). "Aerodynamic for Engineer, 3rd Ed". Prentice-Hall, N.J.
49. Patrick H. And William E. Carscallen (1997). "Compressible Fluid Flow". Mcgraw-Hill, New York.
50. William L. Oberkampf and Frederick G. Blottner (1998). "Issues In Computational Fluid Dynamics Code Verification and Validation". AIAA Journal, Vol. 36, No 5, 687-695.
51. G. Lombardi, M.V. Salvetti and M. Morelli (1998). "Appraisal of Numerical Methods in Predicting the Aerodynamics of Forward-Swept Wings". Journal of Aircraft, Vol. 35, No 4, 561-567.
52. James L. Amick (1950). "Comparison of the experimental pressure distribution on an NACA 0012 profile at high speeds with that calculated by the relaxation method". Technical note 2174, National Advisory Committee for Aeronautics.
53. IRA H.Abbott and Albert E. Von Doenhoff (1959). "Theory of wing sections". Dover Publications, inc. New York.
54. Fluent 5.3 Online Help Manual.
55. John A. Ekaterinaris (2000). "Implicit High-Order-Accurate-In-Space Algorithms for the Navier-Stokes Equations". AIAA Journal, Vol 38, No 9, 1594-1602.

56. D.P. Coiro, P. De Matteis and M. Amato (1992). "Wake Effects On the Prediction of Transonic Viscous Flows around Airfoils". *Journal of Aircraft*. Vol. 29, no 3, 437-443.
57. Linda D. Kral, Mori Mani and John A. Ladd (1996). "Application of Turbulence Models for Aerodynamic and Propulsion Flowfields". *AIAA Journal*, Vol. 34, No 11. 2291-2298.
58. C.C Chuang and C. C. Chieng (1994). "Comparative Study of Higher Order Turbulence Models for Compressible Separated Flows". *AIAA Journal*, Vol. 32, No 8, 1740-17443.
59. G. Jin and M Braza (1994). "Two-Equation Turbulence Model For Unsteady Separated Flows Around Airfoil". *AIAA Journal*, Vol 32, No 11, 2316-2319
60. Hans W. Stock and Werner Haase (2000). "Navier-Stokes Airfoil Computations With En Transition Prediction Including Transitional Flow Regions". *AIAA Journal*, Vol 38, No 11, 2059-2066.
61. William E. Milholen II, Ndaona Chokani and J. Al-Saadi (1996). "Performance of Three-Dimensional Compressible Navier Stokes Codes at Low Mach Numbers". *AIAA Journal*, Vol 34, No 7, 1356-1362.