ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE)
Vol. 7, Issue 4, April 2021

ANALYSIS OF ICE FORMATION OVER AN AIRCRAT WING

*Yogeswaran S [#1], Mathew Joseph [#2], Martin Joseph [#3], Vidhyasagar K [#4], Jithin Malayil Jacob*5*

[#]Student, PSN College of Engineering and technology, Tirunelveli, Tamil Nadu, India

**Assistant Professor, PSN College of Engineering and technology, Tirunelveli, Tamil Nadu, India*

#1701037@psncet.ac.in, 1701021@psncet.ac.in, 1701019@psncet.ac.in, 1701035@psncet.ac.in

*jithinjacob1000@gmail.com

***Abstract*— our goal is to analysis of airplane in-flight icing is a serious problem, still causing many accidents. Formation of a solid ice cover with different structures and shapes of different speed rates and intensities on airplane external surfaces causes an increase of roughness and alters the flow over the lifting devices.**

Taking this problem into account, a numerical simulation has been carried out to model the de-icing simulation over the Airbus A380 wing section in this project. The right wing of the Airbus A380 has been considered for de-icing CFD simulation instead of considering the whole aircraft. The wing section along with the winglets has been modeled used CAD modeling software CATIA V5. The major objective of the work is to predict the aerodynamic drag coefficient of the wing with leading edge rime ice and performing the CFD simulation of de-icing process to compute the total time taken for melting the solid ice over the wing.

The pre-processing work for CFD simulation has been carried out using ANSA and TGRID. The tetrahedral elements are used for meshing the wind tunnel domain. The prism elements which are most commonly used to capture the boundary layer are used to model the ice formation over the upper surface of the wing. The prism layers are layered as 5mm ice over the upper surface of the wing. Solidification and melting model is FLUENT CFD solver is used to solve the melting process with the application of thermal heating of the leading edge part of the wing by using the bleed air from the exit of the compressor. The results of the CFD simulation shows that, the rapid de-icing of the ice of the wing are possible with high thermal heating of the leading edge part of the wing with constrains in structural integrity because of high thermal loading.

***Index Terms* - Aerodynamic lift and drag, airfoil.**

INTRODUCTION

Icing is a natural phenomenon where a surface is incident on Liquid Water Content (LWC) that is present in clouds and ice forms over the surface at certain atmospheric conditions. Icing can also happen during snowstorm and snowfall. We see the roofs of the houses, the roads surfaced with layers of snow. Airfoils that are in use at such snowy places will experience such icing.

Clouds are made of water droplets. This water droplets freeze on the surface it falls on, given the surface temperature is below the freezing point. As time goes on, layer after layer of ice gets accreted on the surface.

This is a major problem being faced by Aerospace Industries worldwide. When aircrafts fly through clouds, leading edges of the wing, inlet leading edge of the engine are affected by ice formation.

This happens to any surface in high-altitude regions, where the temperature is below 0°C. Aircrafts experience icing when they fly through clouds. The water droplets that are present in the clouds hit the aircraft's surface and the water droplets froze on impact that results in ice formation over that surface. The typical areas where the ice forms on aircrafts are the leading edges of the wing, nose, inlet leading edge of the engine cowling, propellers of a turboprop or piston prop aircrafts and leading edges of vertical and horizontal stabilizers.

Types of Ice

There are super cooled droplets at higher altitudes. These droplets form into ice on coming in contact with the aircraft surface. Based on the formation, there are two types of ice

- Rime ice (Dry ice)
- Glaze ice (Wet ice)

The Rime ice, is seen at lower temperatures (-40^{0}C to -10^{0}C) where the super cooled droplets freeze almost immediately onto the surface. In doing so, air is entrapped between the frozen droplets, which give rime ice its characteristic white appearance. Rime ice is normally attached to the leading edge with a streamlined shape where the aerodynamic drag is increased because of surface roughness and early boundary layer transition. Rime ice is less dense with a density of 880 kg/m^3 because of the entrapped air.

Glaze ice occurs at higher temperatures (-18^{0}C to 0^0) where liquid droplets freeze. Therefore, is it clear, with a density of 917 kg/m^3. The droplets impinge on the surface do not freeze instantaneously but form a film of liquid water that runs back over the surface, freezing gradually at various rates. Rime and Glaze ice tend to respectively form round and horned protrusions near the leading edge

Mixed ices are the types of ice that are partly rime and glaze. This mainly depends on the temperature. If the temperature is from the -6°C to -9°C, there is high probability that the ice formed is mixed – the appearance of the ice will be translucent.

ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE)
Vol. 7, Issue 4, April 2021

Effects of icing

It is due to the Lift acting on the airfoil that keeps the aircraft afloat and hence the lift produced plays a pivotal role in the stability of the aircraft. So, the Lift force and hence the Coefficient of Lift of that airfoil plays a very important role in this whole scenario. The most hazardous aspect of icing is its aerodynamic effects. Ice alters the shape of an airfoil, reducing the maximum coefficient of lift and angle of attack at which the aircraft stalls. It also increases drag and adversely affects the aerodynamic efficiency. The reader is referred to references [9-14] for related study on icing.

Therefore, when cruising at a low angle of attack, ice on the wing may have little effect on the lift. However, note that the ice significantly reduces the cl_{max} and the angle of attack at which it occurs (the stall angle) is much lower. Another is the mixed ice situation, where glaze ice is surrounded by delicate feather shaped rime ice formations.

R. Mikalsen, A.P Roskilly [1] in 2009 had made a study on performance of spark ignited free-piston engine generator using CFD. From this performance simulation study it is seen that the velocity of the piston is varying along its stroke length which depends up on the combustion gas pressure on the both sides of the combustion chamber. The in-cylinder gas pressure increases to higher values when the piston moves to TDC when compared to conventional engine. The particular operating characteristics of the free-piston engine were not found to give noticeable performance advantage. R. Mikalsen, A.P Roskilly [2] in 2009 had made an investigation based on design and simulation of a two-stroke free-piston compression engine. From this investigation it is seen that the operating characteristics of the free-piston engine was found to differ significantly from those of conventional engines, giving potential advantages in terms of fuel efficiency(i.e., better scavenging) and emission formation due to fast power stroke expansion. It is also seen than the effects of varying engine stroke length and compression ratio were not to give any large advantages A.Z.M. Fathallah and R.A. Bakar [4] in 2009 had made a study on design optimization of spring which is used for the purpose of return cycle of two stroke single cylinder linear engine. The optimization has been done based upon multilevel optimization approach. The thrust force data was conducted by GT- Power simulation software and optimized by using matlab nonlinear numerical integration technique. The effect of engine load and speed performance was analyzed by dynamic programming technique. They finally concluded that the spring wire diameter of 7mm can be used as spring system linear engine design and most variable performance of engine loads worked properly. R. Mikalsen, A.P Roskilly [5] in 2009 had investigated the free piston engine combustion by using coupled dynamic-multidimensional modeling using cfd. Solution dependent mesh motion had employed in the CFD code. This investigation results in the study of influence of poor ignition timing control in HCCI engines along with the potential benefits of lean-burn , high compression ratio operation. A.Z.M. Fathallah and R.A. Bakar [3] in 2010 had made an effort to optimize the spring design for return cycle of two stroke spark ignition linear engine. From this optimization it is seen than at certain conditions the engine cannot work properly as they predicted. Thus the results were significant drop of Indicated Mean Effective Pressure (IMEP) and impact is reduction of power output. M.M.Noor, K.Kadirgama, Devarajan R., M.S.M.Sani, M.F.M.Nawi, T.F.Yusaf (6) in 2008 had made a study on simulation of in-cylinder characteristics of a motored two stroke engine. They carried out the simulation using CFD software-Fluent. The contour of burned mass fraction of gas inside the cylinder has been seen for different crank angles and for opening and closing of transfer port and exhaust port. Semin, N.M.I.N. Ibrahim, Rosli A. Bakar and Abdul R. Ismail (7) in 2008 had made a study on two-stroke spark-ignition cross loop-scavenged port. The pressures on in-cylinder and intake port are collected and a validation is done with numerical results for 1400 rpm. The experimental results showed, the higher rpm mode produce the higher cylinder pressure than lower rpm.

I. METHODOLOGY

Methodology for CFD simulation of de-icing the ice formation over the leading edge of the aircraft wing for drag reduction involves

i. Modeling the aircraft wing using the CAD software CATIA.

ii. Pre-processing the modeled wing – CAD cleanup, domain creation and meshing.

iii. Applying boundary condition, solver setting and performing the CFD simulation for the modeled wing using FLUENT CFD solver.

iv. Monitoring the drag coefficient for the base case of the wing.

v. Post-processing the results.

vi. Modeling the ice solid 5mm over the leading edge of the wing.

vii. Again unsteady CFD simulation will be carried out along with flow and thermal in order to visualize the melting time of ice.

viii. Temperature on the leading edge increases gradually from 273k to 393 k. For this purpose a profile file has to be written in order to avoid divergence.

ix. Liquid fraction of ice is monitored on the leading edge until it reaches the value 1.

In this project, Airbus A380 Wing - Aerofoil NASA SC(02)-0610 on tip NASA SC(02)-0606 on root, The engine parameters shown in Table 1 are obtained from the engine manufacturer then the CAD model is developed based on the data obtained.

ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

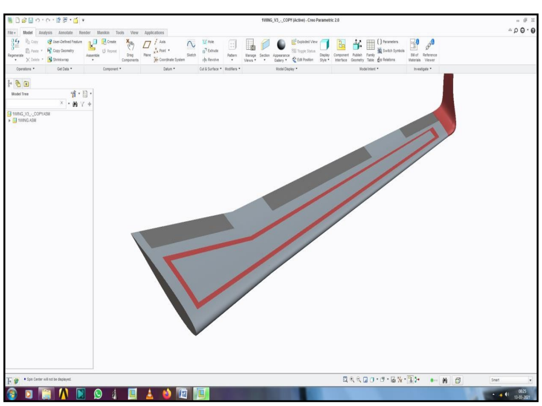
International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE)
Vol. 7, Issue 4, April 2021

Table 1 Airbus A380 wing specification and operating condition

Wing Specification	
Parameter	**Dimension**
Airfoil at root of the wing	NASA SC(02)-0606
Airfoil at tip of the wing	NASA SC(02)-0610
Length of the Wing	38m
Wing Root chord length	17.7m
Wing tip chord length	4.6m
Operating condition at Cruise condition (35000 ft)	
Free stream Mach no	0.8
Free stream velocity	272
Operating pressure (Absolute)	2.391E4 Pa
Ambient temperature	218.92K
Dynamic viscosity	1.434014985E-5 Pa.s

MODEL CONSTRUCTION

Three-dimensional model of the Airbus A380 wing with winglets was designed using the CAD design Creo and exported as IGS files. The IGS or IGES file format is a third party file format most probably supported in all commercially available CFD tool. The IGS files were then imported into ANSA, the mesh generator. In ANSA, the imported geometry is checked for topology. Then the flow domain is extracted from the imported model.



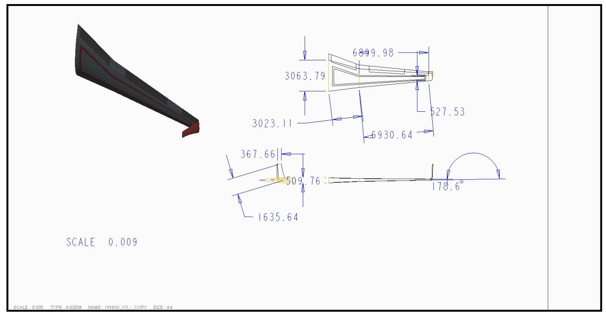


Figure 1 Modeled Airbus A380 (Right) wing with winglets

Here we found the model designed in such a way that the model impinges the float design contour curve profile design such that the ice formation is prevented in surface layer of the wing

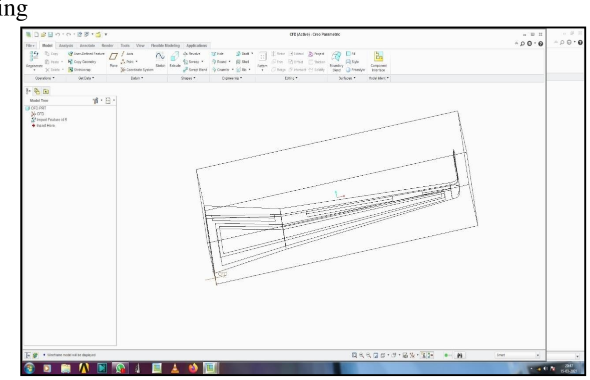


Figure 2 Modeled Airbus A380 (Right) wing with wind tunnel domain

CFD DOMAIN EXTRACTION OF THE MODELED AIRBUS A380 WING WITH WINGLETS

The first and far-most step is cfd preprocessing of modeled Airbus A380 wing is geometry clean up. This cleanup has been done using the ANSA meshing tool which is very robust clean up tool. Extracting the fluid region is the next step in which all the surfaces which are in the contact of fluid are taken alone and all other surfaces are removed completely. Extracted domain of Airbus A380 Aircraft wing with winglets and without engine was kept alone shown in Figure 3

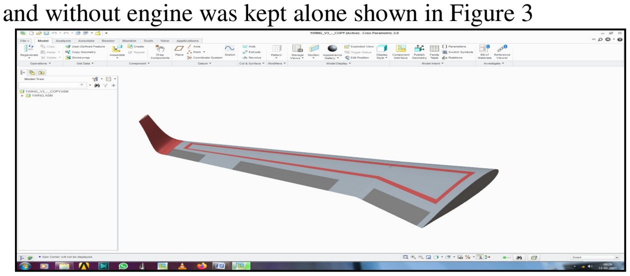


Figure 3 Geometry cleaned

ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE)
Vol. 7, Issue 4, April 2021

SURFACE MESH

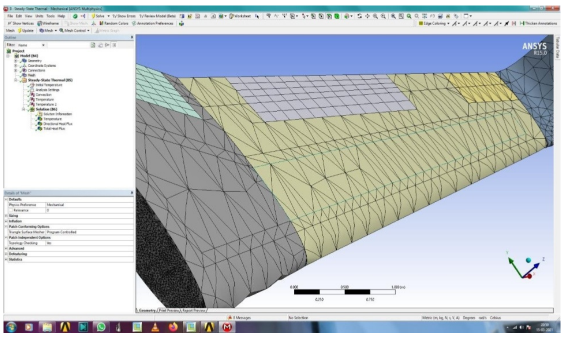


Figure 4 Surface mesh of the Airbus A380 wing (Base case)

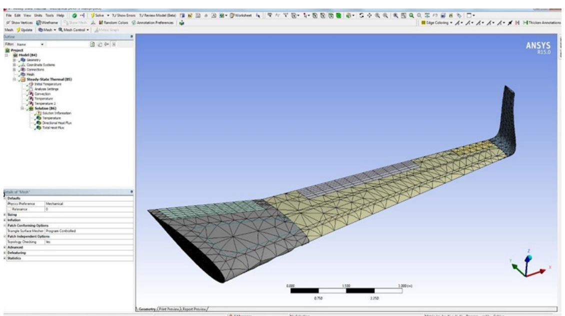


Figure 5 Surface mesh of the Airbus A380 wing (With leading edge ice)

VOLUME MESHING

The volume mesh for the static zone of the extracted domain of the wing with wind tunnel has generated using ANSYS-TGRID which is a robust volume mesh generator. The volume of the static zone of the extracted domain (Wing along with wind tunnel domain) is discretized using tetrahedral elements for interior of the tunnel and prism elements for the formation of ice. The prism elements are wedge shaped elements in which the higher order differential form of the navier-stokes system of equations are solved at each and every cell centroid of the elements for high accuracy results. Prism elements are also used to capture the boundary layer and recirculation in the static flow domain.

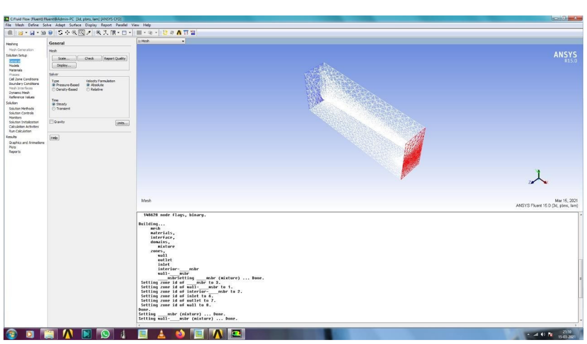


Figure 6 Volume mesh of static region of the domain

Table 2 Meshing details of the Airbus A380 wing with wind tunnel domain

Surface mesh details	
Element Type	Triangular [TRI]
Maximum Quality	0.6
Surface mesh count	250760
Volume mesh details	
Element Type	Tetrahedral [TET-for interior of wind tunnel & Prism (for creating the ice formation in the leading edge)
Maximum Quality	0.87
Volume mesh count	1750570

SOLVER SET UP AND METHODOLOGY

ANSYS-FLUENT is used as the solver for this case.

- External flow over the Airbus A380 wing is assumed to be 3-D, turbulent and compressible in nature.
- Steady simulation has been chosen to predict the drag for the base case of the wing.
- Unsteady simulation has been chosen for predicting the time taken to melt the leading edge ice in de-icing operation.
- Energy equation is activated (Density method changed to ideal gas from constant.
- Solidification and melting model in ANSYS-FLUENT has been chosen to model the melting of the leading edge ice over the wing.
- Spallart Alamras (1 Equation) turbulence model has been chosen to model turbulence of the free stream air flow over the wing.
- SIMPLE (Semi Implicit Pressure Linked Equations) algorithm is used to solve the problem.
- Segregated solver is used for pressure-velocity coupling.

MELTING MODEL AND BOUNDARY CONDITIONS SETUP

- Solidification and melting model in ANSYS-FLUENT'13 has been chosen to model the melting of ice (rapid de-iceing of leading edge ice).
- Default setting has been chosen(pull velocities of phase change.
- Solidification and melting model has been chosen instead of multiphase model in-order to simplify the simulation.

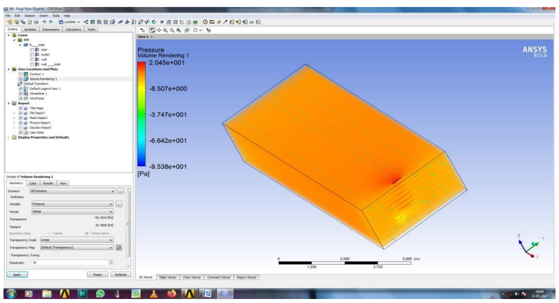
ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

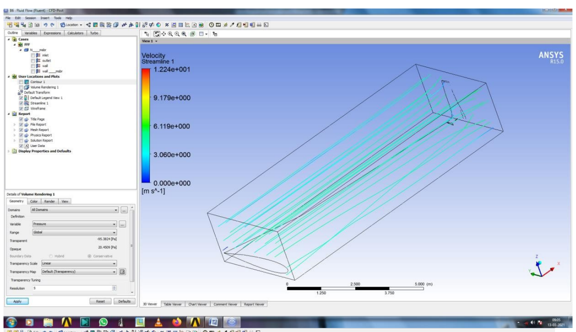
International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE)
Vol. 7, Issue 4, April 2021

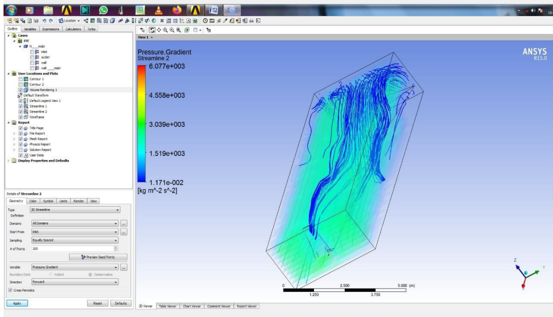
The Boundary conditions for this case are

- Wind tunnel Inlet is assumed to be velocity inlet with velocity of 272 m/s (0.8 mach)
- Temperature at inlet is assumed to be 218.92 K.
- Flow is assumed to be compressible; density of air options in material panel is changed to ideal gas.
- Dynamic viscosity is assumed as 1.434014985E-5 Pa.s
- Outlet is assumed to be pressure outlet with 0 Pascal (static pressure).

Pressure and Velocity Gradient along the wing







MATHEMATICAL MODELLING FOR HERMITE CUBIC SPLINE CURVE PROFILE

The kinematic energy K for an water mass M and wave velocity V

$K=1/2\ MV^2$

Water mass M can be expressed as

$M= \rho AtV$

ρ- Density of Air A-Sectional area t- Time

HENCE, KINETIC ENERGY

$K=1/2\ \rho AtV^3$

Now power is work done for unit mass and for one sec, hence power is

$P=1/2\ \rho V^3$

Now our derivation for pressure, Here the wave power density at temperature T and at pressure P can be written as

$Power = 0.61125 * (P/101325) * (288.15/T) * V^3$

For ideal gas at a standard temp of 15degree and sea level pressure of 101325Pa at a mass of 1.225 Kg per cubic meter

NOMENCLATURE

K - Kinematic energy M - Water mass

V - Wave velocity ρ - Density of water A -Sectional area

t – Time

P – Pressure

Now for the temperature of 22 degree when we calculate final derivation we obtained is

$Power = P*V^3$

$$\text{Speed (rmp)} = \frac{60{,}000 \times \text{speed (m/s)}}{\pi \times \text{diameter (mm)}}$$

Angular velocity (Rad/sec) =v /r

Torque (Nm) = 9.55 x Power (watts) / speed (Rpm)

Here Torque defines the force of action formed around the wing which indicates the

Pressure gradient – 6.077e3 Velocity Gradient – 12.4 m/s

Pressure – 2.45 Pa

INPUT VELOCITY (m/s)	MAX PRESSURE OUTPUT (Pa)	MAX VELOCITY OUTPUT (m/s)	SPEED (RPM)	ANGULAR VELOCITY	POWER(Kw)	TORQUE(NM)
2	2.45	12.4	168	4.47	25	1421.1

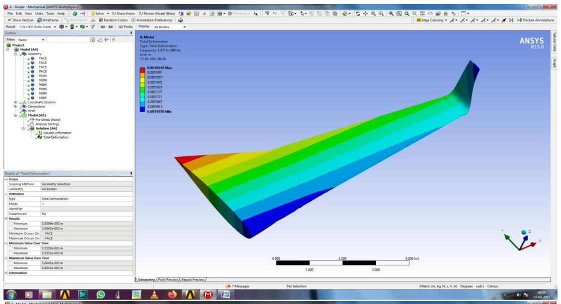
This max value of torque is more than sufficient to break the ice formation in wing. Hence the profile derived designed can be approved for wing design for deicing. Here the model derived can be more sufficient for working condition where the pressure gradient developed at the end of the wing curve produces a reverse back pressure which induces more torque development in the wing, which produces additional force development in the wing against ice formation.

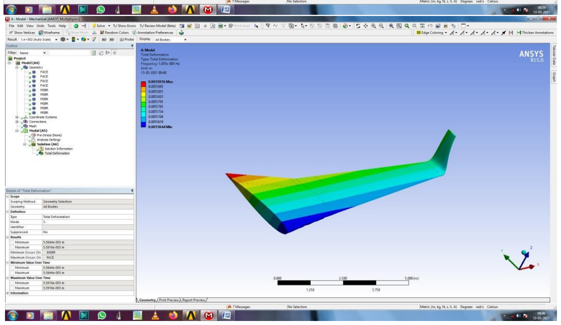
VIBRATION ANALYSIS

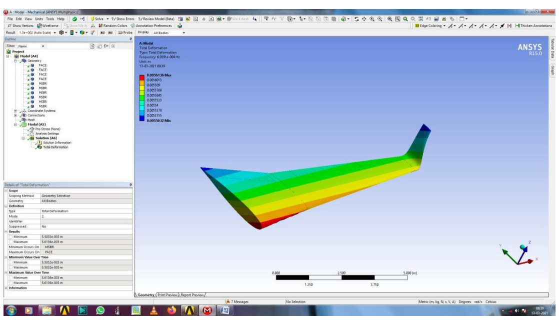
Free end Vibrational analysis

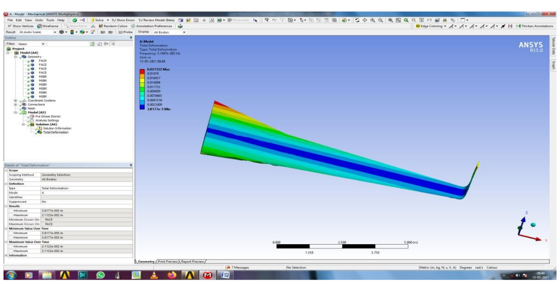
ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

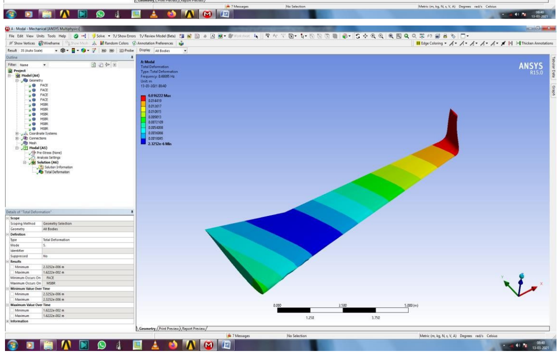
International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE)
Vol. 7, Issue 4, April 2021

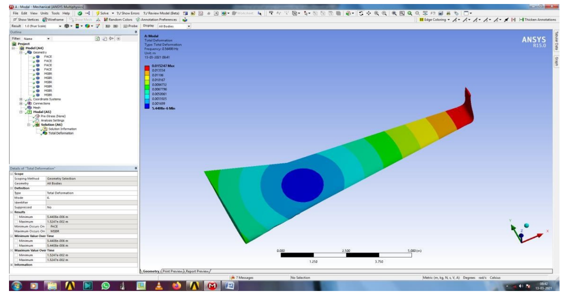




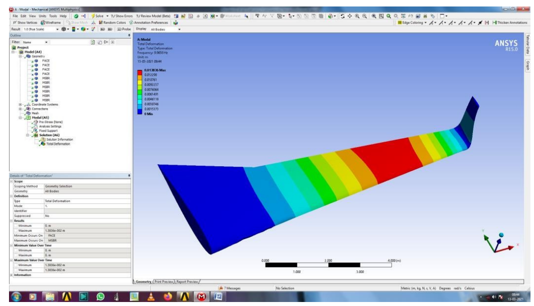


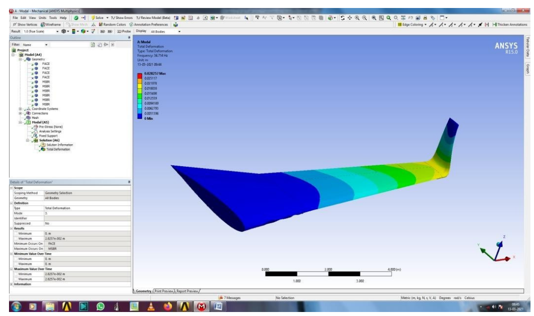


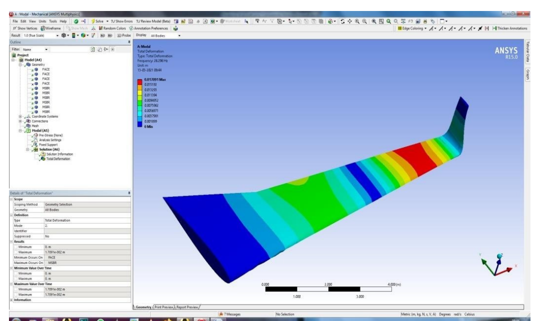




Constrained Vibration







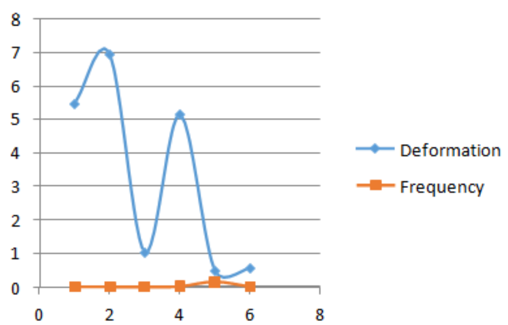
ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

IJARMATE
Your ulti-MATE Research Paper !!!

International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE)
Vol. 7, Issue 4, April 2021

Free Vibration

Deformation M	FrequencyHz
5.47	0.0056045
6.939	0.0056136
1.035	0.0055916
5.1497	0.021122
0.48895	0.16222
0.56498	0.015247



Constrained Vibration

Deformation M	FrequencyHz
9.9658	0.013836
26.296	0.017091
36.714	0.028257
50.04	0.020211
57.033	0.0287
80.535	0.060556

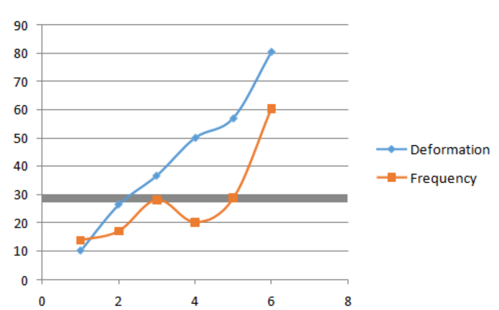
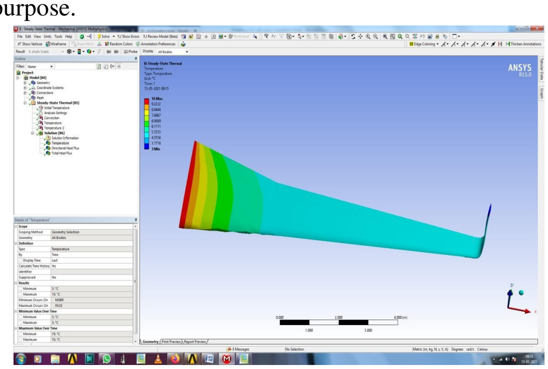


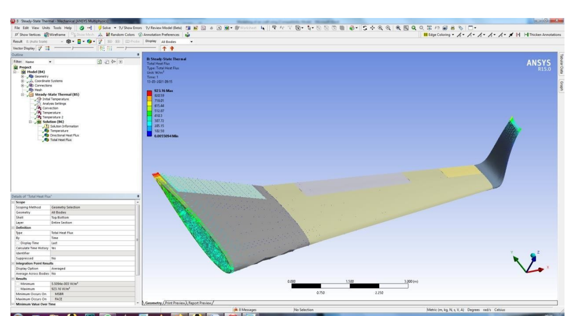
Fig .3 Meshed constituencies

The forced vibration of a structure with added constraint acting at a point is discussed. A set of constrained vibration modes is obtained in terms of the assumed known modes of the unconstrained structure, and it is shown that these constrained modes are orthogonal. It is shown that the unconstrained modes form a complete set for the constrained wing. The forced vibration response can be described in terms of either set of modes. The two descriptions are shown to be equivalent only if the damping is independent of the mode number. The damping may, however, be an arbitrary function of the forcing frequency. The frequency range and number of simultaneous excitation forces used significantly affects the sensitivity of damage detection. Based on this idea, a model independent method of damage detection using measured constrained vibration deflection shapes (VDS) and pre damage data is presented. The constrained VDS are the vibration shapes of the structure that occur when the forcing vector contains elements that cause the structure to vibrate in a desired pattern. The method is analytically tested using a finite- element model of a fixed-fixed beam wherein simulations are performed using one and two rotational excitations, and then a prescribed excitation vector that is comprised of the rotational degrees-of-freedom (DOFS) of one of the mode shapes of the healthy structure. The rotational excitation is used because it is convenient to apply at a large number of locations on the structure using thin piezoceramic patches. The translational vibration of the wing is used in the damage detection algorithm because the translational DOFs are convenient to measure using a scanning laser micrometer. Here the model of the wing found to be under safe condition for deicing effect.

Thermal Analysis

Thermal analysis is a general term defining a technique used to analyze the time and temperature at which physical changes occur when a substance is heated or cooled. Each technique is defined according to the types of physical changes being analyzed. When evaluating material characteristics, it is necessary to use different techniques or a combination of multiple techniques depending on the purpose.





ISSN (ONLINE): 2454-9762
ISSN (PRINT): 2454-9762
Available online at www.ijarmate.com

International Journal of Advanced Research in Management, Architecture, Technology and Engineering (IJARMATE)
Vol. 7, Issue 4, April 2021

Here Found the temperature of the wing does not found to be under mixed conditioning, where the maximum temperature found to be 4 degree all over through the wing which indicates the ice formation is more difficult for such designed model, when we place a thermo coil inside the wing layer.

II RESULTS AND DISCUSSIONS

Here we found the model designed in such a way that the model impinges float design contour curve profile design such that the ice formation is prevented in surface layer of the wing

This max value of torque is more than sufficient to break the ice formation in wing. Hence the profile derived designed can be approved for wing design for deicing. Here the model derived can be more sufficient for working condition wherethe pressure gradient developed at the end of the wing curve produces a reverseback pressure which induces more torque development in the wing, which produces additional force development in the wing against ice formation.

The forced vibration response can be described in terms of either set of modes. The two descriptions are shown to be equivalent only if the damping is independent of the mode number. The damping may, however, be an arbitrary function of the forcing frequency. This simulation has been carried out to predictthe increased drag value because of ice formation.

Here Found the temperature of the wing does not found to be under mixed conditioning, where the maximum temperature found to be 4 degree all over through the wing which indicates the ice formation is more difficult for such designed model, when we place a thermo coil inside the wing layer

III CONCLUSION

Three-dimensional unsteady thermodynamic simulation model is developed to describe the dynamic response of an aircraft wing anti-icing system. This computational fluid dynamics based model involves a complete wing segment including thermal anti-icing bay inside the leading edge. The unsteady, integrated external thermal flow simulation is presented with heat conductivity through the solid skin in a structured mesh. The calculated skin temperature results are satisfactory in their good match with flight test data. The presented research work indicates a strong potential of using computational fluid dynamics in dynamic wing anti-icing system model development and validation. The heating process is simulated at the beginning of this study by applying different basic functions presenting piccolo tube heat flux, and the wing skin responses are discussed. The 3d cfd model involves a complete wing segment with the piccolo type thermal anti-icing bay. In the unsteady integrated internal/external thermal flow simulation with heat conductivity through the solid skin, time dependent boundary condition specifications and proper time steps are investigated. The structured mesh generated increases the unsteady simulations efficiency. The calculated 3d skin temperature dynamic variation coincides with the flight measurements very well. It indicates the possibility of applying cfd simulation data to the anti- icing system development. It may be used in dynamic model tuning, to complement or even be used in place of the fight test data that are expensive and often not complete enough to serve this purpose.

REFERENCES

[1] R. MIKALSEN, A.P ROSKILLY, 2009 "PERFORMANCE SIMULATION OF A SPARK IGNITED FREE-PISTON ENGINE GENERATOR".

[2]R. MIKALSEN, A.P ROSKILLY, 2009 "THE DESIGN AND SIMULATION OF A TWO- STROKE FREE-PISTON COMPRESSION IGNITION ENGINE FOR ELECTRICAL POWER GENETATION".

[3]A.Z.M. FATHALLAH AND R.A. BAKAR, 2009 "DESIGN OPTIMIZATION OF SPRING AND THE EFFECTS ON SCAVENGING AND PRESSURE RATIO FOR TWO STROKE SINGLE CYLINDER SPARK IGNITION LINEAR ENGINE". A.Z.M. FATHALLAH AND R.A. BAKAR, 2010 "THE EFFECT OF SPRING DESIGN AS RETURN CYCLE OF TWO STROKE SPARK IGNITION LINEAR ENGINE ON THE COMBUSTION PROCESS AND PERFORMANCE".

[4]R.MIKALSEN, A.P ROSKILLY, 2009 "COUPLED DYNAMIC-MULTIDIMENSIONAL MODELLING OF FREE-PISTON ENGINE COMBUSTION".

[5]AHMAD AAMAL ARIFFIN, NIK ABDULLAH NIK MOHAMED AND SYARIZAL FONNA, 2006 " SIMULATION OF FREE PISTON LINEAR ENGINE MOTION WITH DIFFERENT INTAKE AND EXHAUST PORT POSITIONS"

[6]MARTIN EKENBERG AND BENGT JOHANSSON, 2008 "THE EFFECT OF TRANSFER PORT GEOMETRY ON SCAVENGING FLOW VELOCITIES AT HIGH ENGINE SPEED"

[7] M.M.NOOR, K.KADIRGAMA, DEVARAJAN R., M.S.M.SANI, M.F.M.NAWI, T.F.YUSAF,2008 "CFD SIMULATION AND VALIDATION OF THE IN-CYLINDER WITHIN A MOTORED TWO STROKE SI ENGINE"

[8]SEMIN, N.M.I.N. IBRAHIM, ROSLI A. BAKAR AND ABDUL R. ISMAIL, "IN- CYLINDER FLOW THROUGH PISTON-PORT ENGINES MODELING USING DYNAMIC MESH".

[9]AHMAD KAMAL ARIFFIN, NIK ABDULLAH NIK MOHAMED, "SIMULATION OF FREE PISTON LINEAR ENGINE MOTION WITH DIFFERENT INTAKE AND EXHAUST PORT POSITIONS".JOURNAL OF APPLIED SCIENCE RESEARCH, 4(1) PP 58-56.

[10] FRANCISCO BROJO, ANTONIO SANTOS, JORGE GREGORIO, 2010 "COMPUTATIONAL ANALYSIS OF THE SCAVENGING OF A TWO-STROKE OPPOSED PISTON DIESEL ENGINE".

