ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

CFD ANALYSIS OF BLENDED WING BODY AND B2 WING

K. Simhachalam Naidu*, G. Shruthi Keerthi, V.V.S. Nikhil Bharadwaj
Assistant Professor, Department of Aeronautical Engineering, MLR Institute of Technology, Hyderabad.
M.Tech Student, Department of Aeronautical Engineering, MLR Institute of Technology, Hyderabad.
B.Tech Student, Department of Aeronautical Engineering, MLR Institute of Technology, Hyderabad.

ABSTRACT
In recent years unconventional aircraft configurations, such as Blended-Wing-Body (BWB) aircraft and B2 wing, are being investigated and researched with the aim to develop more efficient aircraft configurations, in particular for very large transport aircraft that are more efficient and environmentally-friendly. These configurations designate an alternative aircraft configuration where the wing and fuselage are integrated which results essentially in a hybrid flying wing shape. The purpose of this research is to assess the aerodynamic efficiency of a blended wing body aircraft with respect to a B2 wing configuration, and to identify design issues that determine the effectiveness of the "Blended Wing and B2 wing" performances. The approach was undertaken to develop a new conceptual design of a BWB aircraft using Computational Aided Design (CAD) tools and Computational Fluid Dynamics (CFD) software. The research contains several geometry parameters which are varied to investigate the influence on the aerodynamic and stability characteristics of two configurations. In the last part, a special case has been explored in an attempt to improve the stability by changing geometry parameters.

KEYWORDS: BWB aircraft, CAD, CFD, B2 wing Design.

INTRODUCTION
AIM OF THE PROJECT: The aerodynamic characteristics by validating our results with the published papers. The objective of our project is to propose a BWB configuration with an optimized kink angle which would give better lift at subsonic speeds with lesser induced drag as compared to the other configurations. The research methodology contains the study of the geometry parameters which are varied to investigate the influence on the aerodynamic characteristics of the two configurations. The development is broken into three distinct phases: formulation, initial development, and feasibility. Symmetrical coordinates are considered to design and analyze the BWB & B2 configurations.

CONFIGURATIONS OF FLYING WING
Blended Wing Body (BWB) aircraft is a concept where fuselage is merged with wing and tail to become a single entity. BWB is a hybrid of flying-wing aircraft and the conventional aircraft where the body is designed to have a shape of an airfoil and carefully streamlined with the wing to have a desired plan form. If the wing in conventional aircraft is the main contributor to the generation of lift, the fuselage of BWB generates lift together with the wing thus increasing the effective lifting surface area. The streamlined shape between fuselage and wing intersections reduces interference drag, reduces wetted surface area that reduces friction drag while the slow evolution of fuselage-to-wing thickness by careful design may suggest that more volume can be stored inside the BWB aircraft, hence, increases payload and fuel capacity. The BWB concept aims at combining the advantages of a flying wing with the loading capabilities of a conventional airliner by creating a wide body in the Centre of the wing to allow space for passengers and cargo.

Fig: Blended Wing Body Aircraft Designed by NASA

Especially, for very large transport aircraft, the BWB concept is often claimed to be superior compared to conventional configurations in terms of higher lift-to-drag ratio and consequently less fuel consumption.

B2 Wing

It is a special a type of flying wing designed mainly for the military purposes. This B2 wing is used only in Northrop Grumman which is the prime contractor for the US Air Force B-2 Spirit stealth bomber. The B-2 is a low-observable, strategic, long range, heavy bomber capable of penetrating sophisticated and dense air-defense shields.

Fig: Northrop Grumman B2 Bomber Aircraft

It is capable of all-altitude attack missions up to 50,000ft, with a range of more than 6,000nm unrefueled and over 10,000nm with one refueling, giving it the ability to fly to any point in the world within hours. Its distinctive profile comes from the unique 'flying wing' construction. The leading edges of the wings are angled at 33° and the trailing edge has a double-W shape. It is manufactured at the Northrop Grumman facilities in Pico Rivera and Palmdale in California. The B2 after ten years of service finally achieved full operational capability in December 2003. In the first three years of service, the operational B2s achieved a sortie reliability rate of 90%. An assessment published by the USAF showed that two B2s armed with precision weaponry can do the job of 75 conventional aircraft.

DESIGN - ICEM CFD

INTRODUCTION

The Computational analysis of the Blended wing body-“Proposed Model” is carried out using the CFD techniques. In this project design and analysis of the wing is carried using ANSYS ICEM CFD and the Fluent-V6 software’s. The design of the model is carried out using the ANSYS-ICEM CFD software and then it is imported to the Fluent-V6 software for the analysis. ANSYS ICEM CFD is a popular proprietary software package used for CAD and mesh generation. Some open source software includes Open FOAM, Feat-Flow etc. Present discussion is applicable to

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

ANSYS ICEM CFD software. It can create structured, unstructured, multi-block, and hybrid grids with different cell geometries. ANSYS ICEM CFD is meant to mesh a geometry already created using other dedicated CAD packages. Therefore, the geometry modeling features are primarily meant to 'clean-up' an imported CAD model. Nevertheless, there are some very powerful geometry creation, editing and repair (manual and automated) tools available in ANSYS ICEM CFD which assist in arriving at the meshing stage quickly. Unlike the concept of volume in tools like GAMBIT, ICEM CFD rather treats a collection of surfaces which encompass a closed region as BODY. Therefore, the typical topological issues encountered in GAMBIT (e.g. face cannot be deleted since it is referenced by higher topology) don't show up here. The emphasis in ICEM CFD to create a mesh is to have a 'water-tight' geometry. It means if there is a source of water inside a region, the water should be contained and not leak out of the Body.

DESIGN PROCESS

The design process is basically divided into three phases as follows:

Pre-Processing:

This is the first step of CFD simulation process which helps in describing the geometry in the best possible manner. One needs to identify the fluid domain of interest. The domain of interest is then further divided into smaller segments known as mesh generation step. There are different popular Pre-Processing software available in the market including CFD-GEOM, ANSYS ICEM CFD etc.

Solver:

Once the problem physics has been identified, fluid material properties, flow physics model, and boundary conditions are set to solve using a computer. There are popular commercial software available for this including ANSYS FLUENT, ANSYS CFX etc. All these software have their unique capabilities. Using this software, it is possible to solve the governing equations related to flow physics problem.

Post-Processing: The next step after getting the results is to analyze the results with different methods like contour plots, vector plot, streamlines, data curve etc. for appropriate graphical representations and report. Some of the popular post-processing software includes ANSYS CFD-Post etc.

Steps for Designing in ICEM CFD

- Extract the coordinates from the draft sketch taken from reference paper.
- Open ICEM CFD software and import coordinates using: File> Import Geometry> Formatted Point Data.
- Now create splines using: Geometry> Create/Modify Curve> from points> select the points> Apply> Ok.
- Create a control volume using approximate coordinates and join the points.
- Further create surface: Geometry> Create/Modify surface> Simple surface> from curves> Select curves (first control volume and then body)> Apply> Ok.
- Now extrude the geometry: Geometry> Create/Modify surface> Sweep surface> Vector method> through points> Select 2 points with 4mm thickness between them> then select the whole geometry> Apply> Ok.
- Create parts by using right mouse click on Parts column in the left side, the parts to be created include:

a) INLET
b) OUTLET
c) LEADING EDGE
d) TRAILING EDGE
e) WALLS
f) SYMMETRY

- Now creating a body: Geometry> Create/Modify surface> Create Body> type part as LIVE >Material point> Select 2 opposite points> Apply> Ok.
- Now generate a Mesh: Mesh> Global Mesh setup> Global Mesh parameters> Global Mesh size> Enter the values.
- Mesh> Part Mesh setup> Enter Maximum size > Apply.

- Mesh> Compute Mesh> Volume Mesh> Tetra/Mixed mesh> Robust (octree) mesh method> Select entity > Compute.
- Creating output file to Fluent V6: Output> Output to FLUENT V6> Part boundary conditions> Accept> Select destination folder> Grid Dimension-3D> Default settings> Done> Output file created (".msh" file).

DESIGN OF BWB WITH 38.3^0 KINK ANGLE

- Extract the coordinates from the draft sketch taken from reference paper.
- Open ICEM CFD software and import coordinates using: File> Import Geometry> Formatted Point Data> Select the coordinates file> Ok.
- Now create splines using: Geometry> Create/Modify Curve> from points> select the points> Apply> Ok.
- Create a control volume using approximate coordinates and join the points.
- Further create surface: First Control volume: Geometry> Create/Modify surface> Simple surface> from curves> Select curves> Apply> Ok and similarly repeat the step for creating surface of body.
- Now extrude the geometry: Geometry> Create/Modify surface> Sweep surface> Vector method> through points> Select 2 points with 4mm thickness between them> then select the whole geometry> Apply> Ok.
- Create parts by using right mouse click on Parts column in the left side, the parts to be created include:
  a) INLET
  b) OUTLET
  c) LEADING EDGE
  d) TRAILING EDGE
  e) WALLS
  f) SYMMETRY
- Now creating a body: Geometry> Create/Modify surface> Create Body> type part as LIVE >Material point> Select 2 opposite points> Apply> Ok.
- Now generate a Mesh: Mesh> Global Mesh setup> Global Mesh parameters> Global Mesh size> Enter the values.
- Mesh> Part Mesh setup> Enter Maximum size as 0.4 for body and 1 for walls & symmetry> Apply.
- Mesh> Compute Mesh> Volume Mesh> Tetra/Mixed mesh> Robust (octree) mesh method> Select entity > Compute.

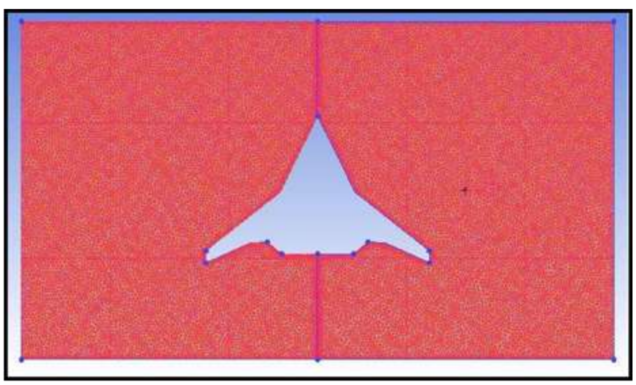


Fig: Meshed Model of Blended Wing Body with kink angle 38.3

Creating output file to Fluent V6: Output> Output to FLUENT V6> Part boundary conditions> Accept> Select destination folder> Grid Dimension-3D> Default settings> Done> Output file created ("fluent.msh" file).

DESIGN OF BWB WITH 41.3º KINK ANGLE

- Extract the coordinates from the draft sketch taken from reference paper.
- Open ICEM CFD software and import coordinates using: File> Import Geometry> Formatted Point Data> Select the coordinates file> Ok.
- Now create splines using: Geometry> Create/Modify Curve> from points> select the points> Apply> Ok.
- Create a control volume using approximate coordinates and join the points.

- Further create surface: First Control volume: Geometry> Create/Modify surface> Simple surface> from curves> Select curves> Apply> Ok and similarly repeat the step for creating surface of body.
- Now extrude the geometry: Geometry> Create/Modify surface> Sweep surface> Vector method> through points> Select 2 points with 4mm thickness between them> then select the whole geometry> Apply> Ok.
- Create parts by using right mouse click on Parts column in the left side, the parts to be created include: a) INLET b) OUTLET c) LEADING EDGE d) TRAILING EDGE e) WALLS f) SYMMETRY
- Now creating a body: Geometry> Create/Modify surface> Create Body> type part as LIVE >Material point> Select 2 opposite points> Apply> Ok.
- Now generate a Mesh: Mesh> Global Mesh setup> Global Mesh parameters> Global Mesh size> Enter the values.
- Mesh> Part Mesh setup> Enter Maximum size as 0.4 for body and 1 for walls & symmetry> Apply.
- Mesh> Compute Mesh> Volume Mesh> Tetra/Mixed mesh> Robust (octree) mesh method> Select entity > Compute.

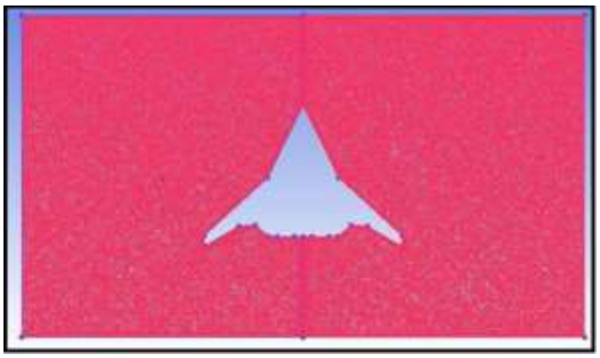


Fig: Meshed Model of Blended Wing Body with kink angle 41.3

Creating output file to Fluent V6: Output> Output to FLUENT V6> Part boundary conditions> Accept> Select destination folder> Grid Dimension-3D> Default settings> Done> Output file created ("fluent.msh" file)

DESIGN OF B2 WING

- Extract the coordinates from the draft sketch taken from reference paper.
- Open ICEM CFD software and import coordinates using: File> Import Geometry> Formatted Point Data> Select the coordinates file> Ok.
- Now create splines using: Geometry> Create/Modify Curve> from points> select the points> Apply> Ok.
- Create a control volume using approximate coordinates and join the points.
- Further create surface: First Control volume: Geometry> Create/Modify surface> Simple surface> from curves> Select curves> Apply> Ok and similarly repeat the step for creating surface of body.
- Now extrude the geometry: Geometry> Create/Modify surface> Sweep surface> Vector method> through points> Select two points with 4mm thickness between them> then select the whole geometry> Apply> Ok.
- Create parts by using right mouse click on Parts column in the left side, the parts to be created include: a) INLET b) OUTLET c) LEADING EDGE d) TRAILING EDGE e) WALLS f) SYMMETRY
- Now creating a body: Geometry> Create/Modify surface> Create Body> type part as LIVE >Material point> Select 2 opposite points> Apply> Ok.
- Now generate a Mesh: Mesh> Global Mesh setup> Global Mesh parameters> Global Mesh size> Enter the values.
- Mesh> Part Mesh setup> Enter Maximum size as 0.4 for body and 1 for walls & symmetry> Apply.
- Mesh> Compute Mesh> Volume Mesh> Tetra/Mixed mesh> Robust (octree) mesh method> Select entity > Compute.

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

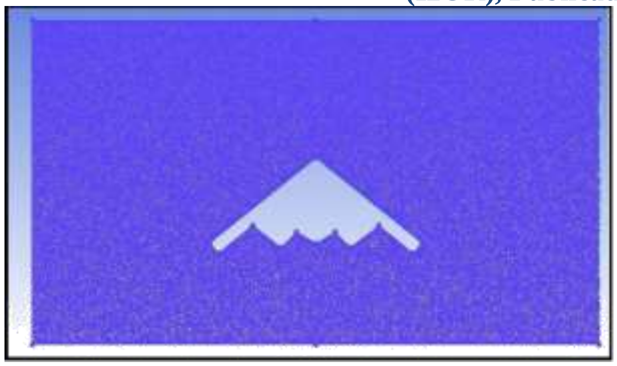


Fig: Meshed Model of B2 Wing

MESHING:

Mesh generation is one the most critical aspects of engineering simulation. Too many cells may result in long solver runs, and too few may lead to inaccurate results. ANSYS Meshing technology provides a means to balance these requirements and obtain the right mesh for each simulation in the most automated way possible. ANSYS Meshing technology has been built on the strengths of stand-alone, class leading meshing tools. The strongest aspects of these separate tools have been brought together in a single environment to produce some of the most powerful meshing available. As CFD has developed, better algorithms and more computational power have become available to CFD analysts, resulting in diverse solver techniques. One of the direct results of this development has been the expansion of available mesh elements and mesh connectivity (how cells are connected to one another). The easiest classifications of meshes are based upon the connectivity of a mesh or on the type of elements present. The highly automated meshing environment makes it simple to generate the following mesh types:

1. Tetrahedral
2. Hexahedral
3. Prismatic inflation layer
4. Hexahedral inflation layer
5. Hexahedral core
6. Body fitted Cartesian
7. Cut cell Cartesian

Consistent user controls make switching methods very straight forward and multiple methods can be used within the same model. Mesh connectivity is maintained automatically. Different physics requires different meshing approaches. Fluid dynamics simulations require very high-quality meshes in both element shape and smoothness of sizes changes. Structural mechanics simulations need to use the mesh efficiently as run times can be impaired with high element counts. ANSYS Meshing has a physics preference setting ensuring the right mesh for each simulation.

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

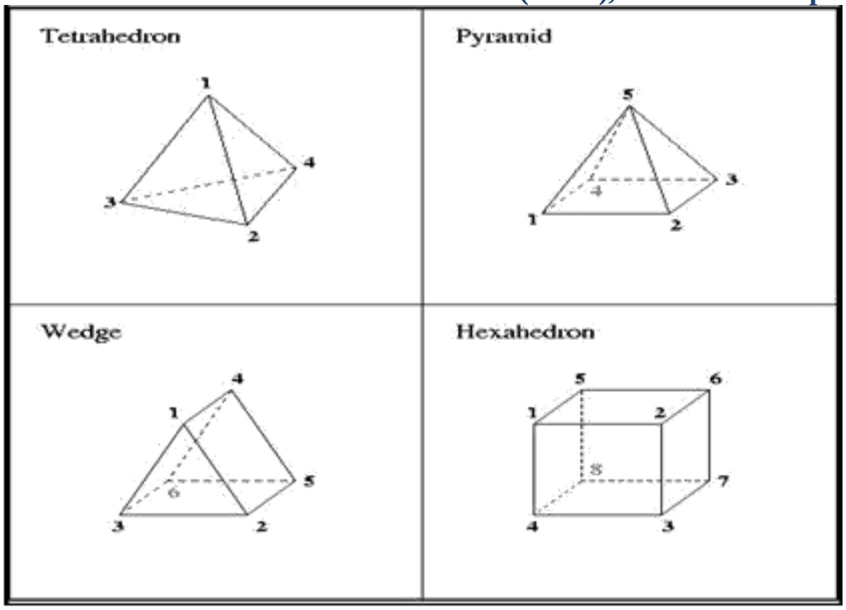


Fig: Tetrahedron and Hexagonal Mesh

There are often some misunderstandings regarding structured/unstructured mesh, meshing algorithm and solver. A mesh may look like a structured mesh but may/may not have been created using a structured algorithm based tool. For e.g., GAMBIT is an unstructured meshing tool. Therefore, even if it creates a mesh that looks like a structured (single or multi-block) mesh through pain-staking efforts in geometry decomposition, the algorithm employed was still an unstructured one. On top of it, most of the popular CFD tools like, ANSYS FLUENT, ANSYS CFX, etc. are unstructured solvers which can only work on an unstructured mesh even if we provide it with a structured looking mesh created using structured/unstructured algorithm based meshing tools. ANSYS ICEM CFD can generate both structured and unstructured meshes using structured or unstructured algorithms which can be given as inputs to structured as well as unstructured solvers, respectively.

GRID INDEPENDENCE TEST

The Grid Independence Test is found out to ensure the following:

- Error values have reduced to an acceptable value
- Monitor points for our values of interest have reached a steady solution
- The domain has imbalances of less than 1%

	Max size=0.4	**Max Size=0.7**	**Max Size=0.9**
BWB-38.3	707179	707176	635239
BWB-41.3	645971	1360960	1360960
B2	695768	1348048	1391536

Fig: Varying Cell Count with Respect to Maximum Mesh Size

After comparing the different mesh sizes, an observation is made that maximum mesh size of 0.4 is giving better results than the other sizes. Hence we chose Max mesh size of 0.4 for mesh generation.

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

CFD ANALYSIS

In this we discuss about the CFD analysis processes using the FLUENT V6 Software for analysis of Blended Wing Body and B2 Wing models. The Various methods and procedures opted for the analyses are being discussed for the better understanding of the analysis. The various input parameters and boundary conditions are also reviewed and the solution generation is done.

For this project, complete process has to be repeated for three different models:

- Blended body with 38.3ºkink angle
- Blended body with 41.3º kink angle
- B2 wing

The Above mentioned models are designed and undergone with the above mentioned steps that will result in the drafting of new suitable design that is capable of generating better performance.

FLUENT:

FLUENT has two solvers: Segregated solver and Coupled solver. Using either method, FLUENT will solve the governing integral equations for the conservation on mass and momentum, and energy if required and other scalars such as turbulence and chemical species. In both cases a control-volume-based technique is used that consists of division of the domain into discrete control volumes using a computational grid, integration on the governing equations on the individual control volumes to construct algebraic equations for the discrete dependent variables such as velocities, pressure, temperature, and conserved scalars and Linearization of the discretized equations and solution of the resultant linear equation system to yield updated values of the dependent variables. The segregated solver is the solution algorithm in which, the governing equations are solved sequentially (i.e., segregated from one another). The coupled solver solves the governing equations of continuity, momentum, and (where appropriate) energy and species transport simultaneously (i.e., coupled together). Governing equations for additional scalars will be solved sequentially (i.e., segregated from one another and from the coupled set) using the procedure described for the segregated solver. Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. Each iteration consists of the following steps: Fluid properties are updated based on the current solution (If the calculation has just begun the fluid properties will be updated based on the initialized solution). The continuity, momentum, and energy and species equations are solved simultaneously, where appropriate, equations for scalars such as turbulence and radiation are solved using the previously updated values of the other variables. A check for convergence of the mass & momentum is made. These steps are continued until convergence criteria are met. In both the segregated and coupled solution methods, the discrete, non-linear governing equations are linearized to produce a system of equations for the dependent variables in every computational cell. The resultant linear system is then solved to yield an updated flow-field solution. The manner in which the governing equations are linearized may take an 'Implicit' or 'Explicit' form with respect to the dependent variable (or set of variables) of interest. In 'Implicit' type of linearization, for a given variable, the unknown value in each cell is computed using a relation that includes both existing and unknown values from neighboring cells. Therefore, each unknown will appear in more than one equation in the system, and these equations must be solved simultaneously to give the unknown quantities. The coupled implicit approach solves for all variables (p, u, v, w, T) in all cells at the same time. In 'Explicit' linearization, for a given variable, the unknown value in each cell is computed using a relation that includes only existing values. Therefore each unknown will appear in only one equation in the system and the equations for the unknown value in each cell can be solved one at a time to give the unknown quantities. Each equation in the coupled set of governing equations is linearized explicitly which results in a system of equations with N equations for each cell in the domain. And likewise, all dependent variables in the set will be updated at once. The solution is updated using a multi-stage (Runge - Kutta) solver. Here you have the additional option of employing full approximation storage (FAS) multi-grid scheme to accelerate the multi-stage solver is adiabatic. The coupled explicit approach solves for all variables (p,u,v,w,T) in one cell at a time.

OVERVIEW OF PHYSICAL MODELS IN FLUENT:

Fluent provides comprehensive modelling capabilities for a wide range of incompressible and compressible, laminar and turbulent fluid flow problems. SteadyState or transient analyses can be performed. In Fluent, a broad range of mathematical models for transport phenomena is combined with the ability to model complex geometries. Examples of Fluent applications include laminar non-Newtonian flows in process equipment conjugate heat transfer in turbo machinery and automotive engine components, pulverized coal combustion in utility boilers, external aerodynamics, flow through compressors, pumps, and fans; and multiphase flows in bubble columns and fluidized beds. To permit modelling of fluid flow and related transport phenomena in industrial equipment and processes, various useful features are provided. These include porous media, lumped parameter (fan and heat exchanger), stream wiseperiodic flow and heat transfer, swirl, and moving reference frame models. The moving reference frame family of models includes the ability to model single or multiple reference frames. A time-accurate sliding mesh method, useful for modelling multiple stages in turbo machinery applications, for example, is also provided, along with the mixing plane model for computing time-averaged flow fields. Another very useful group of models in Fluent is the set of free surface and multiphase flow models. These can be used for analysis of gas-liquid, gas-solid, liquid-solid, and gas-liquid-solid flows. For these types of problems, Fluent provides the volume-of-fluid (VOF), mixture, and Eulerian models, as well as the discrete phase model (DPM). The DPM performs Lagrangian trajectory calculations for dispersed phases (particles, droplets, or bubbles), including coupling with the continuous phase. Examples of multiphase flows include channel flows, sprays, sedimentation, separation, and cavitations. Robust and accurate turbulence models are a vital component of the Fluent suite of models. The turbulence models provided have a broad range of applicability, and they include the effects of other physical phenomena, such as buoyancy and compressibility. Particular care has been devoted to addressing issues of near-wall accuracy via the use of extended wall functions and zonal models. Various modes of heat transfer can be modelled, including natural, forced, and mixed convection with or without conjugate heat transfer, porous media, etc. The set of radiation models and related sub-models for modelling participating media are general and can take into account the complications of combustion. A particular strength of Fluent is its ability to model combustion phenomena using a variety of models, including eddy dissipation and probability density function models. A host of other models that are very useful for reacting flow applications are also available, including coal and droplet combustion, surface reaction, and pollutant formation models.

VISCOUS MODEL – INVISCID FLOW:

Inviscid flow analyses neglect the effect of viscosity on the flow and are appropriate for high-Reynolds-number applications where inertial forces tend to dominate viscous forces. One example for which an inviscid flow calculation is appropriate is an aerodynamic analysis of some high-speed projectile. In a case like this, the pressure forces on the body will dominate the viscous forces. Hence, an inviscid analysis will give you a quick estimate of the primary forces acting on the body. After the body shape has been modified to maximize the lift forces and minimize the drag forces, you can perform a viscous analysis to include the effects of the fluid viscosity and turbulent viscosity on the lift and drag forces. Another area where inviscid flow analyses are routinely used is to provide a good initial solution for problems involving complicated flow physics and/or complicated flow geometry. In a case like this, the viscous forces are important, but in the early stages of the calculation the viscous terms in the momentum equations will be ignored. Once the calculation has been started and the residuals are decreasing, you can turn on the viscous terms (by enabling laminar or turbulent flow) and continue the solution to convergence. For some very complicated flows, this is the only way to get the calculation started. Various modes of heat transfer can be modelled, including natural, forced, and mixed convection with or without conjugate heat transfer, porous media, etc. Since inviscid flow problems will usually involve high-speed flow, you may have to reduce the under-relaxation factors for momentum (if you are using the pressurebased solver) or reduce the Courant number (if you are using the density-based solver), in order to get the solution started. Once the flow is started and the residuals are monotonically decreasing, you can start increasing the under-relaxation factors or Courant number back up to the default values.

VISCOUS MODEL - LAMINAR FLOW:

A laminar flow is characterized by a smooth flow of one lamina of fluid over another. Fluid elements move in well-defined paths and they retain the same relative position at successive cross section of the flow passage.

TURBULENCE MODELS

Fluent provides the following choices of turbulence models:

SPALART-ALLMARAS MODEL:

The Spalart-Allamaras is a relatively simple one equation model that solves a modelled transport equation for the kinematic eddy (turbulent) viscosity. In its original form, the Spalart-Allmaras model is effectively a low Reynolds-number model, requiring the viscous- affected region of the boundary layer to be properly resolved. In Fluent, however, the Spalart-Allmaras model has been implemented to use wall functions when the mesh solution is not sufficiently fine. This might make it the best choice for relatively crude simulations on coarse meshes where accurate turbulent flow computations are not crucial. Furthermore, the near wall gradients of the transported variable in the model are much smaller than the gradients of the transported variable in the k-Ɛ and k-ɷ models. This might make the model less sensitive to numerical error when non-layered meshes are used near walls. For instance, it cannot be relied on to predict the decay of homogeneous, isotropic turbulence. Furthermore, one equation models are often criticized for their inability to rapidly accommodate changes in length scale, such as might be necessary when the flow changes abruptly from wall-bounded to a free shear flow.

STANDARD K-ɷ MODEL:

Standard k-ɷ model incorporates modifications for low-Reynolds-number effects, compressibility, and shear flow spreading. The k-ɷ model predicts free shear flow spreading rates are in close agreement with measurements for far wakes, mixing layers, round and radial jets, and is thus applicable to wall-bounded flows and free shear flows.

CHOOSING A TURBULENCE MODEL:

It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. To make the most appropriate choice of model for your application, you need to understand the capabilities and limitations. The purpose of this section is to give an overview of issues related to the turbulence models provided in Fluent. The computational effort and cost in terms of CPU time and memory of the individual models is discussed. While it is impossible to state categorically which model is best for a specific application, general guidelines are presented to help you choose the appropriate turbulence model for the flow you want to model.

MATERIAL SELECTION

NAME	AIR
MATERIAL TYPE	FLUID
DENSITY	IDEAL GAS
CP (SPECIFIC HEAT)	CONSTANT (1006.43)
THERMAL CONDUCTIVITY	CONSTANT(0.0242)
VISCOSITY	CONSTANT(1.7894e-05)

Table : Material Selection for the Models

BOUNDARY CONDITIONS

INLET 1	VELOCITY INLET (246 m/s)
OUTLET 1	PRESSURE OUTLET
LEADING EDGE 12	INTERIOR

TRAILING EDGE 12	INTERIOR
TRAILING EDGE 3	INTERIOR
TRAILING EDGE 4	INTERIOR
SYMMETRY 1	WALL

Table: Boundary Conditions of the Models

REFERENCE VALUES

AREA	646 M^2
DENSITY	0.38
ENTHALPY	0
LENGTH	1
PRESSURE	11220 Pa
TEMPERATURE	300 K
VELOCITY	30 m/s
VISCOSITY	1.7894e-05
RATIO OF SPECIFIC HEATS	1.4

Table: Reference Values

SOLUTION

SCHEME	SIMPLE
GRADIENT	LEAST SQUARE CELL BASED
FLOW	SECOND ORDER UPWIND
TURBULENT KINETIC ENERGY	FIRST ORDER UPWIND

TURBULENT DISSIPATION RATE	FIRST ORDER UPWIND

MONITORS

Edit the residual print and plots giving 1e-06 as absolute criteria in x, y and z velocity, momentum, continuity, and energy equations then click ok. Create the drag, lift and moment then click to console and plot and then select the desired wall zone where lift, drag coefficients are required.

SOLUTION INITIALIZATION

INITIALIZATION METHODS	STANDARD INITIALIZATION
COMPUTE FROM	INLET
REFERENCE FRAME	RELATIVE TO CELL ZONE

Table: Solution Initialization

SETUP

- Right click on setup Ansys fluent 14.0 window opens up as shown in fig. 4.1.
- Click on double precision as it yields more accurate drag prediction and it is recommended for real cases, as our project deals with 3-dimensional geometry hence double precision is recommended.
- Give processing options as serial.
- Then click on ok.
- Fluid flow (fluent) FLUENT [3d, dp, pbns, lam] [ANSYS CFD] pops up.

Fig: Fluent launcher pop-up

Steps for Analysis in fluent for BWB 38.3º kink angle

- General - Density- based-absolute- steady.
- Models - Viscous- K- epsilon(2 equation)
- Materials - air-constant
- Boundary conditions: Inlet velocity- 246 m/sec Gauge pressure- 11220 Pa Outlet- 11220 Pa
- Solution methods - Second Order upwind
- Monitors - plot Cl, Cd, Cm
- Solution initialization - Standard Initialization- compute from inlet
- Run calculation - number of iterations-10,000-Reporting interval-Calculate and the iteration proceeds and wait for the monitors to get converged.

- Results - Graphs and Animations-contours and vectors of velocity, pressure.
- Plots: In plots we can get the XY PLOT for the converged results.
- Reports: From this we can get the forces at any required point.

Steps for Analysis in fluent for BWB 41.3º kink angle

- General - Density- based-absolute- steady.
- Models - Viscous- K- epsilon(2 equation)
- Materials - air-constant
- Boundary conditions: Inlet velocity- 246 m/sec Gauge pressure- 11220 Pa Outlet- 11220 Pa
- Solution methods - Second Order upwind
- Monitors - plot Cl, Cd, Cm
- Solution initialization - Standard Initialization- compute from inlet
- Run calculation - number of iterations-10,000-Reporting interval-Calculate and the iteration proceeds and wait for the monitors to get converged.
- Results - Graphs and Animations-contours and vectors of velocity, pressure.
- Plots: In plots we can get the XY PLOT for the converged results. Reports: From this we can get the forces at any required point.

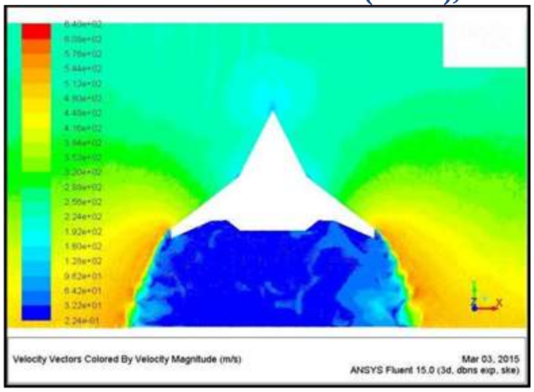
Steps for Analysis in Fluent for B2 Wing

- General - Density- based-absolute- steady.
- Models - Viscous- K- epsilon(2 equation)
- Materials - air-constant
- Boundary conditions: Inlet velocity- 246 m/sec Gauge pressure- 11220 Pa Outlet- 11220 Pa
- Solution methods - Second Order upwind
- Monitors - plot Cl, Cd, Cm
- Solution initialization - Standard Initialization- compute from inlet
- Run calculation - number of iterations-10,000-Reporting interval-Calculate and the iteration proceeds and wait for the monitors to get converged.
- Results - Graphs and Animations-contours and vectors of velocity, pressure.
- Plots: In plots we can get the XY PLOT for the converged results.
- Reports: From this we can get the forces at any required point.

RESULTS AND DISCUSSIONS

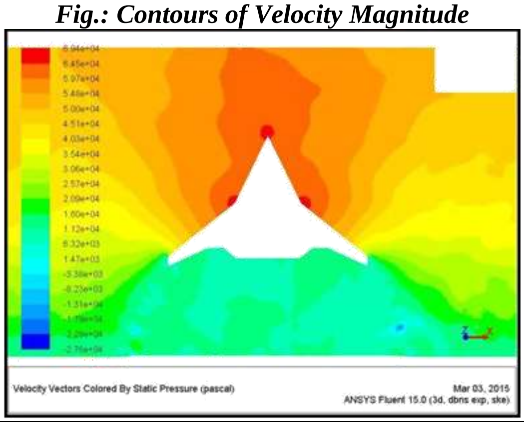
The Blended wing body and B2 wing is designed based on the dimensions from the Faliang Wang reference papers and then analysis is carried. The results are plotted and are validated. Further based on the obtained results new model with increased kink angle is designed that has greater efficiency and lift generation at subsonic and supersonic speeds. This Chapter contains all the results and validations of the wing that are obtained during the course of project.

ANALYSIS RESULTS OF BWB



Min. Velocity	32 m/s (KINK)
Max. Velocity	640 m/s (WING TIP)

Fig.: Contours of Velocity Magnitude

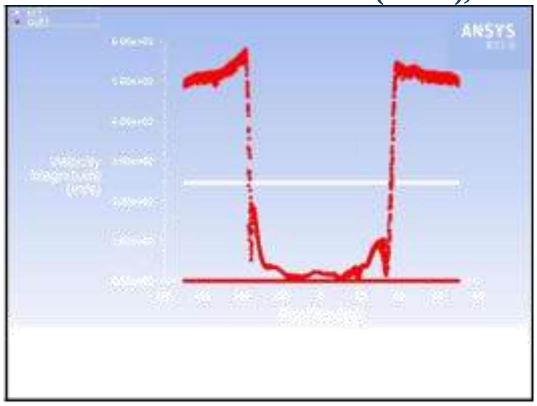


Min. Pressure	20900 Pa (WING TIP)
Max. Pressure	69400 Pa (NOSE & KINK)

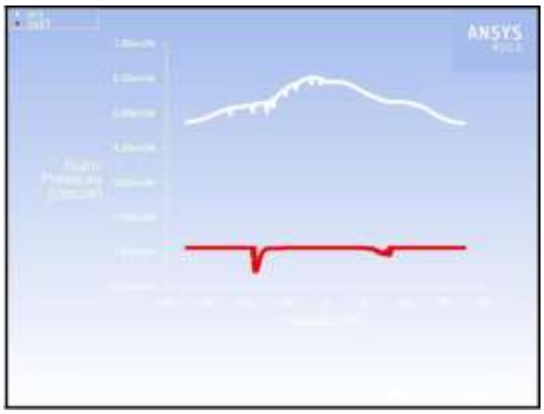
Fig: Contours of Static Pressure

From the above contours of static pressure, the maximum pressure is found at the nose and kink section which is of 69400Pa. The minimum pressure is observed at the wing tips which is 16000Pa. The main observation from the above contours is that the pressure is decreasing along the wing from nose to the wing tips.

Pressure and Velocity Plots



Graph: Velocity Magnitude with Respect to Position



Graph: Static Pressure with Respect to Position

"Force Report"

Center of Pressure - Set Coordinate x=0 (m)

Zone	y	z
sym1	28.84596	6021.236
sym2	-143.802	4864.034
bwb_le1	25.65686	2.571738
bwb_te1	-0.13994	2.420839
bwb_re1	5.26E+14	-3.0322
bwb_re2	12.97338	2.178023
bwb_re3	-45.5166	2.068119
fs_wall1	-14.2886	1.556326
Net	-12.2288	-69.8763

Flux Report

Mass Flow Rate	(kg/s)
bwb_le1	0
bwb_re1	0
bwb_re2	0
bwb_re3	0
bwb_te1	0
fs_wall1	0
in1	52294.92
int_live	-51672.9
out1	-49329.9
sym1	0
sym2	0
Net	2965

Table: Centre of Pressure Report and Mass Flow Rate Report

Surface Integral Report

Flow Rate

Static Pressure	(pascal)(kg/s)
bwb_le1	0
bwb_re1	0
bwb_re2	0
bwb_re3	0
bwb_te1	0
fs_wall1	0
in1	1.90E+09
int_live	-1.28E+09
out1	-5.91E+08
sym1	0
sym2	0
Net	23005080

Surface Integral Report

Area	(m2)
bwb_le1	389.4392
bwb_re1	68
bwb_re2	35.23634
bwb_re3	124.1209
bwb_te1	22.35767
fs_wall1	640.0001
in1	559.4286
int_live	490379.4
out1	559.4495
sym1	10550.84
sym2	10551.24
Net	513829.3

Table: Surface Integral Reports

The above reports show the variations of Centre of pressure, Mass flow rate, Static pressure and Area with respect to parts of the model. Net values of the reports are shown in the tables.

Validation

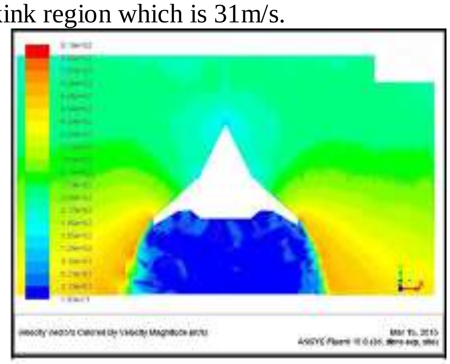
The Blended wing body is designed using the draft sketch given in the reference paper and analyzed at same boundary conditions. The results obtained from the reference paper and the results from our designed model are tabulated and an observation is made that the results are nearly matching i.e. the software used for analysis in reference paper is different from us. But satisfactory results are obtained from the analysis, Hence we can say that the reference paper is being validated with our designed blended wing body with 38.3 kink angle.

	Cl	**Cd**	**L/D**
Reference Model	0.226	0.0107	21.121
Designed Model	0.123	0.00437	28.60

Table: Comparison of Cl, Cd and L/D Values of Reference Model and the Designed Model for Validation

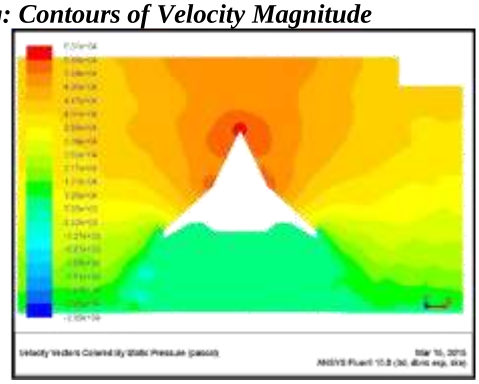
Results of Proposed Model

From the above contours of velocity magnitude we can see that the initial inlet velocity is reduced from 246m/s to 217m/s due to initial shock formations. The maximum velocity is observed at wing tips - 619m/s and the minimum velocity is observed at the nose and kink region which is 31m/s.



Min. Velocity	***31 m/s (KINK)***
Max. Velocity	***619 m/s (WING TIP)***

Fig: Contours of Velocity Magnitude

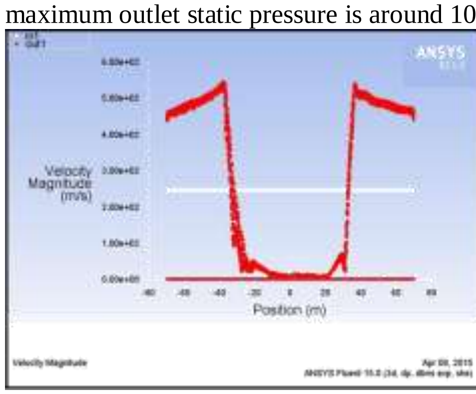


Min. Pressure	12500 Pa (WING TIP)
Max. Pressure	63100 Pa (NOSE)

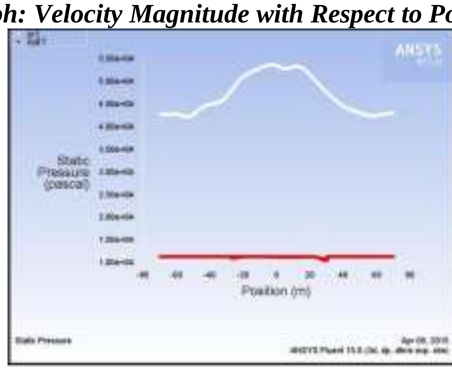
Fig: Contours of Static Pressure

Pressure and Velocity Plots

The XY plots of Velocity Magnitude and the Static pressure are plotted separately with respect to their relative position. An observation is made that the inlet velocity is constant at the inlet and the outlet velocity is changing along with the position on the body i.e., the maximum outlet velocity is found at the wing tips around 550m/s and minimum velocity is found at aircraft middle section which is 0m/s. The maximum static pressure at the inlet region is found to be 55000Pa at the nose part and the maximum outlet static pressure is around 10000Pa along the whole body.



Graph: Velocity Magnitude with Respect to Position



Graph: Static Pressure with Respect to Position

		"Force Report"	
Center of Pressure - Set Coordinate (m)			
Zone		y	z
far1		-2.38613	2.007025
sym12		19.4413	-98.2261
le12		-0.90797	1.711047
tr1		-1.73258	1.519442
tre12		-20.4305	2.825051
tre3		20.16989	2.042771
tre4		-6.30E+09	353827.2
tre5		0	0
-------	------	------------	------------
Net		-1.00658	-0.15627

Flux Report		
		(kg/s)
Mass Flow Rate		
-------------------		
far1		0
in1		52348.8
int_live		-53597.8
le12		0
out1		-53764.9
sym12		0
tr1		0
tre12		0
tre3		0
tre4		0
tre5		0
-------------------		------------
Net		-55013.9

Table: Centre of Pressure Report and Mass Flow Rate Report

 ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

Surface Integral Report	
Flow Rate	
Static Pressure	(pascal)(kg/s)
far1	0
int1	2.48E+09
int_live	-1.33E+09
le12	0
out1	6.08E+08
sym12	0
tr1	0
tre12	0
tre3	0
tre4	0
tre5	0
Net	2.34E+08

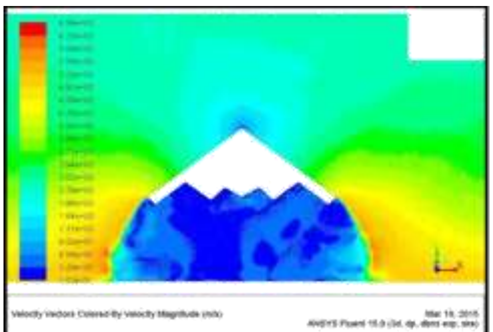
Surface Integral Report	
Area	(m2)
far1	840
in1	560
int_live	473032
le12	331.1167
out1	560
sym12	21165.13
tr1	20.79667
tre12	79.57246
tre3	68.04855
tre4	33.62372
tre5	34.4
Net	496524.7

Table: Surface Integrals Report

The above reports show the variations of Centre of pressure, Mass flow rate, Static pressure and Area with respect to parts of the model. Net values of the reports are shown in the tables.

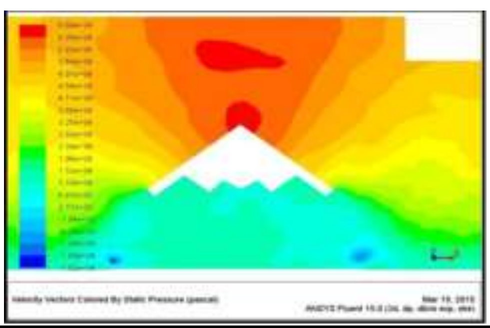
Results of B2 wing

The B2 wing body is designed using the draft sketch available from our reference paper and analyzed it under the same boundary conditions as for the other two models.



Min. Velocity	32 m/s (NOSE)
Max. Velocity	654 m/s (WING TIP)

Fig: Contours of Velocity Magnitude

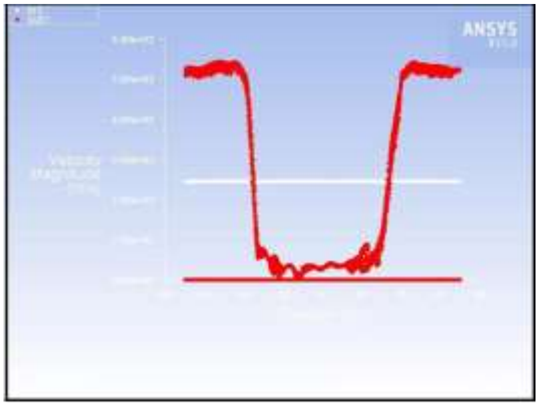


Min. Pressure	19600 Pa (WING TIP)
Max. Pressure	66900 Pa (NOSE)

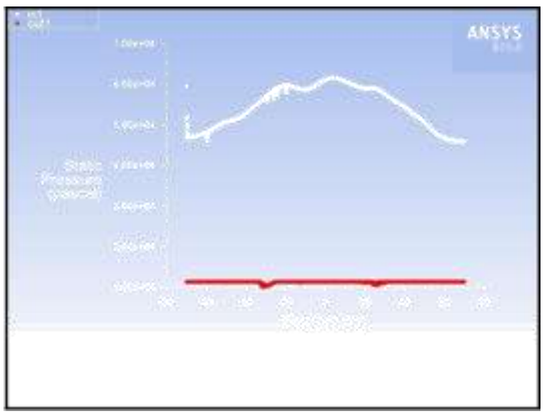
Fig: Contours of Static Pressure

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

Pressure and Velocity Plots



Graph: Velocity Magnitude with Respect to Position



Graph: Static Pressure with Respect to Position

		"Force Report"		
Center of Pressure- Set Coordinate x=0 (m)				
Zone		y		z
geom		-164.617		-2685.22
le1		-315.808		0.476863
te1		5.74898		2.167773
te2		-5.77945		2.280756
te3		20.96318		2.14499
te4		-31.1373		2.072028
te5		21.96514		2.055107
wall		-47.2448		2.038383
------	-----	-------------	--	-------------
Net		-38.5681		48.00696

Table: Centre of Pressure Report

"Flux Report"		
Mass Flow Rate		(kg/s)
geom		0
in1		52325.58
int_live		-33597.5
le1		0
out1		-54904.3
te1		0
te2		0
te3		0
te4		0
te5		0
wall		0
Net		-2578.73

"Surface Integral Report"		
Area		(m2)
geom		21410.88
in1		559.7517
int_live		493278.5
le1		268.6195
out1		559.8475
te1		44.45582
te2		42.99209
te3		70.72652
te4		86.45924
te5		26.04612
wall		639.9512
Net		516988.2

Table: Flux Report-Mass Flow Rate report and the Surface Integral Report

COMPARISON OF RESULTS

Comparison of Static pressures

	BWB (38.3 kink angle)	**BWB (41.3 kink angle)**	**B2 Wing**
NOSE	69400 Pa	63100 Pa	66900 Pa
KINK	69400 Pa	63100 Pa	-
WING TIP	16000 Pa	12500 Pa	19600 Pa

Comparison of Velocities

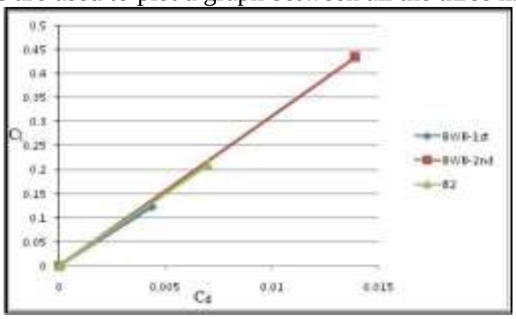
	BWB (38.3 kink angle)	**BWB (41.3 kink angle)**	**B2 Wing**
NOSE	32.2 m/s	31.1 m/s	32.9 m/s
KINK	64.2 m/s	62.1 m/s	-
WING TIP	640 m/s	619 m/s	654 m/s

C_l, C_d values are obtained from the results of fluent analysis

	Values from Journal	**BWB**	**BWB**	**B2 Wing**
C_l	0.226	0.123	0.435	0.2103
C_d	0.0107	0.00437	0.01388	0.00698

L/D	21.121	28.60	31.33	30.12

After comparative observations we can conclude that the Blended wing body with 41.3 kink angle has better aerodynamic performance than the other two models i.e. this model has increased Lift coefficient and with reduced drag coefficient. The same results are used to plot a graph between all the three models.



Graph: C_l v/s C_d Graph of All Three Configurations

X axis= Coefficient of Drag - C_d
Y axis= Coefficient of Lift - C_l
The lift curves of three configurations are presented in the above Graph. It is clear that the Blended wing body with 41.3 kink angle has the highest lift curve slope while the other has low lift comparatively.

CONCLUSION

The Blended Wing Body "proposed model" is designed for its better efficiency and high performance at subsonic speeds. The drafted design is analyzed and the results are validated with the results of the reference papers. The lift generated by the proposed model is found to be more than the other two analyzed models. There was also a reduction in drag for the proposed design of the blended wing body. One of the most significant differences between Blended Wing Body and B2 wing aircraft is that the body of blended wing body generates the lift. Additionally, according to the results, there is reduction of drag for blended wing body than the other configuration. Therefore, with extended lift generation surface as well as reduced drag, the Blended wing body with 41.3 degrees kink angle has better performance. The proposed two-dimensional model is thus successful in acquiring higher lift and lower drag in the speeds of up to 0.85 Mach for the kink angle of 41.3 degrees. Hence we can say that the proposed new model is better in performance than the blended wing body model with a kink of 38.3 degrees and the B2 wing model.

FUTURE SCOPE

The proposed model which is analyzed by us is a two dimensional model, because of the less feasibility of resources and requirements we could not go for a three dimensional model. So it can be experimented for its perfectness in three-dimensional model at subsonic and supersonic speeds and can be used in the real time. Since it has a higher amount of lift and lower drag, this can prove to be a useful resource in better performance of aircrafts especially commercial aircrafts.Further research could pay more attention on investigating the impact of a wide range of parameters like the wing span and the position of the kink etc. Since there are so many parameters closely linked together in a bended wing body aircraft, it is of great interest to further research on the optimization of those parameters.

REFERENCES

[1] Faliang Wang -"The Comparison of Aerodynamic and Stability Characteristics between Conventional and Blended Wing Body Aircraft", Cranfield University, 2012.
[2] Liebeck,R.H,-Design Of The Blended Wing Body Subsonic Transport, Journal of Aircraft, vol.41, no.1, pp.10-25, 2004.
[3] Aeroservoelastic Characteristics of the B-2 Bomber and Implications for Future Large Aircraft R. T. Britt*, J. A. Volk, D. R. Dreim, and K. A. Apple white.
[4] Aerodynamic Analysis of a Blended-Wing-Body Aircraft Configuration, Toshihiro Ikeda,School of

Aerospace, Mechanical and Manufacturing Engineering Science, Engineering and Technology Portfolio RMIT University March 2006.

[5] Wind Tunnel Experiments and CFD Analysis of Blended Wing Body (BWB) Unmanned Aerial Vehicle (UAV) at Mach 0.1 and Mach 0.3, Wirachman Wisnoe, Rizal Effendy Mohd Nasir, Wahyu Kuntjoro, and Aman Mohd Ihsan Mamat, May 2009.

[6] A Review Of Swept And Blended Wing Body Performance Utilizing Experimental, Fe And Aerodynamic Techniques, Hassan Muneel Syed, M. Saqib Hameed & Irfan A. Manarvi, Sept 2011.

[7] Aeroservoelastic Characteristics of the B-2 Bomber and Implications for Future Large Aircraft, R. T. Britt*, J. A. Volk, D. R. Dreim, and K. A. Applewhite.

[8] The Blended-Wing-Body Super Jumbo Jet Concept, National Aeronautics and Space Administration, Langley Research Center, July 1997.

[9] Boeing Blended Wing Body Project, Amy Brzezinski, Tanya Cruz Garza, Julia Thrower, Brady Young, 2013.

[10] College Of Aeronautics Blended Wing Body Development Programme, H. Smith, College Of Aeronautics, Cranfield University, 2000.