ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Airframe Engine Integration Aerodynamic Studies Using CFD Simulations

M Sai Sri Soumya
MLR Institute of Technology and Management,
Department of Aeronautical Engineering,
Hyderabad, India.

R.Srawanthi
Mahaveer Institute of Science and Technology,
Department of Aeronautical Engineering,
Hyderabad, India.

V.Vamshi
MLR Institute of Technology and Management,
Department of Aeronautical Engineering,
Hyderabad, India.

Abstract:

Airframe – Engine integration performance is significantly different from the performance of individual components like wing and engine for the given values of Mach number, angle of attack and power setting. So, focus in the area of integration studies of airframe – engine must be carried out for the aerodynamic better performance in the regards of low cruise drag, better climb rate, and long range and endurance flights. A CFD simulation study will be carried out in this area as CFD studies plays and extensive role in the aircraft design. In this thesis, numerical simulations will be carried out on a civil transport airplane wing – engine configuration in the cruise Mach number range of 0.73 to 0.85 using ANSYS CFD package. Based on the flow field visualizations, the physical phenomenon that takes place in engine/airframe integration will be discussed.

Key words: ***low cruise drag, climb rate long range and endurance flights.***

1. Introduction:

The design of a new engine for a subsonic civil transportation aircraft has to be done taking into consideration several points of view. Every aspect considered will have an impact on the final user. For airline companies, fuel economy is one of the most important issues to consider for the selection of a new aircraft. For this reason, aircraft manufacturers present different options of engine suitable for the final aircraft system. Each engine manufacturer offers different options of performance, not only for structural and flight requirements, but also economical issues. Environmental issues are other important aspects to consider in the aeronautic industry because the regulations of noise and emission levels are becoming very demanding.

1.1 Environmentally Friendly Aero Engine (VITAL):

VITAL will provide a breakthrough in noise and emissions engine architecture. This breakthrough will be achieved by the development of new commercial engines taking advantage of the novel lightweight and low noise technologies. With this improvement the SFC is going to be reduced without having an impact on noise and emissions.

VITAL will design, manufacture and rig test three novel fan architectures of turbofan engines for commercial aircraft. These new technologies (Direct Drive Turbofan, Geared Fan Turbofan and Contra-Rotating Turbofan) will provide a breakthrough in noise and emission reduction.

VITAL will provide a major advance in developing the next generation of commercial aircraft engines. The aim of VITAL is the development and validation of engine technologies that alone will provide the following:
1.6 dB noise reduction
2.7% reduction in CO2 emissions

1.2 Objectives of the project

The main objectives of the project are the following:
1.Create a methodology for modelling the behaviour of the drag produced by an installed VHBR engine.
2.To simulate the behaviour of the flow around a nacelle for a VHBR engine at different flight stages and using different geometries for each of the nacelle components.
3.To link drag force with the different parameters considered in the project. These parameters are divided in two categories: nacelle geometry parameters (including Dmax, Dhigh, length, fore body length and scarf angle) and distance to the wing (including nacelle

penetration and vertical distance to the wing). This relationship will allow modeling a correlation of the drag with some parameters related to the installation.

1.3 Contribution to knowledge

The concept of VHBR is currently under development and different issues arise from the research being carried out. One of the main difficulties of this concept is the size of the fan. Some of the most important variables involved in the overall design are the weight and the frontal area of the fan. The integration of the engine to the airframe is essential to take advantage of the benefits obtained from the VHBR concept.

The integration of a VHBR engine requires a highly coupled Wing-Body-Nacelle-Pylon system to avoid the production of undesirable effects. Also, when on the ground, the space available to install the engine is very limited making integration of big engines, like a VHBR engine, a very complex task. For a detailed analysis of a VHBR integration methodology see the work carried out by Berry (1994).

This novel project consists of a study about the influence of the installation of a VHBR engine on the installed thrust by evaluating the drag produced by the engine.

2. Literature review:

One of the most important advantages of the Gas Turbine Engine over other power sources is the high power/weight ratio. This is one of the main reasons why in 1929 Sir Frank Whittle submitted to his superiors his idea of using a gas turbine engine instead of piston engines to power aircraft (The Jet engine, 1986). He built his first experimental engine in 1937. From that point on the technologies used in Gas Turbine Engines have been improving to get a better performance.

The performance requirements of the engines vary depending on the type of mission the aircraft will be designed for. For a civil aircraft engine design thrust and fuel consumption are two of the most important parameters to be considered from the performance point of view. The range of an aircraft is defined by the fuel consumption and fuel capacity. Specific Fuel Consumption in turbofan engines is lower than in turbojet engines.

Some of the improvements of these characteristics would be at the expense of others therefore the design process requires a balance of the variables depending on the requirements of the whole system.

The design of the after body is focused on the nozzle. For subsonic aircraft the nozzle used is a convergent nozzle due to the low pressure ratio across it. Convergent-divergent nozzle is a very expensive technology due to the extra weight added to the nacelle and the low benefit obtained from it for a subsonic commercial aircraft (Cohen, 1987). The main concern of nozzle designers is thrust, but the boundary layer separation and noise production are also very important. A bad design would produce an early separation of the boundary layer decreasing the pressure behind the nacelle and deliver more drag.

3. Methodology:

Geometric Model Design

The first stage of the project is based on 3D simulations of the wing. Wing is analyzed at different angles of attacks and speeds.

The model generated is a wing for a VHBR. The overall dimensions of the wing were given by DLR F6 Aircraft. These dimensions are for a real nacelle, therefore the keel and crown have different values for many of the geometric parameters, mainly for the inlet. Special attention should be given to boundary layer separation in the generated model.

The consideration for the base design values given below:

- Intake tip diameter is the average of both the keel and crown diameters given by VITAL.
- The throat base diameter is set to keep the same throat area. It is circular with the centre between the keel and the crown. The average of the diameter of the throat at both points is the diameter of the axisymmetric model.
- The outside diameter is the average diameter of both maximum diameters.
- The nozzle outlet diameters were set according to the area required by the cycle design.
- For the length of the intake cowl, an average of the lengths of the keel and crown given was taken.
- Also the inner lengths were given and were different. An average was used.
- The other dimensions were taken from those provided by VITAL or assumed.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

A NACA-1 profile was chosen for the external shape of the fan, a super ellipse for the lip and a sp-line for the nozzle cowl and the inner part of the intake.

Analysis:

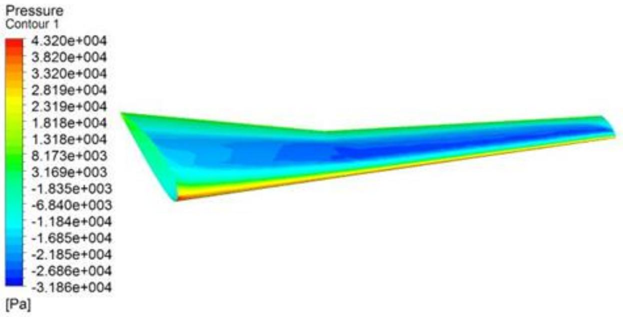
Present CFD analysis is carried out in ANSYS CFX. Boundary conditions are defined in CFX pre and CFX solver runs the solution and results are tabulated from CFD POST.

Analysis is carried at following cases:

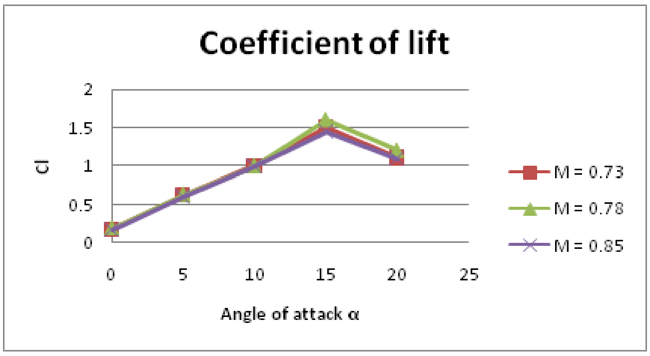
S.No	Mach Number	Angle of attack
1	0.73	0^0, 5^0 ,10^0 and 15^0
2	0.78	0^0, 5^0 ,10^0 and 15^0
3	0.85	0^0, 5^0 ,10^0 and 15^0

Result and discussions:

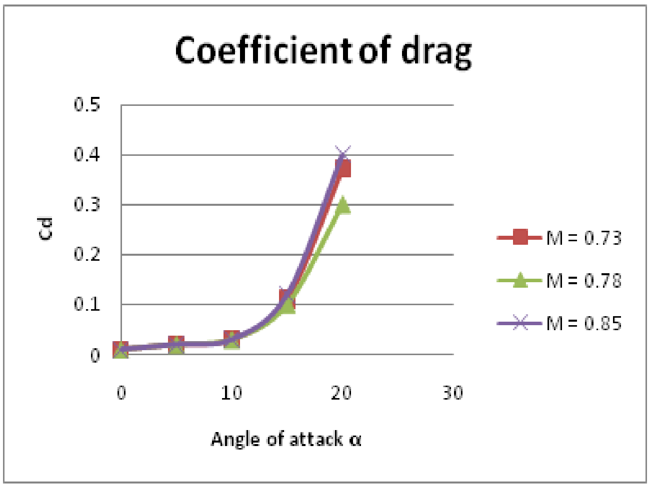
M = 0.73 AT 50 ANGLE OF ATTACK



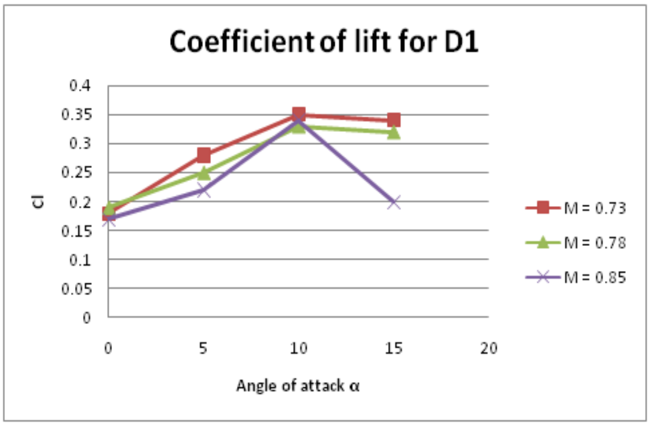
The pressure contour at 0.73 mach and AOA at 50 is max at 4.320e+0.04 and min at -3.186e+0.4



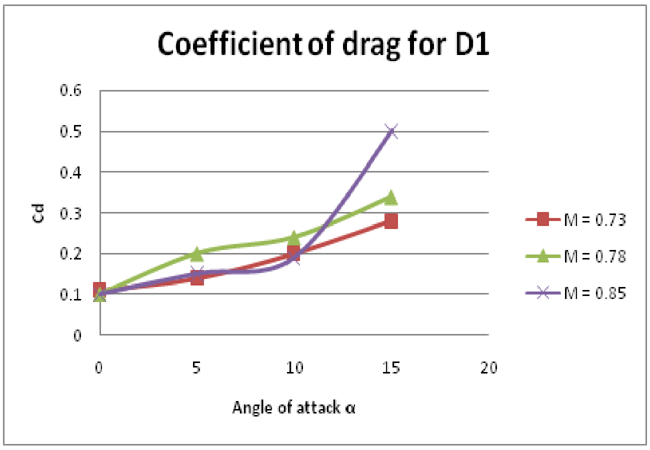
Coefficient of lift graph atr various speeds



Coefficient of drag at various speeds



When the engine diameter is 50m the CL Vs AOA graph shows us as the AOA increases CL also increases

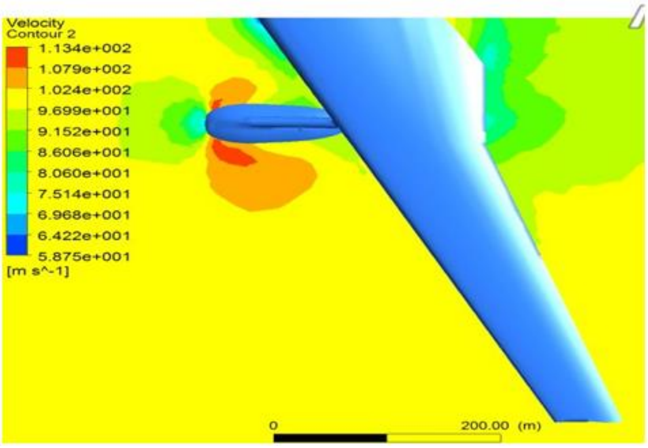


When the engine diameter is 50m the CD Vs AOA graph shows us as the AOA increases CD also increases

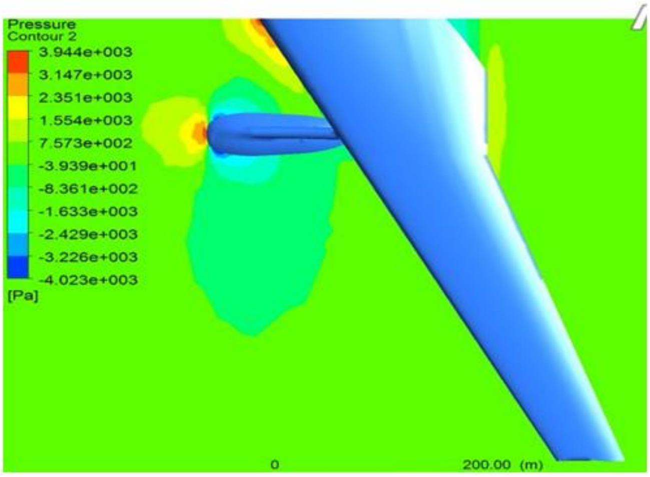
ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Airframe Engine Pylon Integration results

M = 0.73 AT 00 DEG ANGLE OF ATTACK



Velocity contour of engine pylon at M=0.73 at 00 AOA



Pressure contour of engine pylon at 0.73 at 00 AOA

Conclusion:

Out of the work done during the project and the analysis of the results the following conclusions were taken:

- The methodology used for a parametric analysis of drag proved to be suitable based on the results obtained. The results obtained allowed important conclusions to be made that will enable the knowledge of engine/airframe interaction to progress. Future work based on the same methodology would allow more CFD results to be obtained in order to calculate mathematical approximations of drag generation of an installed VHBR engine for further applications.
- The Dmax has a great impact on both the nacelle diameter and pylon height individually. There is also a significant change on the drag, but mostly because of the change of the wet surface area. Although there is a big impact on the nacelle diameter and pylon height, the changes are opposite to each other and cancel out in most of them.
- The Dmax changes the smoothness of the pressure changes. A Dmax similar to the Phigh generates large changes of velocity and therefore will increase the spillage drag and may create shockwaves decreasing the pulling force. For the geometry and the range of values analyzed the total change of drag is very small although the change of drag on the nacelle diameter and pylon height is quite significant.
- Highlight diameter has a high impact mostly on the FB and in the pre-entry drag. There is some influence on MB but it is very small. Pre-entry drag depends mostly on the Phigh. Spillage drag is more dependant of the Phigh than Dmax. Something similar to the Dmax would happen with the changes of Phigh. A Phigh close to the Dmax will generate sudden velocity changes and could generate shockwaves increasing the drag.

The results of the simulations show how there is a critical point for the total force mainly determined by the changes of the forces on the airframe and the velocity shockwaves for large P values.

Acknowledgment:

This is an acknowledgement of the intense drive and technical competence of many individuals who have contributed to the success of project. A project is always an effective work. A successful project is a fruitful culmination of efforts by many people, some were directly involved and some others quietly encouraged and supported from the background. The project is not completed if one fails to acknowledge all these individuals who have been instrumental in successful completion of the project.

I express sincere thanks to my guide R SRAWANTHI, Asst. Professor, Department of Aeronautical Engineering, MIST for his guidance and encouragement for the successful completion of my project work.

I wish to express my gratitude to our internal guide MR. V VAMSHI Asst professor, Department of Aeronautical Engineering, for giving his encouragement at each step of our project. His guidance helped me at all the time of project and writing of this documentation.

It is my pleasure to thank all the faculty members of the Department of Aeronautical Engineering, for their esteemed suggestions and help, which were instrumental in the completion of my project.

Figures:

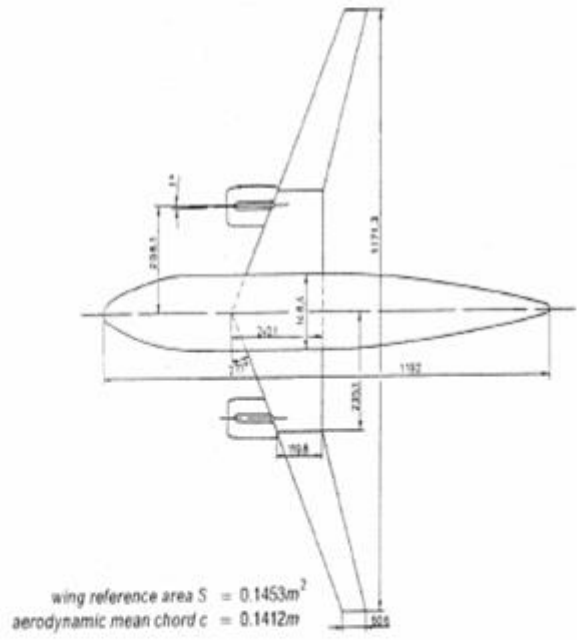
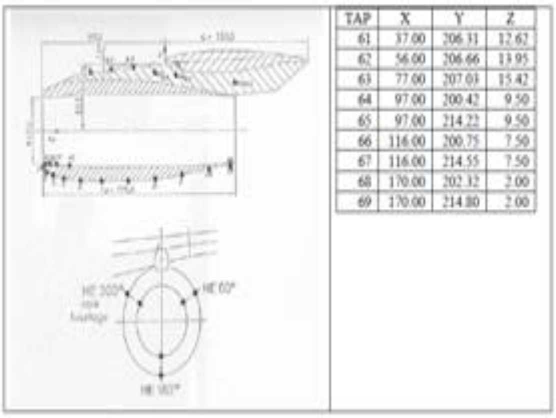


Fig 1 Geometrical dimensions of wing



TAP	X	Y	Z
61	37.00	206.31	12.62
62	56.00	206.66	13.95
63	77.00	207.03	15.42
64	97.00	200.42	9.50
65	97.00	214.22	9.50
66	116.00	200.75	7.50
67	116.00	214.55	7.50
68	170.00	202.32	2.00
69	170.00	214.80	2.00

Fig 2 Geometrical dimensions of Engine Nacelle

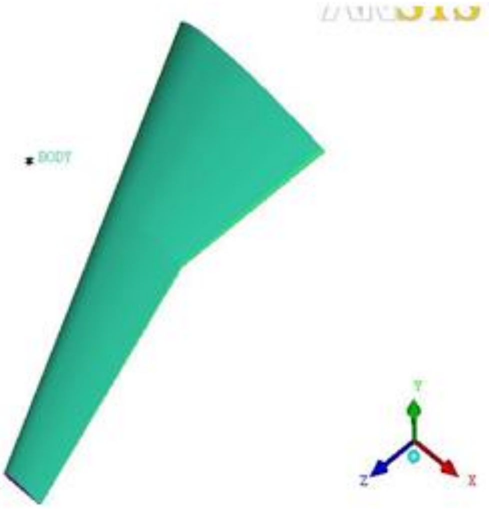


Fig 3: Wing model designed in ICEM CFD

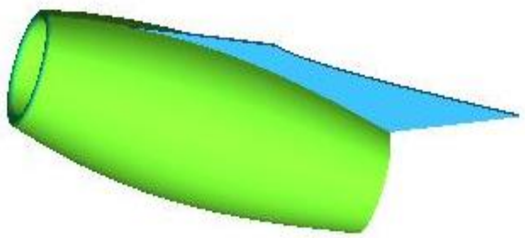


Fig 4 Engine Pylon model designed in ICEM CFD

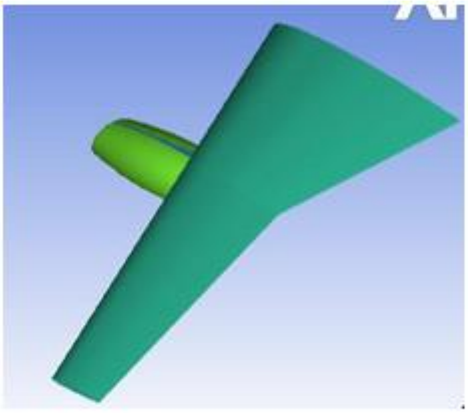
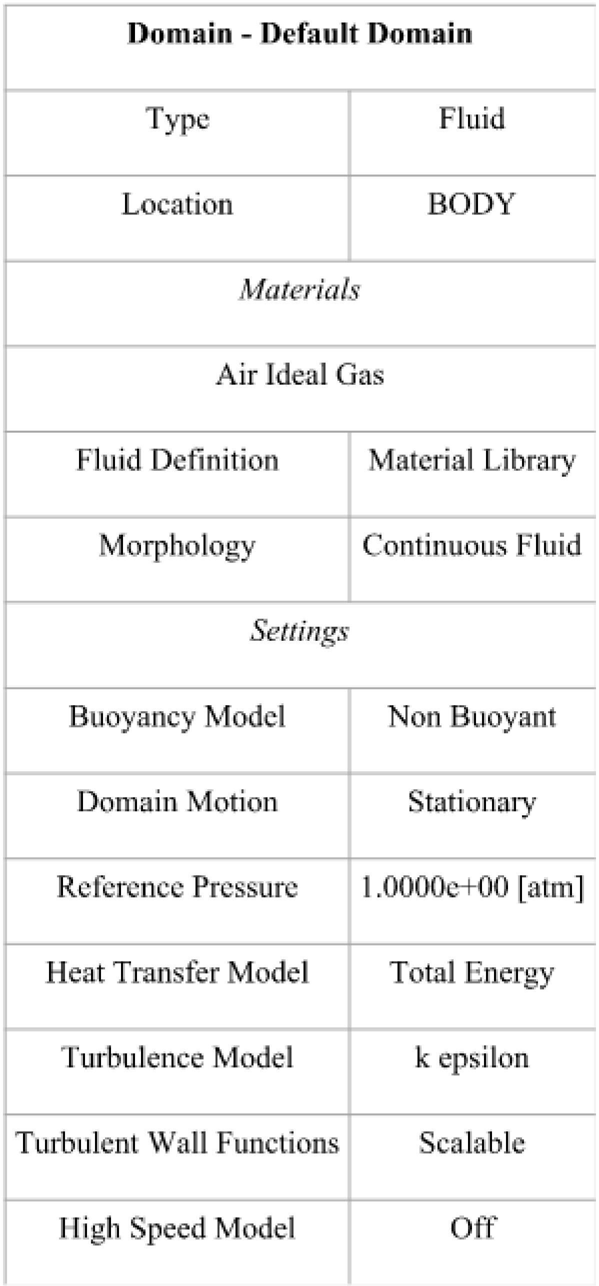


Fig 5 Airframe Engine Pylon integrated model designed in ICEM CFD

Tables:

Domain	Nodes	Elements
Default Domain	109977	584556

Table 1: mesh information



Domain - Default Domain	
Type	Fluid
Location	BODY
Materials	
Air Ideal Gas	
Fluid Definition	Material Library
Morphology	Continuous Fluid
Settings	
Buoyancy Model	Non Buoyant
Domain Motion	Stationary
Reference Pressure	1.0000e+00 [atm]
Heat Transfer Model	Total Energy
Turbulence Model	k epsilon
Turbulent Wall Functions	Scalable
High Speed Model	Off

Table 2 Domain physics

REFERENCES

1. Designing high-bypass nacelle inlets. 2005. Aerospace Engineering, 25(6), pp. 14-16.

2. NACA 1-series geometry representation for computational fluid dynamics. 1994.

3. Practical intake aerodynamic design. 1993. Washington, DC/Oxford, United Kingdom, American Institute of Aeronautics and Astronautics, Inc./Blackwell Scientific Publications.

4. The Jet Engine. 1986. 4th edition. Derby: Rolls-Royce.

5. Report of the Definitions Panel on Definitions to be Used in the Description and Analysis on Drag. 1958. ARC CP-369. London, UK: Aeronautical Research Council.

6. Report of the Definitions Panel on the Definitions of the Thrust of a Jet Engine and of the Internal Drag of a Ducted Body. 1955. ARC CP-190. London, UK: Aeronautical Research Council. AIMER, B., 2005.

7. Gas Turbine Intake Design - Peak Mach Number Response to Changes of Intake Lip Shape, (Unpublished MSc thesis), Cranfield University, Cranfield.

8. ALBERS, J.A., MILLER, B.A. and NATIONAL AERONAUTICS AND SPACE ADMINISTRATION. LEWIS RESEARCH CENTER, Cleveland, OH, 1973.

9. Effect of subsonic inlet lip geometry on predicted surface and flow Mach number distributions. NASA-TN-D-7446; E-7522; Pagination 56P.

10. ANDERSON, B., 1988. CFD application to subsonic inlet airframe integration, VKI, Intake Aerodynamics, Volume 2 59 p (SEE N89-16748 09-02); United States, 1988.

11. ANDERSON, B., GIBB, J. and NATIONAL AERONAUTICS AND SPACE ADMINISTRATION. LEWIS RESEARCH CENTER, Cleveland, OH., 1992.

12Application of computational fluid dynamics to the study of vortex flow control for the management of inlet distortion, United States, 1992, .

13. ANDERSON, B.H. and TOWNE, C.E., 1993. Application of computational fluid dynamics to inlets. Practical intake aerodynamic design (A95-18876 03-02), Washington, DC/Oxford, Inc./Blackwell Scientific Publications, pp. 395-430.

14. ANDERSON,J. D. JR., 1995. Computational fluid dynamics - The basics with applications. Inc.: New York: McGraw-Hill.

15. High speed test results of subsonic, turbofan scarf inlets, AIAA, SAE, ASME, and ASEE, Joint Propulsion Conference and Exhibit, 29th, Monterey, CA; United States; 28-30 June 1993; AIAA, SAE, ASME, and ASEE, Joint Propulsion Conference and Exhibit, 29th, Monterey, CA, 28-30 June 1993 1993, pp13.

16. ANTOINE, N.E. and KROO, I.M., 2002. Optimizing Aircraft and Operations for Minimum Noise. AIAA Paper, 5868, pp. 1-3.

17. BALKOTA, N., 2005. Gas Turbine Intake Design - Quality Evaluation of Axisymmetric Intake Flow CFD Simulations, (Unpublished MSc thesis), Cranfield University, Cranfield.

18. BELL, R.A. and CEDAR, R.D., 1991. An inverse method for the aerodynamic design of three-dimensional aircraft engine nacelles, In Pennsylvania State University, Third International Conference on Inverse Design Concepts and Optimization in Engineering Sciences (ICIDES-3) p 405-417 (SEE N92-13928 05-01); United States.

19. BENSON, T. J., 1988. CFD application to supersonic/hypersonic inlet airframe integration, VKI, Intake Aerodynamics, Volume 2 62 p (SEE N89-16748 09-02); United States, 1988.

20. BERRY, D.L., 1994. The Boeing 777 engine/aircraft integration aerodynamic design process, ICAS, Congress, 19th, Anaheim, CA, Proceedings. Vol. 2; 18-23 Sept. 1994, Washington. United States.

21. BOPPE, C.W., 1984. Elements of computational engine-airframe integration, American Institute of Aeronautics and Astronautics, Aerospace Sciences Meeting, 22nd, Reno, NV, Research supported by the Grumman Aerospace; 9-12 Jan. 1984

22. BORE, C.L., 1993. Some contributions to propulsion theory - The Stream Force Theorem and applications to propulsion. Aeronautical Journal, 97(964), pp. 138-144.

23. BRODERSEN, O., 2003. Numerical Analysis of Aerodynamic Airframe-Engine Installation Effects for Transport Aircraft.

24. BURGSMUELLER, W. and HOHEISEL, H., 2000. ENIFAIR- EU research into engine integration on future transport aircraft. Air & Space Europe, 2(2), pp. 81-85.

25. BURGSMUELLER, W., HOHEISEL, H. and DE LA PUERTA, B., 1996. European research cooperation on jet engine airframe integration, aspects of engine airframe integration for transport aircraft; Proceedings of the DLR Workshop, Braunschweig, Germany; 6-7 Mar. 1996; Cologne pp6-1 to 6-26.

26. BURGSMUELLER, W., HOHEISEL, H. and KOOI, J.W., 1994. Engine/airframe interference on transport aircraft with ducted prop fans - The European research program Duprin, ICAS, Congress, 19th, Anaheim, CA, Proceedings. Vol. 3; United States; 18-23 Sept. 1994.