Drag Prediction on NASA's Common Research Model Using CFD Tool

Arvindh R.
Aravindh S.
Anwar H.
Infant Dean Jones D.
B.E. Aeronautical Engineering
Adhiyamaan College of Engineering,
Hosur, India
raravinth@hotmail.com
aravinds1293@hotmail.com

Vijaya Kumar R.
Asst. professor
Department of Aeronautical Engineering
Adhiyamaan College of Engineering,
Hosur, India
vijaymtechnal@gmail.com

Abstract

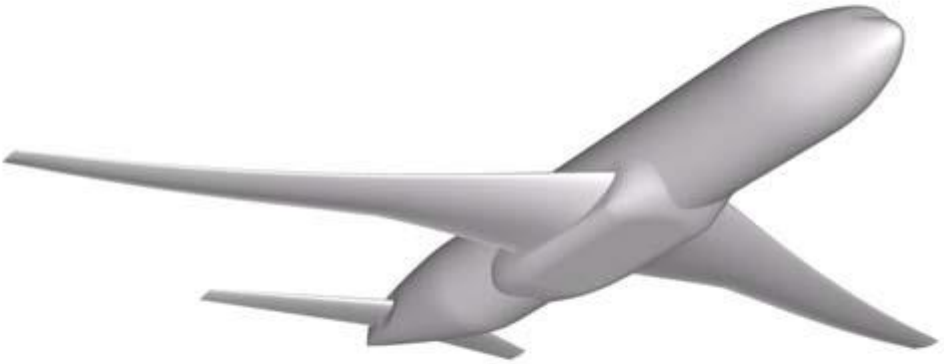
The Common Research Model utilized by the Drag Prediction Workshop is employed to investigate the accuracy of CFD codes in transonic aerodynamic flow analysis. The codes used in this study is ANSYS FLUENT (unstructured). Solutions are to be obtained for the wing-body configuration with the horizontal tail at incidence angles of 0. Predictions of cruise drag and pitching stability are carried out. The drag polar and pitching stability graphs are obtained from the CFD results

Keywords: *common research model, drag polar, pitching stability, CFD.*

1. INTRODUCTION:

Effective use of computational fluid dynamics (CFD) is a key ingredient in successful design of modern commercial and military aircraft. The combined pressures of market competitiveness, dedication to the highest of safety standards, and desires to remain a profitable business enterprise all contribute to make intelligent, extensive, and careful use of CFD a major strategy for product development. The application of CFD today has revolutionized the process of aerodynamic design. CFD has joined the wind tunnel and flight test as primary tools of the trade.[1-4] Because of the tremendous cost involved (and potential risk) in flight testing, modern aircraft development places heavy focus on the use of CFD and the wind tunnel prior to flight. Particularly, CFD is used to provide understanding and insight as to the source of undesirable flight characteristics, whether they are observed in sub-scale model testing or in the full-scale flight testing. Collectively, flight testing, wind tunnel testing, and CFD all contribute to minimizing risk and uncertainty in a new airplane product. Therefore, validation and improvement of CFD technology is a requirement and an ever-present necessity for aerospace companies.

The main focus of this research was cruise drag prediction for the NASA Common Research Model (CRM). The CRM is representative of a commercial wing-body-tail transport configuration and is shown. It was jointly designed by NASA and Boeing, the overall design, fabrication, and testing was led by NASA and the detailed aerodynamic design was led by the Boeing Company[6]. The configuration was designed for a cruise Mach number of 0.85 at a fixed lift coefficient of 0.5 and a Reynolds number of 40 million per mean aerodynamic chord. The wing has an aspect ratio, AR, of 9.0, a mean aerodynamic chord, MAC, of 7.005 meter, and a taper ratio of 0.275. A trailing edge thickness of 0.0003556 meter for the 2.7% scale wind-tunnel model was used to accommodate fabrication limits. The Yehudi break is set at 37% span.

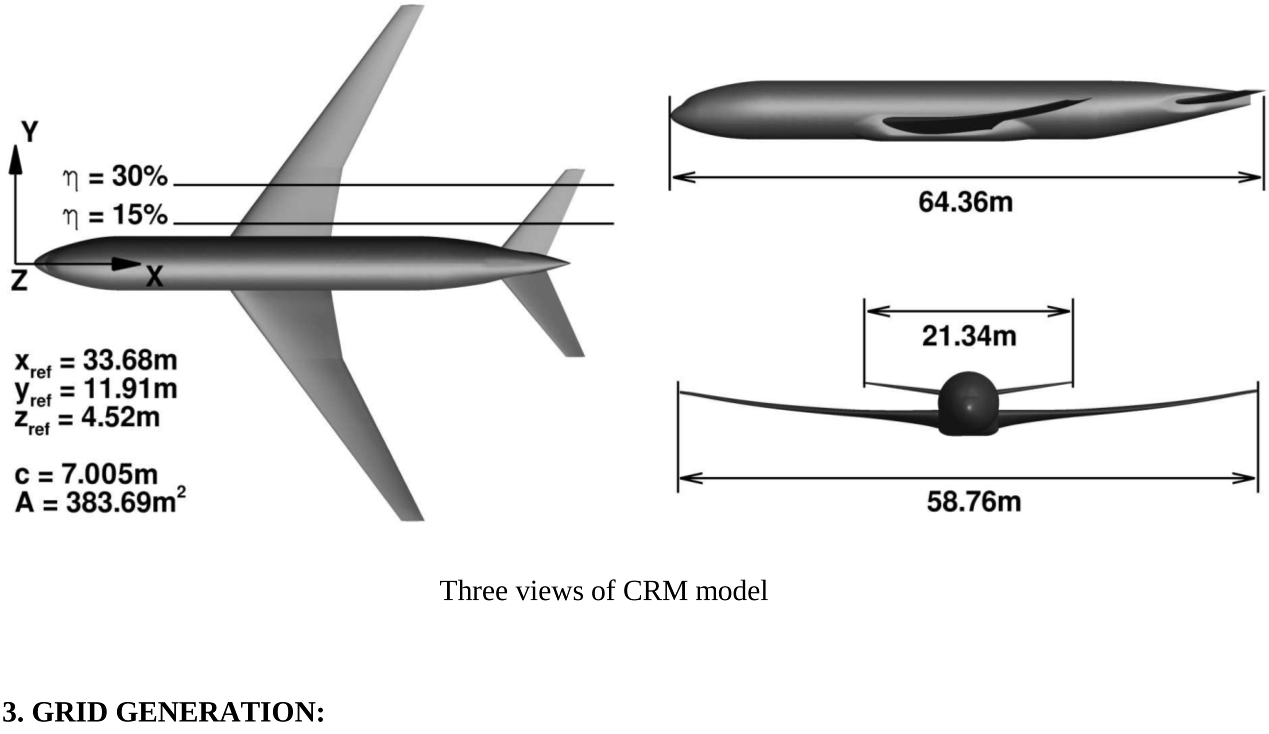


Common Research Model wing body horizontal tail geometry.

2. CRM MODEL GEOMETRY SPECIFICATIONS

The NASA Common Research Model (CRM) [6] consists of a contemporary supercritical transonic wing and a fuselage that is representative of a wide body commercial transport aircraft. The CRM is designed for a cruise Mach number of M= 0.85 and a corresponding design lift coefficient of C_L= 0.5

The aspect ratio is 9.0, the leading edge sweep angle is 35 deg, the wing reference area (S) is 0.2796 m^2, the wing span (b) is 1.586 meter, and the mean aerodynamic chord (c) is 0.189 meter. The model moment reference center is located 0.90932 meter back from the fuselage nose and 0.0298 meter below the fuselage centerline.



Three views of CRM model

3. GRID GENERATION:

ICEM-CFD grid generator was used to develop the mesh for ANSYS FLUENT. Accurate representations of the geometry, along with generating a high quality mesh, are critical first steps in obtaining an acceptable solution. Three different meshes (coarse, medium, fine) for solver were generated based on the DPW4 requirements. The mesh details associated with each solver is shown below.

Grid type	Coarse	Fine	Extra fine
Elements	3563261	7091456	25194224
Nodes	621016	1240573	4438511
Surface mesh	All Trias	All Trias	All Trias
Volume mesh	All Tetrahedrons	All Tetrahedrons	All Tetrahedrons
Mesh type	Patch dependent	Patch dependent	Patch dependent

Description of the meshes for ANSYS FLUENT

4. ICEM-CFD Unstructured Mesh Generation:

The IGES file was imported in ICEM-CFD with tri tolerance of 0.0001 and topo tolerance of 0.001. Geometry clean up carried out in ICEM with build topology tolerance 0.3. Mesh was carried out with patch dependent method and the surface are meshed with only trias elements. First fine mesh was generated then with those parameters in fine mesh as base coarse mesh and medium grids were generated. The global element size of three grids are given in table 2. The planform of surface mesh over the common research model are shown in figure

GRID	**GLOBAL ELEMENT SIZE**
Fine grids	1024
Medium grids	2048
Coarse grids	4096

Global element size for different grids

The surface mesh was carried out with only trias elements on the surface with patch dependent as mesh method. The volume mesh was initialised with tetra/mixed mesh type and mesh method as robust (octree) [8]. The volume mesh are generated with tetrahedral elements

The guidelines set forth by DPW for the grid-generation requirements were closely followed with the following exception. The coarse and medium grids were assessed to provide insufficient resolution on the leading edge of wing and horizontal tail. To remedy this, the leading edge spacing on the coarse and medium grids was set to be similar to the fine grid. This gives rise to the nomenclature coarse-fine and medium-fine grid used for the FLUENT work.

5. RUN CONDITIONS:

Case 1 – NASA Common Research Model (CRM) Wing-Body Common Grid Study:

1. Mach = 0.85, CL = 0.500 (±0.001)
2. Drag Polar for alpha = 2^0

Grid refinement series from the Common Grid Sequence consisting of three grid levels with

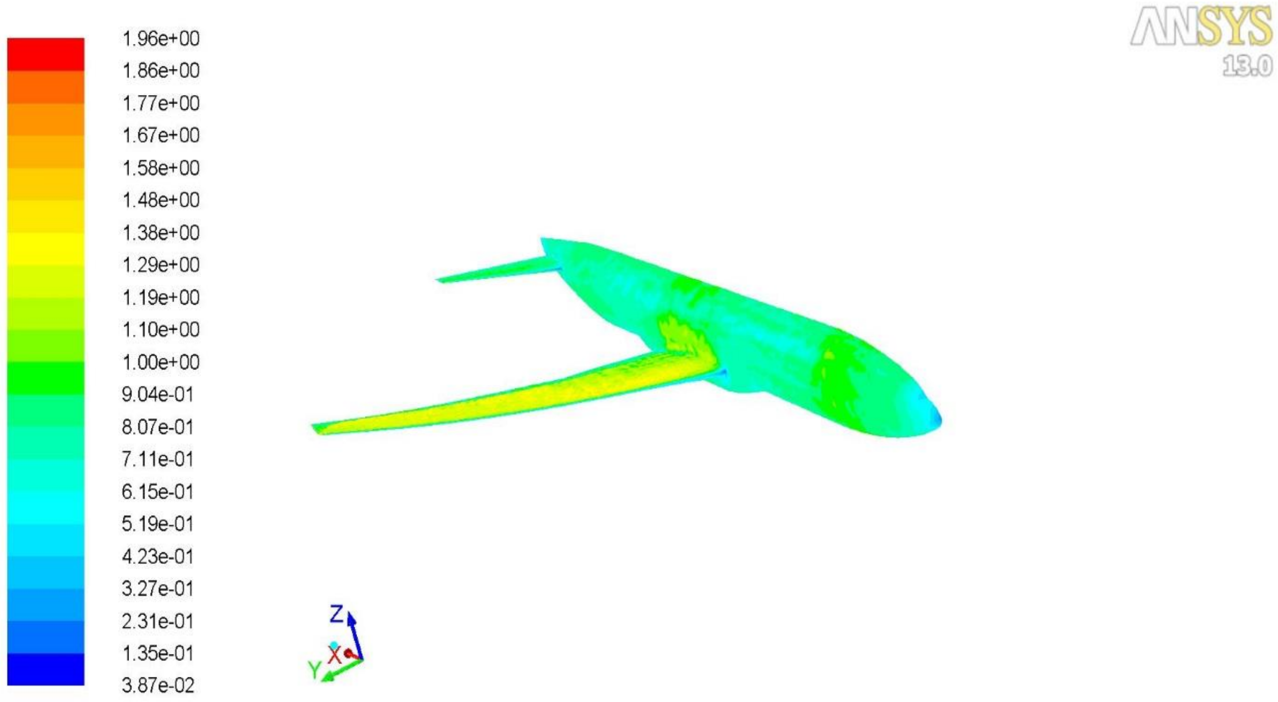
- Chord Reynolds Number RE = $5x10^6$ based on C_{REF} = 7.005 m
- Reference Temperature = 100^0 f
- Moment reference center is X_{REF} = 33.68 m, Z_{REF} = 4.52 m
- Chord Reynolds Number Rn = $5x10^6$.
- Reference pressure = 201326.9 Pa

6. PRELIMINARY SETTINGS IN SOLVER:

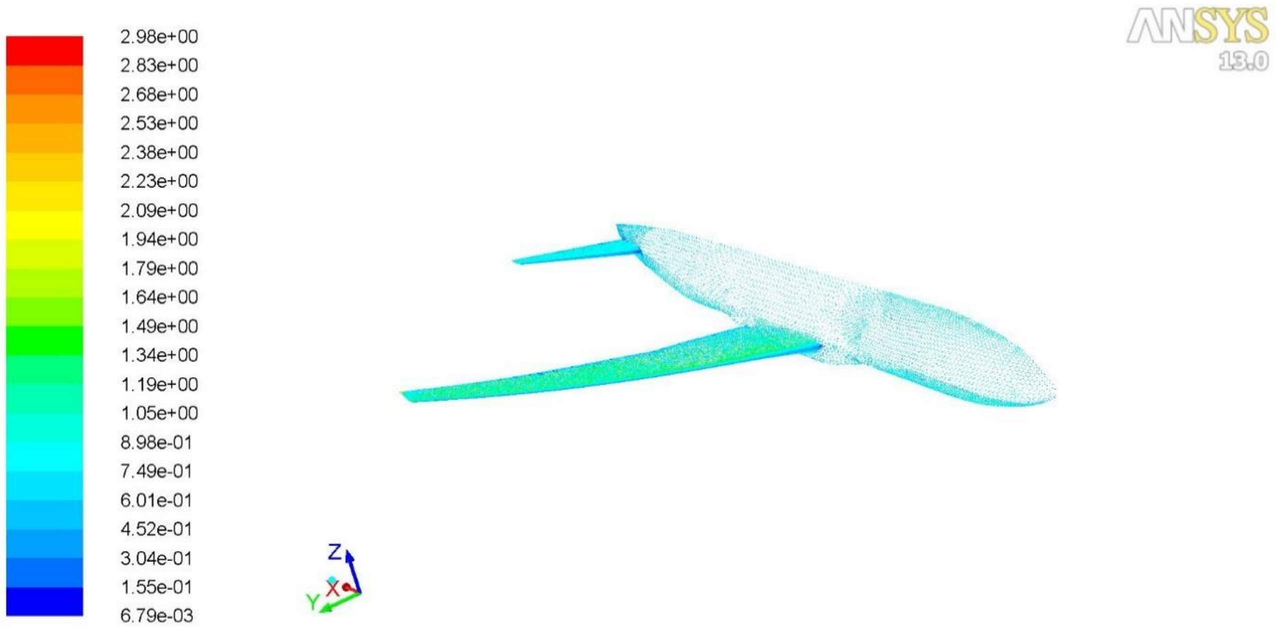
- Solver used : ANSYS FLUENT 14.0
- Solver type : Density-Based
- Model : Turbulent
- Solving Equation : Sparlart Almaras
- Fluid used : Air (as Ideal Gas)
- Solution methods
    - Formulation : Implicit
    - Flux type : AUSM
    - Gradient : Least squares cell based
    - Flow type : second order upwind

7. RESULTS AND DISCUSSION:

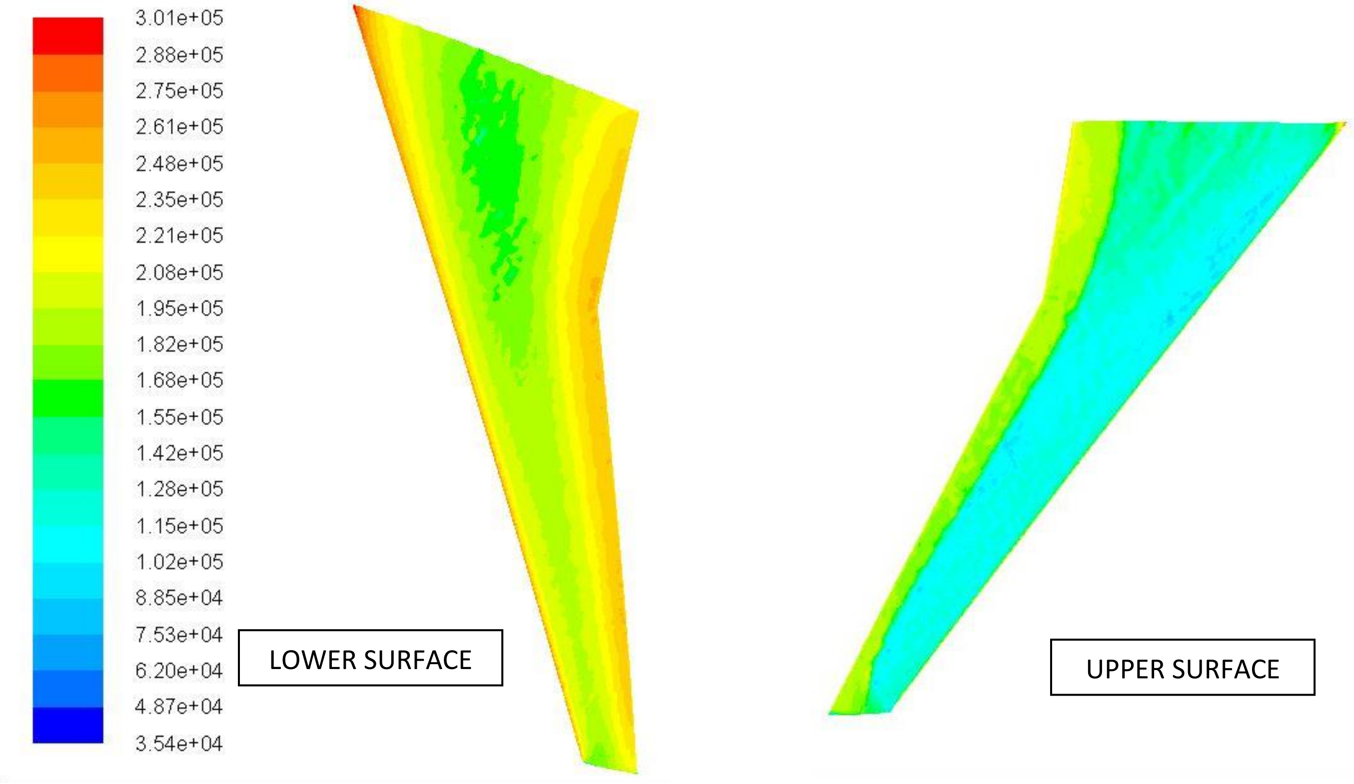
The iteration were stopped when the scaled monitor residual values showed alternate fluctuation of same bandwidth. The scaled residual, lift, drag, moment monitors are shown in figure. The results such as the pressure distribution, velocity vectors, mach number contours are shown in figure



Pressure contour of the CRM with MACH 0.85

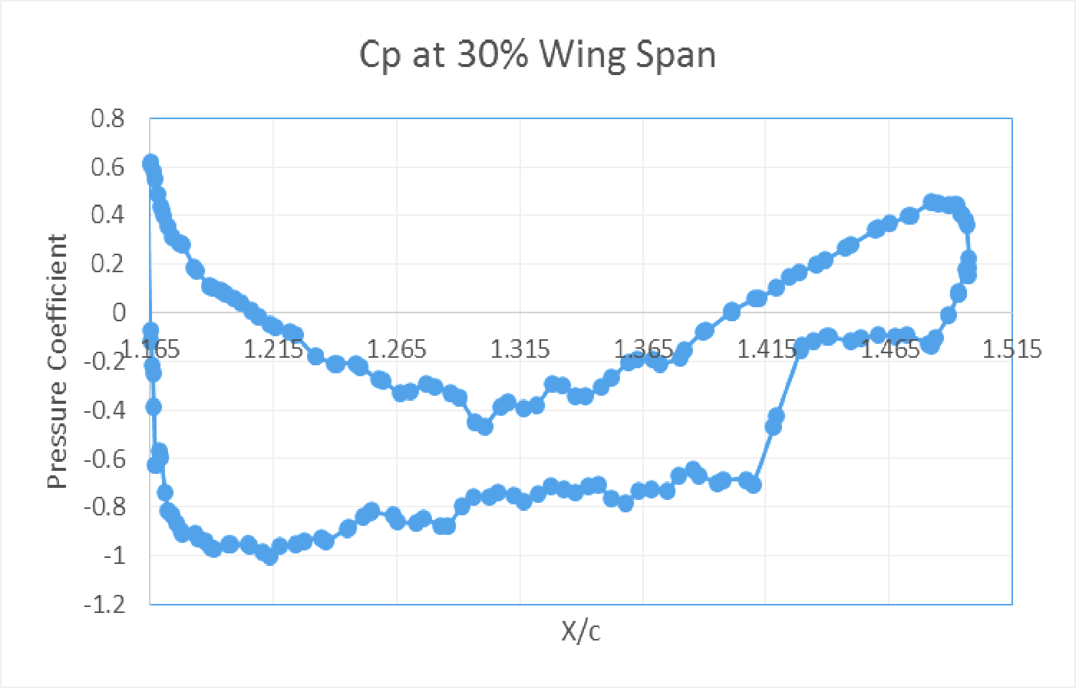


Velocity vectors of the CRM in MACH 0.85

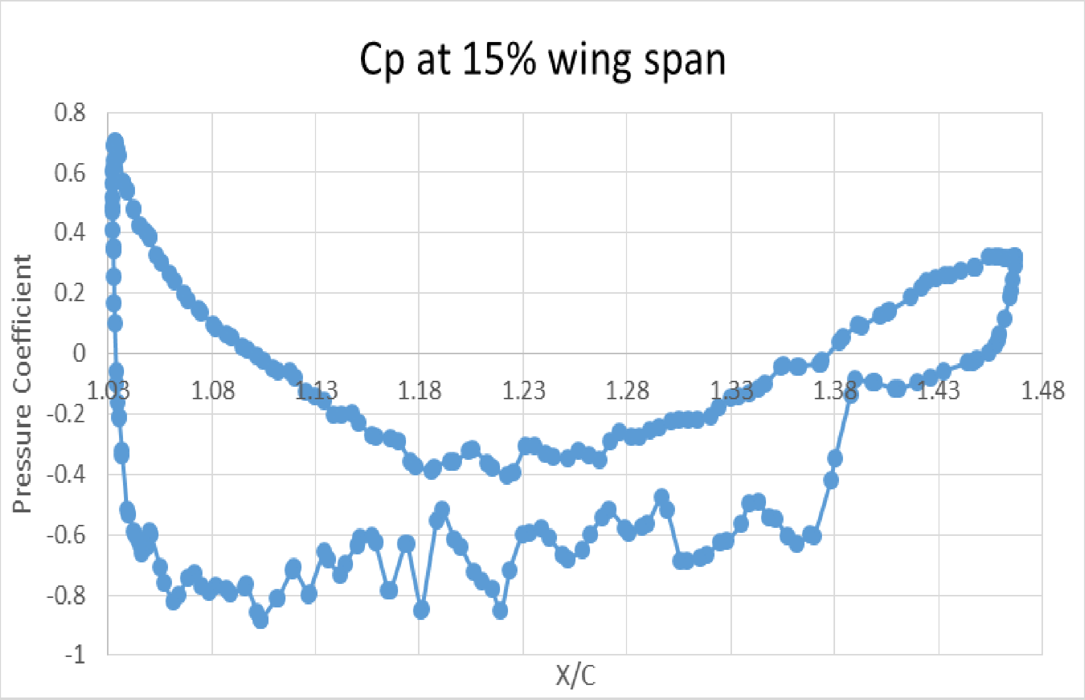


Pressure distribution over the wing of CRM

The Cp distribution over the wing at 15% and 30% of wing span are shown in figure for 2^0 angle of attack.

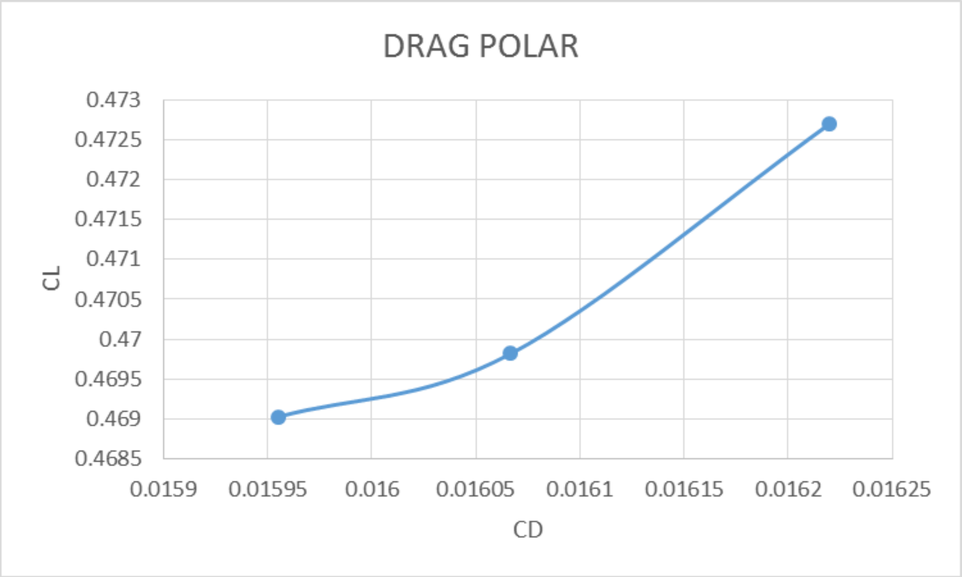


Cp distribution at 30% of wing span

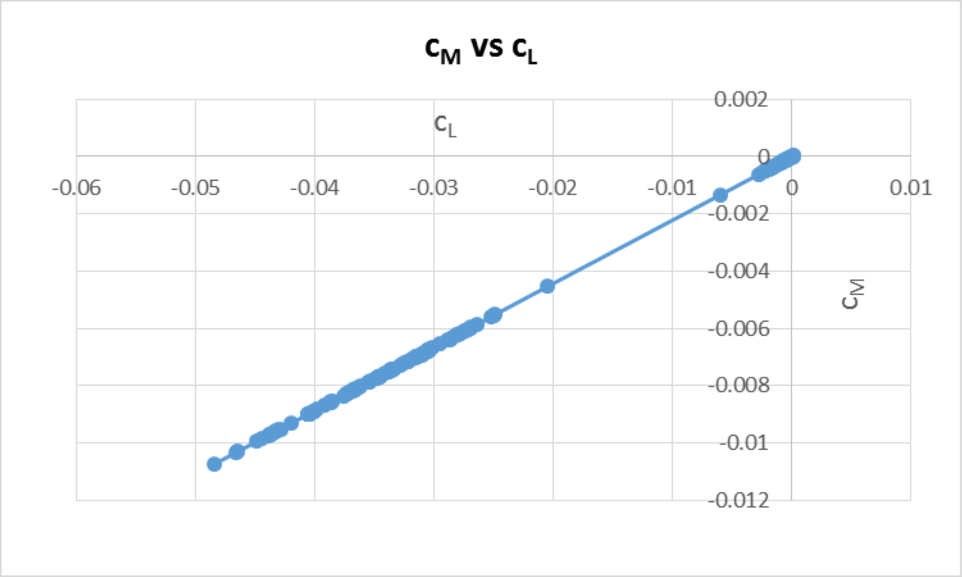


Cp distribution at 15% of wing span.

The drag polar curve is shown in figure. The drag polar curve is the graphical representation coefficient of lift vs coefficient of drag i.e. CL vs CD. This graph shows at given CL in which CD is minimum.



From the above graph the max CL is 0.00047 but the max CL at which CD is minimum is 0.00046.The pitching moment graph is shown it is the graph between c_M and c_L. The pitching moment graph is to show the pitching moment stability.



The negative slope for the positive angle of attack indicates stability in pitching.

8. ACKNOWLEDGEMENTS:

First and foremost, our thanks to our parents for their constant support in every aspect of our life and blessed us to successfully pursue our Bachelor of Engineering and to successfully accomplish our project work. We would like to express our heart full thanks to Dr. G. RANGANATH, M.E., Ph.D., Principal, Adhiyamaan college of Engineering, Hosur for his endorsement of this project work. We are thankful to our external guide Mr. VISHAL S. SHIRBHATE, scientist C, NAL, Bangalore and Mr. VIJAYA KUMAR R., Asst. Professor, Dept. of Aeronautical Engineering, ACE, Hosur for their valuable and continuous support in completing this project work. We are grateful to Dr. J. ARIVUDAINAMBI, M.E., Ph.D., Professor and HoD, Department of Aeronautical Engineering, for his valuable and continuous support in completing the project work.

9. REFERENCES

[1]. Rubbert, P. E., “AIAA Wright Brothers Lecture: CFD and the Changing World of Airplane Design” ICAS 94-0.2, September 1994.

[2]. Tinoco, E. N., “The Changing Role of Computational Fluid Dynamics in Aircraft Development,” AIAA Paper 98-2512, June 1998.

[3]. Johnson, F. T.,Tinoco, E. N., and Yu, N. J., “Thirty Years of Development and Application of CFD at Boeing Commercial Airplanes, Seattle,” AIAA Paper 2003-3439, June 2003.

[4]. Vassberg, J. C., Dehaan, M. A., Rivers, S. M., and Whals, R. A.”Development of a Common Research Model for AppliedCFD Validation Studies,” AIAA Paper 2008-6919, August 2008.

[5]. 4th AIAA CFD Drag Prediction Workshop,”http://aaac.larc.nasa.gov/tsab/cfdlarc/aiaa-dpw/.

[6]. “ICEM-CFD help engine” by ANSYS.