International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 7, July - 2014

A Study of Lift Enhancement and Pressure Drag Reductionby Providing Ducted Passage in the Fuselage

Srinivasan P*
Lecturer, Department of Aeronautical Engineering, DSCET, Chennai, Tamilnadu, India
Dhanasekar V, Akilan K
B.E. Department of Aeronautical Engineering, DSCET, Chennai, Tamilnadu, India
Mahendiran S
M.E. Department of Aerospace Engineering, MIT, Chennai, Tamilnadu, India

Abstract: - The current study of fuselage segment describes about how to enhance the lift of the aircraft.Simultaneously, how to reduce the pressure drag during takeoff phase by providing ducts with bypasspassage. The duct extends inside the fuselage segment from nose cone to trailing edge of the fuselage section. It reduces the pressure drag at the stagnation point of the nose cone by bypassing the air stream and causing high pressure difference on the fuselage to obtain enhanced lift. The study reveals the importance of enhanced lift and pressure drag reduction for heavy cargo aircrafts during takeoff which will reduce the takeoff distance. The computational evaluation has been done along with grid independence study using Gambit 2.3.16 and Fluent 14.5 and experimental model is tested in low subsonic wind tunnel condition. The comparison results obtained is compatible.

Keywords: Enhanced Lift, Pressure Drag, Internal ducts, Nose cone, fuselage, CFD, Experimental.

INTRODUCTION

The fuselage is the major portion of the aircraft which connects the nose cone and the empennage section. Apart from that the wings of the aircraft also originate from the fuselage, so technically the fuselage covers the huge area which will result in increment of pressure drag, because pressure drag is directly proportional to the frontal area. For the past two decades drag due to fuselage has been a major technical problem for the aircraft design engineers in all kinds of aircrafts especially in military cargo aircrafts and passenger aircrafts. Later on the aerodynamic design improvement decreased the complexity in fuselage construction but the problem is still in existence. With the help of modern technology many of the research people of various countries implemented their ideas to enhance the lift and to reduce the pressure drag by applying computational fluid dynamics (CFD) principles and solving it using solvers like Fluent, CFX, Autodesk simulation software etc., to obtain approximate and acceptable results.

The purpose of this study is carried out to counterpart the pressure drag and to enhance the lift of the aircraft by implementing our concept of providing ducts inside the fuselage segment from nose cone to trailing edge which will be very much useful during takeoff for heavy aircrafts to reduce their runway distance. Not only for heavy aircrafts this concept can be applied to any kind of aircraft but most preferable one iscargo aircraft. The internal duct with bypass passage is extended throughout the fuselage whichbypasses the airstream at the stagnation point so that pressure drag will automatically get reduced. And the bypass passage that originated from the duct used to cause high pressure difference that would enhance the lift of the aircraft. The main advantage of this concept is to overcome the numerical instability of the normal fuselage segment. And the bypassing airstream can also be used for engine cooling by redirecting it over the engine section through additional ducts and the same airstream can be pre-heated artificially to prevent the aircraft from icing problems at different whether conditions. This will increase the reliability of the aircraft and as well as it will improve the performance. Other than this advantage the aircraft fuel consumption can also be reduced due to minimized pressure drag which makes our concept even more effective. And also the same airstream can used to produce thrust in the engine by placing the engine normal to the duct all of these ideas comes under our future works on ducted fuselage segment. This conceptual idea is implemented computationally to predict the results then the experimental work is carried out using wind tunnel to compare with the Cl, Cdcurve values obtain from computational results (CFD ANALYSIS).

By considering Boeing 737 as reference model our owngeometry of the fuselage is designed in CATIA V5R20 on both 2-D and 3-D. The 2-D model is meshed using Gambit 2.3.16 tool in unstructured scheme of meshing (TRI –MESH) with a skew of 0.5 the solved using Ansys

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 7, July - 2014

Fluent 14.5 solver. And all the post processing is done in the same solving tool.

Our Experimental model is tested in low speed subsonic wind tunnel condition and all readings are observed and the model is photographed using digital camera.

NOMENCLATURE:

V_w -velocity at the wall (m/s)

Cl -coefficient of lift

Cd -coefficient of drag

Re -Reynolds number

SWF -Standard wall function

K -Turbulent kinetic energy

Ɛ -Turbulent dissipation rate

GOVERNING EQUATIONS:

The current study is carried out based on the following assumptions

Steady condition

A system in a steady state has numerous properties are unchanging in time. This means that for those properties p of the system, the partial derivative with respect to time is zero.

$$\frac{\partial p}{\partial t} = 0$$

2-Dimensional

Bi-dimensional space is a geometric model of the planar projection of physical universe. The two dimensions are commonly called length and width. Both directions lie in the same plane.

Incompressible flow

The isochoric flow (incompressible flow) refers to a flow in which the material density is constant within a fluid parcel-an infinitesimal volume that moves with the velocity of the fluid. An equivalent statement that implies incompressibility is that the divergence of the fluid velocity is zero.

$$\nabla \cdot \mathbf{u} = 0.$$

Inviscid flow

An inviscid flow is the flow of an ideal fluid that is assumed to have no viscosity.

The Euler equation governing inviscid flow is:

$$\rho \left(\frac{\partial}{\partial t} + \mathbf{u} \cdot \nabla \right) \mathbf{u} + \nabla p = 0$$

CONTINUITY EQUATION

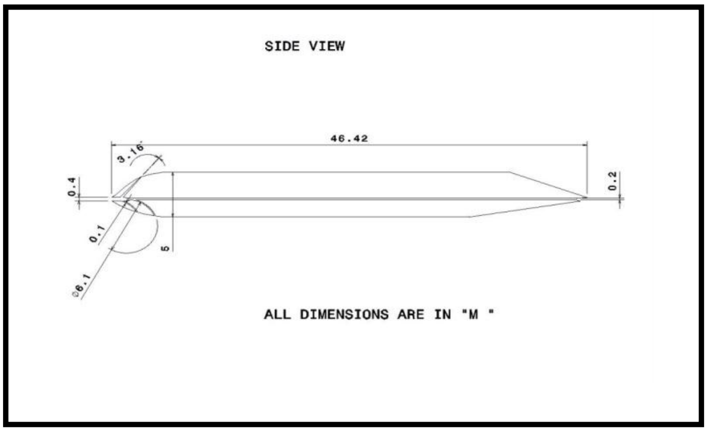
$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$

NAVIER STOKES EQUATION

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) = X - \frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)$$

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}\right) = Y - \frac{\partial P}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right)$$

GEOMETRY SETUP:



The Geometry of the fuselage sectionis designed in 2-Dimensional using CATIA V5R20 considering Boeing 737 as reference model and the dimension of our fuselage section is listed below

DIMENSIONS:

- Length of the fuselage = 46.42 m
- Height of the fuselage = 5 m

- Initial duct diameter = 0.4 m
- Constant duct diameter = 0.2 m
- Bypass tube diameter = 0.1 m

DOMAIN:

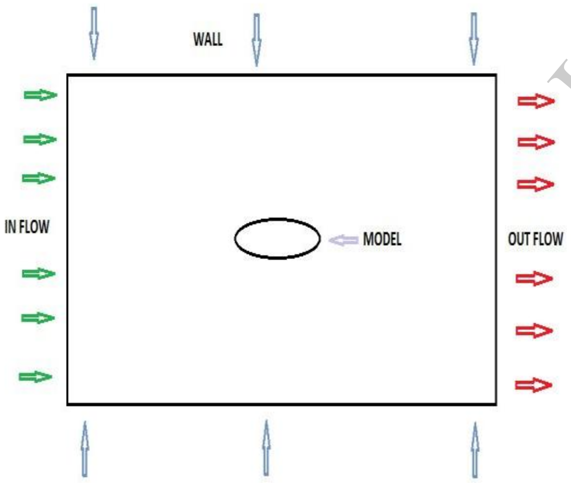
The domain selection and domain size place a vital role in convergence criteria to obtain acceptable results. Thedomain size must be 10:1 ratio of the model length and height. Choosing a proper domain will reduce the complexity of meshing the entire domain. Therefore, Rectangular domain is chosen and the domain size is arbitrarily taken as 10 times the model size.

BOUNDARY CONDITIONS:

The Dirichlet's boundary condition is adopted for our model which means that the boundary condition applied is fixed boundary conditions.

The boundary conditions assigned for the present study are

1. No slip boundary condition (V_w=0)
2. Velocity inlet
3. Wall
4. Outlet vent

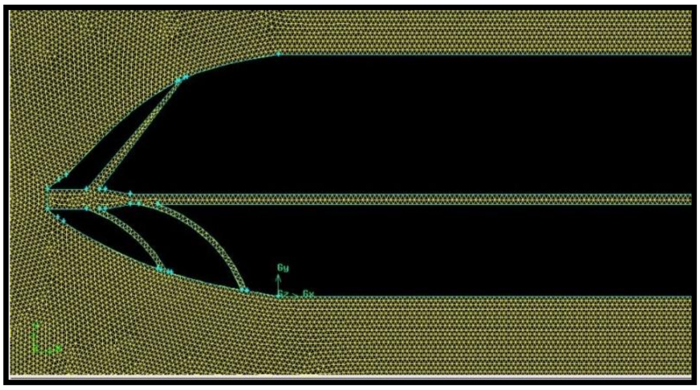


BOUNDARY CONDITIONS

MESH DETAILS:

The computational analysis of any model is highly depends on the mesh. The quality of mesh is directly proportional to the accuracy of the solution. Technically the number nodes existence in the mesh decides the level of accuracy for determining the node importance and accurate results grid independence study is preferred.

In the present study the model and the entire domain is meshed using unstructured scheme(TRI –pave) methodology is adopted. Using size function, face mesh tools the domain meshed with an interval size varying from 4-7 for normal and ducted fuselage segment. The skew of the meshed domain is checked and controlled by varying size function to minimize it below 0.5. The normal fuselage mesh, ducted fuselage mesh and the grid independence study method is also carried out using the same scheme.



DUCTED FUSELAGE MESH

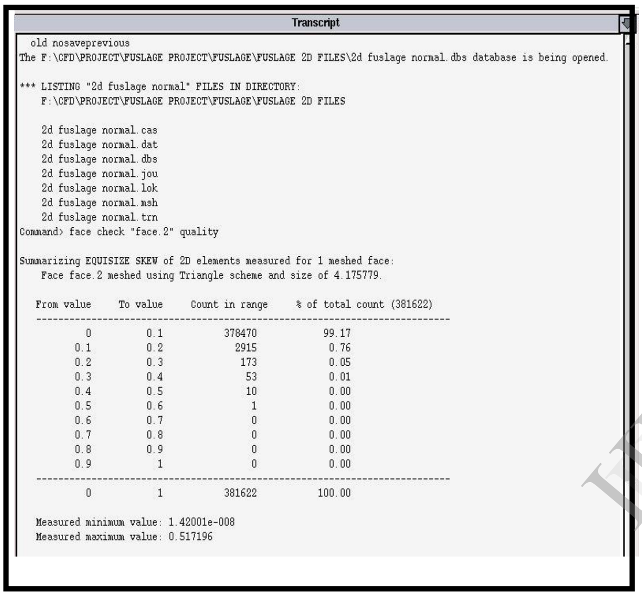
Examining Mesh Quality and Grid Independence Study:

The mesh quality depends upon the blocking in case of 3-D and size function or edge mesh in case 2-D. Therefore mesh quality is directly proportional to effective usage of size function or edge mesh to mesh the model. In order to verify or to crosscheck our self we need to examine the mesh for skew values and to visualize the worst element in the mesh. This feature in Gambit 2.3.16 allows us to examine our mesh and increasing our mesh quality. Normally the skew value ranges from 0 to 1. The quality of the mesh is high when the skew value is nearer to zero.

To identify the proper mesh size we need to perform the grid independence study means that we have to vary the node intensity to obtain best results. For our convenience we chosen 10 m/s as our constant velocity inlet value and we varied our number of nodes

GRID INDEPENDENCE TABLE

No.of nodes	57403	64509	75486	85295	102403	107219
Cd value	1.24	1.25	1.2	1.17	1.15	1.17
Skew	0.51	0.53	0.56	0.54	0.53	0.56
Mesh interval size	4	3.8	3.6	3.4	3.2	3
Reynolds No.	568	568	568	568	568	568



```
Transcript
  old nosaveprevious
The F:\CFD\PROJECT\FUSLAGE PROJECT\FUSLAGE\FUSLAGE 2D FILES\2d fuslage normal.dbs database is being opened.

*** LISTING "2d fuslage normal" FILES IN DIRECTORY:
    F:\CFD\PROJECT\FUSLAGE PROJECT\FUSLAGE\FUSLAGE 2D FILES

    2d fuslage normal.cas
    2d fuslage normal.dat
    2d fuslage normal.dbs
    2d fuslage normal.jou
    2d fuslage normal.lok
    2d fuslage normal.msh
    2d fuslage normal.trn
Command> face check "face.2" quality

Summarizing EQUISIZE SKEW of 2D elements measured for 1 meshed face:
    Face face.2 meshed using Triangle scheme and size of 4.175779.

   From value    To value    Count in range    % of total count (381622)
   ---------------------------------------------------------------------
          0         0.1          378470            99.17
        0.1         0.2            2915             0.76
        0.2         0.3             173             0.05
        0.3         0.4              53             0.01
        0.4         0.5              10             0.00
        0.5         0.6               1             0.00
        0.6         0.7               0             0.00
        0.7         0.8               0             0.00
        0.8         0.9               0             0.00
        0.9           1               0             0.00
   ---------------------------------------------------------------------
          0           1          381622           100.00

   Measured minimum value: 1.42001e-008
   Measured maximum value: 0.517196
```

Gambit 2.3.16 Skew check results

TURBULENCE MODEL SELECTION:

The turbulence model used for our study purpose is **k-ε** with **SWF**

$$\frac{\partial(\rho k)}{\partial t}+\nabla.(\rho \boldsymbol{U} k)=\nabla.\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\nabla k\right]+P_k-\rho\varepsilon$$

$$\frac{\partial(\rho \varepsilon)}{\partial t}+\nabla.(\rho \boldsymbol{U} \varepsilon)=\nabla.\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\nabla \varepsilon\right]+\frac{\varepsilon}{k}(C_{\varepsilon 1}P_k-C_{\varepsilon 2}\rho\varepsilon)$$

Where

$$k=\frac{1}{2}\left(\overline{u'^2}+\overline{v'^2}+\overline{w'^2}\right)$$

$$\varepsilon=2\nu\overline{e'_{ij}e'_{ij}}\,; \quad e'_{ij}=\frac{\partial u'_i}{\partial x_j}+\frac{\partial u'_j}{\partial x_i}$$

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon}$$

$$P_k=\mu_t\nabla \boldsymbol{U}.(\nabla \boldsymbol{U}+\nabla \boldsymbol{U}^T)-\frac{2}{3}\nabla.\boldsymbol{U}(3\mu_t\nabla.\boldsymbol{U}+\rho k)$$

The (empirical) constants in the k-ε model are usually:

$$C_\mu=0.09, \sigma_k=1.0; \sigma_\varepsilon=1.30$$
$$C_{1\varepsilon}=1.44, C_{2\varepsilon}=1.92$$

$$\boxed{k=1.5(IU_\infty)^2}$$

where I is the turbulence intensity $=\dfrac{\left(\overline{u'^2}\right)^{1/2}}{U_\infty}$

$$\boxed{\varepsilon=\frac{k^{3/2}}{\delta_L}} \quad \text{(for free shear flow)}$$

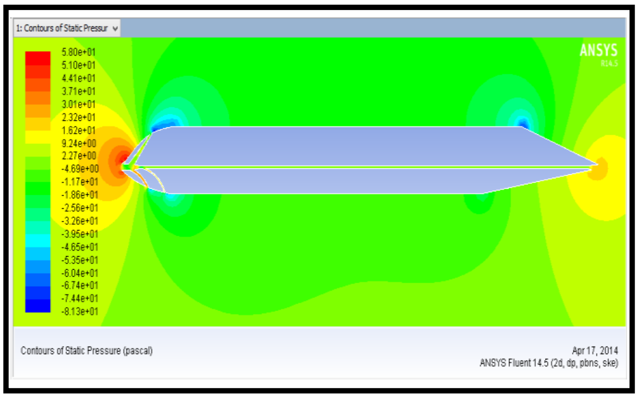
where δ_L is characteristic eddy length $\approx 0.1\delta$ and δ is the characteristic shear layer width

COMPUTATIONAL METHOD:

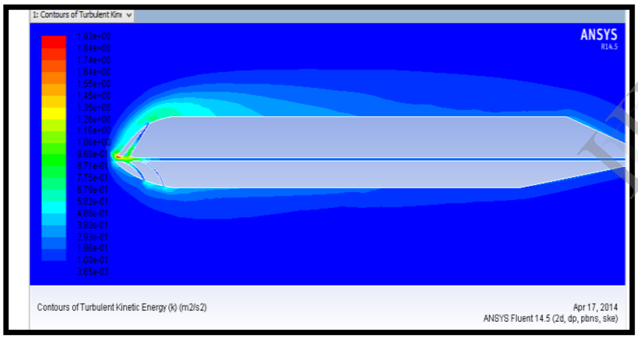
The computational method is carried out using a finite volume based solver Ansys Fluent 14.5 for solving the governing continuity and Navier Stokes equations. SIMPLE ALGORITHM used for solving our problem is pressure velocity coupling. The results is taken for three different velocity 10 m/s, 50m/s, 80m/s by varying the inlet boundary condition. The solution is considered to be converged when the residual is in the order of $10^{-4}$for continuity and navier stokes. The model is been iterated with total of 1000 iteration and the solution is converged in-between 600th- 650th iteration. The k-Ɛwith SWF turbulence model is used to solve the model.

The overall solving process is monitored properly and the results is obtainedand verified from the CFD post processor and the comparison of plots is done using Microsoft offices excel –graphs. The pressure counters, velocity counter and Cl, Cd values are obtained from the solver for each velocities and the results are plotted in excel sheet to obtain the lift enhancement curve, pressure drag reduction curve. Even though the computational results are compatible the experimental results are carried out to prove our concept and the results obtained are compared with computational results for verification.

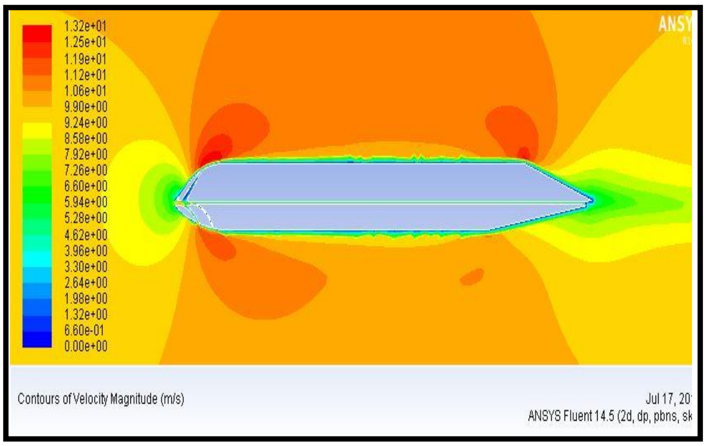
FLUENT RESULTS



Pressure contour



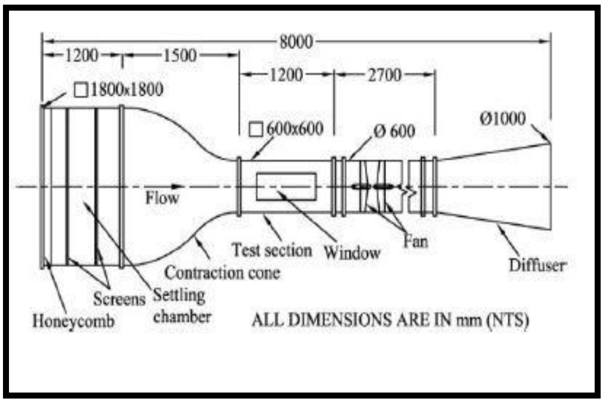
Turbulence contour for ducted model



Velocity contour

EXPERIMENTAL RESULTS:

The experimental results are obtained by testing the scaled model in low subsonic wind tunnel at KCG College of technology, Chennai. The experimental model is made with probes along the longitudinal direction as well as circumferential direction to take exact readings at the inner ducts and bypass passage. The model is tested for five different rpm to compare the results with computational results. The experimental readings are evaluated using theoretical formulae to get Cl, Cdvalues for our model.



WIND TUNNEL DIAGRAM

The experimental model specifications:

Length of the fuselage= 0.4642m

Height of the fuselage=0.05m

Initial duct diameter= 0.0004m

Constant duct diameter= 0.0002m

Bypass tube diameter= 0.0001m

The experimental calculations are plot in excel sheet to plot the Cl, Cd values of normal fuselage and ducted fuselage. Later on the results are compared with computational results to prove our conceptual idea.

ISSN: 2278-0181
Vol. 3 Issue 7, July - 2014

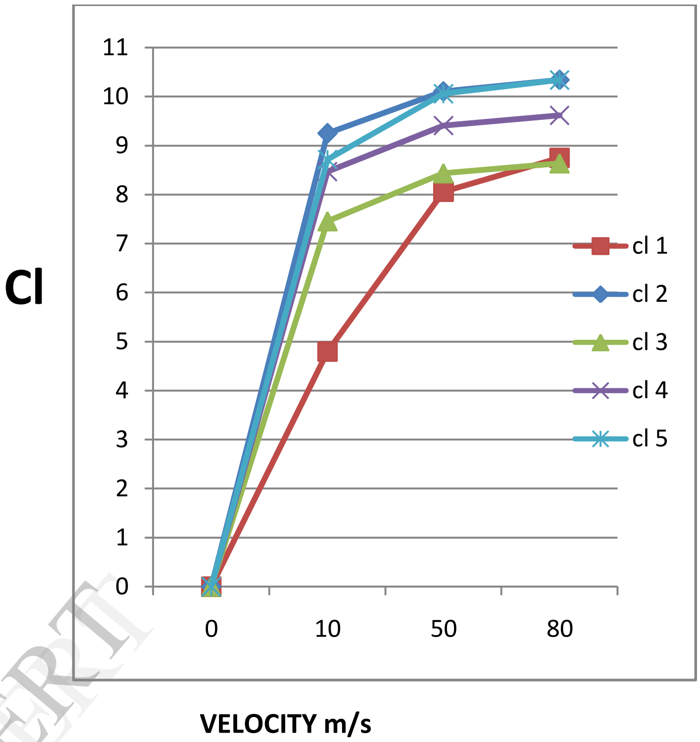
Experimental Model Inside The Wind Tunnel

RESULT COMPARISON:

From the result obtained and excel plots the results of the experimental and the computational values approximately the same. Hence our conceptual idea of enhancement of lift is proved experimentally and computationally. Therefore the result obtained from this study founded a solution for major problem in the aircraft. This solution can be implemented on any kind of aircraft to improve the performance and to enhance the lift and reduce the pressure drag which will provide more reliability for the aircraft at any whether condition, So that aircraft design and modern technology will enters into the new revolution.

Plotcomparison

Cl curve:



Where:

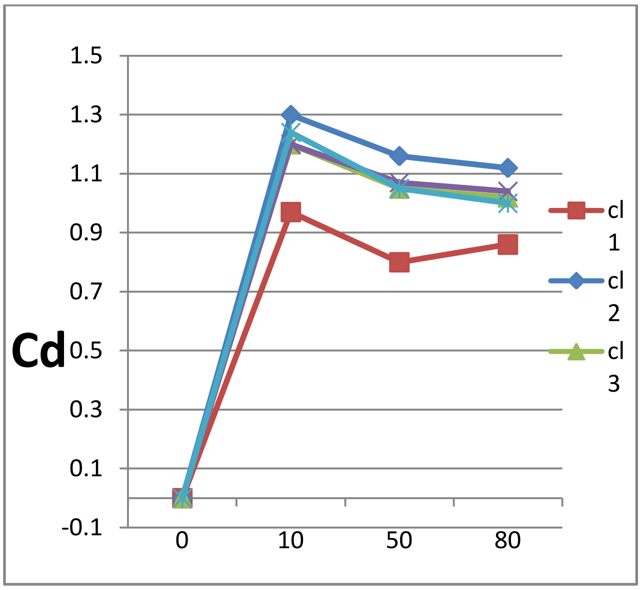
cl 1- Cl of normal fuselage

cl 5- Cl of ducted fuselage

cl 2, cl 3, cl 4- Cl of Duct position varied fuselage segment (trial and error methodology)

The CL vs. Velocity curve indicates that the ducted model (cl 5) attains maximum Lift than the normal model (cl 1)

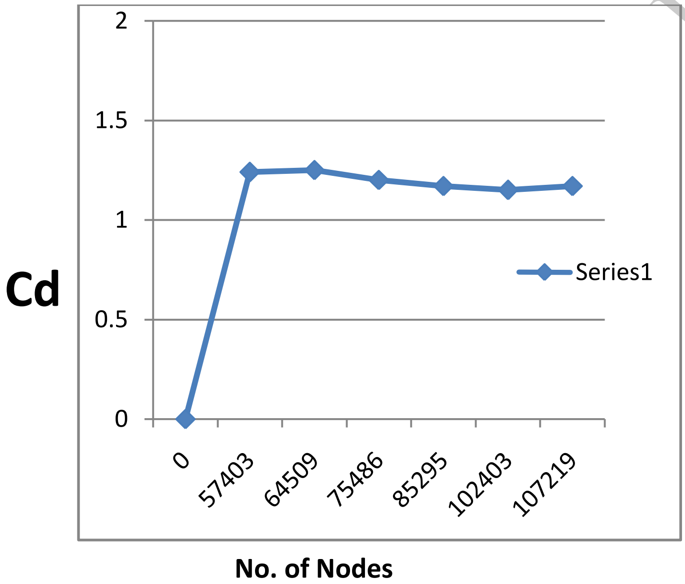
Cd curve:

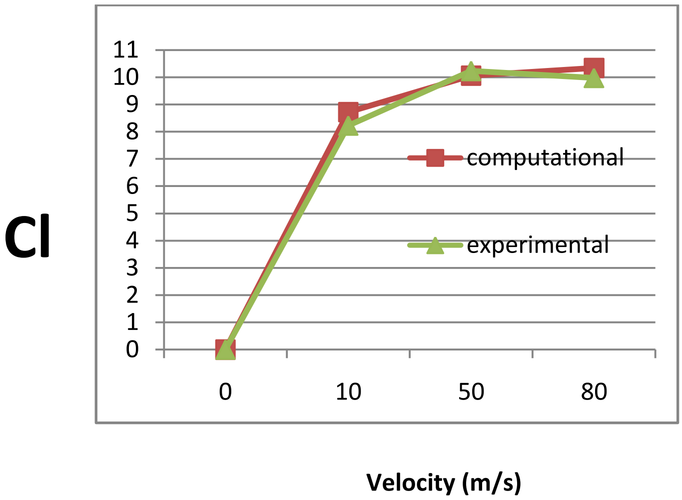


Where: **VELOCITY m/s**

cl 1- Cd of normal fuselage
cl 5- Cd ducted fuselage
cl 2, cl 3, cl 4- Cd of Duct position varied fuselage segment (trial and error methodology)
From the Cd vs. velocity curve we can understand that cd for the ducted model (cl5) reduces with increase in velocity whereas Cd of the normal model (cl 1) keep on increasing with increase in velocity

GRID INDEPENDENCE STUDY CURVE





EXPERIMENTAL vs. COMPUTATIONAL

CONCLUSION:

We conclude that our research study found out a new solution for major problems that normally occurs in the fuselage. Apart from that this concept can improve performance of defense aircrafts and heavy aircrafts. This concept will be more useful for the next generation of aircraft for anti-icing and de-icing by pre heating the airstreamand it can also be used to cool the engine etc., all these studies are our future works.

FUTURE WORKS

- To increase the performance of the aircraft by using this ducted concept as a direct inlet stream to the engine
- To improve reliability of the aircraft by redirecting the ducted stream after pre –heating it to avoid icing on the aircraft engine
- To improve engine performance by cooling the nacelle by the airstream from the duct

REFERENCES:

1. An investigation of air tactical cargo aircraft aft body drag reduction based on cfd analysis and wind tunnel tests by Masoud Mirzaeia,Mohammad Hadi Karimia,Mohammad Ali Vazirib
2. Turbulent drag reduction by means of alternating suction and blowing jets by Takehiko Segawaa, Hiroshi Mizunumab, Koki Murakamib, 1, Feng-Chen Lia, 2, Hiro Yoshidaa
3. Recent experience with different methods of drag prediction by C.P. van Dam,
4. Aircraft viscous drag reduction using riblets by P.R Viswanath,
5. Flow management techniques for base and after body drag reduction by P.R. Viswanath
6. Three-dimensional aspects of cylinder drag reduction by suction and oscillatory blowing by Tom Shtendel, Avi Seifert,