AEROSPACE ENGINEERING GRADUATE STUDENS SYMPOSIUM

(AEGSS-2015)

IIT Kanpur

INDEX

Schedule

Aerospace Engineering Graduate Students' Symposium AEGSS-2015

Venue & date

Outreach Auditorium

11th April 2015

Program synopsis

Registration – 08:15-09:00 AM

Number of technical sessions - 5

Number of presentations - 35

Organizing Committee

Dr. Sathesh Mariappan (Convener)

Dr. Mangal Kothari

Dr. Abhishek

Dr. Alakesh. C. Mandal

Dr. Debopam Das

Aerospace Engineering Graduate Students Symposium
AEGSS-2015

Inaugural Address

08:30-08:35	Welcome address by Convenor	Dr. Sathesh Mariappan
08:35-08:40	Address from the Head, Aerospace Engineering	Prof. Sanjay Mittal
08:40-08:55	Address from the Director	Prof. Indranil Manna
08:55-09:00	Vote of thanks	Dr. Abhishek

Technical Sessions

Session-I: Session chair – Prof. Sanjay Mittal

09:00-09:15	Anshul Khandelwal	An experimental investigation of flow over symmetrical as well as asymmetrical sinusoidal oscillating airfoil
09:15-09:30	Pratik Das	Numerical study of supersonic flow past a cylindrical after body
09:30-09:45	Prabhakran B	Leader follower strategy for quad rotor based on NDI tracking controller design
09:45-10:00	Srivastava.S,Khan.A.S	Experimental investigation and constitutive modelling of rubber-like materials
10:00-10:15	Srihari Dinesh Kumar.J	Active control of combustion noise in gas turbine combustor by fuel modulation
10:15-10:30	K.Chawla	Constraint effects on the fiber/matrix interface strength of a laminated composite
10:30-10:45	P.M.Bagade	Study of effects of free stream turbulence
10:45-11:00	S. Aditya Varma	3-D impact angle constrained missile engagements
11:00-11:15	Tea	

Session-II: Session Chair – Prof. C. S. Upadhyay

11:15-11:30	Eswar Sunder, Balamurugan Ganesan	Experiments on axisymmetric boundary layer at an elevated level of freestream turbulence
11:30-11:45	S.S.Singh	Measuring dynamic properties of epoxy film using laser induced stress waves
11:45-12:00	M.C.Keerthi	Effect of adjacent blade oscillation on the forces on a blade of a compressor cascade
12:00-12:15	Afzal	Modelling of spin stabilized artillery shells
12:15-12:30	Prasoon Suchandra	Direct numerical simulation of 2D compressible flow around pitching airfoil
12:30-12:45	Arpita Dash	Numerical investigation of turbulent lifted H2 flames in a vitiated co-flow using RANS-CMC approach
12:45-01:00	Joydeep Bhowmik, Saurav Kumar Ghosh	Wind tunnel testing and aerodynamic study of a large ornithopter
01:00-01:15	Akhil Mulloth	Spatio-temporal wave front – various types possible for Blasius profile and their characteristics
01:15-02:15	Lunch	Outreach lawn

Session-III: Session Chair – Prof. P. M. Mohite

02:15-02:30	Tulsi Ram Sahu,M.Furquan	Vortex induced vibration of a circular cylinder with attached flexible splitter plate
02:30-02:45	Shashank Sharma	Lattice Boltzmann simulations of coaxial shaped jet
02:45-03:00	Rajbir	Comparison of heat release rate measurement by chemiluminescence and two microphone technique
03:00-03:15	Vaibhav Dabaria	Linear stability analysis of measured inflectional velocity profiles in separated boundary layer flows
03:15-03:30	Tea	

Session-IV: Session Chair – Prof. A. K. Ghosh

03:30-03:45	Rahul Kumar Soni, Nitish Arya	Computation of supersonic flow past backward facing step in OpenFOAM
03:45-04:00	Raghavendra Prasad	Inverse simulation method for prediction of helicopter blade loads
04:00-04:15	Abhishek Dhiman	Experimental investigations of the effects of increased free-stream turbulence level on a pressure induced separation bubble
04:15-04:30	Harikant	Effect of global acceleration in the response function of premixed flame
04:30-04:45	Ajit Kumar	Information rich formation
04:45-05:00	Shabeeb	Control of laminar separation bubble over M-300 airfoil at low Reynolds numbers using passive blowing and suction
05:00-05:15	Ramana M V	A cooperative pursuit-evasion game of a high speed evader
05:15-05:30	Babu	Stiffness predictions of a 3D random chopped fiber reinforced composites using random quential adsorption
05:30-05:45	Tea	

Session-V: Session Chair – Prof. Debopam Das

05:45-06:00	Yesgat Admassu L	Filler shape and volume fraction effect on the fracture behavior of epoxy composites
06:00-06:15	Saravana Kumar L, Alok M	Numerical prediction of transitional flow over thick airfoil models
06:15-06:30	S.I.Haider	Reduced order modeling for flow past cylinder using the enstrophy-based POD
06:30-06:45	Naman Rawal	Conceptual design studies of a coaxial mono tiltrotor / tiltwing
06:45-07:00	U. Yasar Arafath	Experimental study on leading edge effects in bypass transition
07:00-07:15	Sudharsan.K,Dinesh Kumar.S.J	Numerical investigation of combustion acoustic instability in atmospheric can combustor
07:15-07:30	V. S. Caesar	An experimental investigation of a geometry-induced separation bubble
07:30-07:45	Varun Bhatt	Planform optimization of finite wing
07:45-07:50	Valedictory	Prof. Debopam Das
07:50-09:00	Dinner	Outreach lawn

An Experimental Investigation of Flow over Symmetrical as well as Asymmetrical Sinusoidal Oscillating Airfoil

Anshul Khandelwal, Kamal Poddar and Debopam Das

Department of Aerospace Engineering, IIT Kanpur 208016, India

The flow over a NACA 0012 airfoil oscillating in pitching motion has been studied using surface pressure measurements and phase-locked PIV measurements. The effect of Reynolds number, reduced frequency and asymmetry in pitch has been studied for different set of oscillation amplitudes and mean angles. The phenomenon of dynamic stall is observed with the formation of dynamic stall vortex which is seen to originate from laminar separation bubble at all frequencies, and it also induces a trailing edge vortex as it passes over the trailing edge. Reverse flow near leading edge region originate locally, and is not propagated from trailing edge. The effect of different parameters on pressure distribution, integrated coefficients and the flow field has been documented and the reasons discussed, both in the fully-developed and partially-developed dynamic stall regimes. In the fully-developed regime, increase in upstroke pitch rate causes a more energetic vortex to be formed, but in partially-developed regime, the strength of vortex decreases. Effect of Reynolds number on the sectional coefficients and overall flowfield is minor, but on the surface pressure is noteworthy. The mechanism of various events such as flow reversal, dynamic stall vortex formation, its eventual convection followed by complete flow separation and finally flow reattachment has been discussed.

Numerical Study of Supersonic Flow Past a Cylindrical After Body

Pratik Das and Ashok Dey

Department of Aerospace Engineering, IIT Kanpur 208016, India

Objects like bullets, projectiles, missiles and rockets traveling at supersonic velocity are subjected to massive pressure drag due to the low pressure region behind the base of these objects. As the boundary layer separates from the body at the trailing edge of these objects, a low pressure recirculation region is formed behind the object causing massive drag force to act upon the object. It has been observed that with increasing Mach number of free-stream flow the pressure in the recirculation reduces and under sudden

circumstances the drag caused by this pressure difference can contribute up to two third of the total drag acting on the body(Dutton et al. 1994). This massive amount of drag force acting on these objects during supersonic flight compromises the range of operation these objects. Primary concern being recovery of the base pressure, over the past years several active and passive techniques, such as boat-tailing(chamfering the corners of the trailing edge), base cavity(introducing a cylindrical cavity on base surface), base bleed(injecting subsonic jet into wake region) and base burning(slow burning charge at base) have been developed to increase the base pressure and reduce overall drag on the bodies travelling at supersonic speed. In past, these techniques have been employed in an empirical manner, without thorough understanding of the complicated fluid dynamic processes involved with them. In order to design optimal techniques for base pressure recovery it is imperative to achieve a thorough understating of the complex fluid dynamic processes that occur in the wake region and how these different techniques influence the flow physics in the wake region.

In this current study, the near wake of a cylindrical after body aligned with a uniform Mach 2.46 flow has been numerically investigated using Reynolds Averaged Navier-Stokes equations (k-epsilon two equation model) and Large Eddy Simulation (dynamic sub-grid scale eddy viscosity model). Mean flow field properties obtained from numerical simulations, such as axial velocity, pressure on base surface has been compared with the experimental results. It has been found that k-epsilon model fails to predict the flow properties in the recirculation region where better agreement has been observed between the data obtained from LES and measurements. Data obtained from LES has been further analyzed to investigate the turbulent flow field in the wake region. Parameters like turbulent kinetic energy and primary Reynold's stress have been calculated and compared with the results obtained from experiment in order to achieve a better understanding of the role of turbulence in the flow field.

Leader Follower strategy for quad rotor based on NDI tracking controller design

Prabhakran B and Mangal Kothari
Department of Aerospace Engineering, IIT Kanpur 208016, India

Multi-rotor platforms have gained a huge amount of interest recently - as a research platform, due to its VTOL capabilities, easy fabrication, as well as a high maneuverability. The flight performance of conventional (PID) linear controller is often very low compared to the nonlinear model based controllers. Nonlinear controllers are capable of achieving the stringent high performance maneuvers by exploiting the vehicle dynamics to the maximum extent. This work primarily focus on

the nonlinear controller design for the quad rotor with two loop structure, based on novel nonlinear dynamic inversion approach. Also it has been found that a slight modification in the control architecture of dynamic inversion approach resulted in improving the performance of the controller to a greater extent. Finally the results are compared with the linear PID control strategy to emphasis the advantages of modified nonlinear dynamic inversion approach. Also a redundant leader follower strategy based on modified NDI controller will be discussed in the last part of the talk.

Experimental Investigation and Constitutive Modelling of Rubber-like Materials

Srivastava, S.,, Khan, A.S., Kitey, R., Upadhyay, C.S.

Department of Aerospace Engineering, IIT Kanpur 208016, India

Rubber and rubber like materials are of prime importance in aerospace, civil and mechanical engineering as well as in the rapidly emerging biomedical field. The tire and tubes industry is the largest consumer of these materials. Due to the usage of flexible material for sealants in the aircraft and space craft windows, a great amount of attention is being given to understand the dynamic and thermo mechanical behavior of rubber materials. The characteristics of human soft tissues are quite similar to the mechanical behavior of rubber material.

Because of highly nonlinear elastic properties of rubbery material the modeling of its mechanical behavior has always been a big challenge. Several models are available for modeling rubber and rubber-like material but they are often limited to small strains and are valid only for specific deformations. Higher order models with more number of parameters show a good conformation with the experimental data but they impose great challenges in determining the required parameters. In this work a new model is proposed and its efficacy is demonstrated with quasi-static experimental aids under large strains.

An invariant based hyperelastic continuum model is formulated using appropriate Gibbs free energy function. The model is extended to include the viscous effects by considering two interacting networks, the state dependent and the rate dependent netwprks, contributing towards equilibrium and non- equilibrium energies, respectively. It is demonstrated that the model encapsulates Mullin's effect quite well and is capable of capturing material behavior under a generalized multiaxial loading conditions. The modeling parameters are determined by conducting uniaxial tension, compression and dynamic mechanical experiments. The effect of strain rate and the cyclic loading on the material's nonlinearity is investigated. The proposed hyperelastic model forms the basis for more complex elasto-plastic, viscoelastic and viscoplastic models. These models when used with techniques like finite element method provide a powerful tool in solving complex problems.

Key words: Viscoelastic, Rubber, Constitutive Modelling, Continuum, Hyperelasticity.

Active control of combustion noise in gas turbine combustor by fuel modulation

Srihari Dinesh Kumar. J, Sathesh Mariappan, and Abhijit Kushari
Department of Aerospace Engineering, IIT Kanpur 208016, India

The proposed study will be on the sources for the combustion noise occurring in swirl-stabilized spray combustor. Combustion noises is classified as direct and in-direct combustion noise. The present study is focused on the investigation of direct combustion noise. It occurs when the volume of the gas fluctuates due to the fluctuations in heat release, caused perhaps due to flow turbulence [1]. This results in sound waves which propagate outside the boundary of the flame. The radiated acoustic waves are reflected from the boundaries of the combustion chamber, perturbing the fuel flow rate and hence the spray characteristics. This eventually leads to perturbation in the heat release rate and thus a feedback loop is established. At certain conditions, if the heat release drive the acoustic oscillations as Rayleigh's criterion "when the heat is given to the air at the moment of greatest condensation, or be taken from it at the moment of greatest rarefaction, the vibration is encouraged" is satisfied [2]. These acoustic oscillations grow leading to discrete tonal sound and this phenomena is termed as combustion instability.

Experiments are performed in a scaled down swirl stabilized liquid fuelled gas turbine combustor, where a new scheme for open-loop control of combustion noise using periodic fuel injection is employed without drastically altering the combustor design or forfeiting its performance[3]. Fuel is modulated in the frequency range of 0.6 to 5 Hz with various duty cycles [25-75%] using square wave. Fuel modulation is achieved using a custom-made solenoid valve circuit. The modulated fuel enters the air-blast atomizer, through a turbine flow meter. Heat release measurements show dominant peak at low frequency oscillations, and in the occurrence of combustion noise, unsteady heat release rate is the driving source. Hence, these fuel flow modulations are sent at low frequency [4].

The main objective of this paper is to investigate the potential of active control to damp the combustion noise in laboratory scaled gas turbine combustor. Unsteady pressure transducer is used to capture the sound pressure level inside the combustor. A reduction in sound pressure level of 14dB is achieved by modulating fuel with 50% duty cycle at 1.5Hz.

REFERENCES

[1] W. C. Strahle, M. Muthukrishnan, and D. H. Nealej, “Coherence between Internal and External Noise Generated by a Gas Turbine Combustor,” *AIAA J.*, vol. Vol.15., no. No.7, pp. 1018–1024, 1977.

[2] Lord Rayleigh, “The explanation of certain acoustical phenomena,” *Nature*, vol.18, pp 319-322. 1878.

[3] S. Acharya and J. H. Uhm, “Characteristics and Control of Combustion Instabilities in a Swirl-Stabilized Spray Combustor,” *Combust. Process. Propulsion.* vol. 19, no. 3, pp. 157–168, 2005.

[4] P. A Hield and M. J. Brear, “Comparison of Open and Choked Premixed Combustor Exits During Thermoacoustic Limit Cycle,” *AIAA J.*, vol. 46, no. 2, pp. 517–526, 2008.

Constraint effects on the fiber/matrix interface strength of a laminated composite

K. Chawla and R. Kitey

Department of Aerospace Engineering, IIT Kanpur 208016, India

Failure in laminated composite is a complex phenomenon. In general multiple modes are involved in a failure process which could have been triggered by a very different failure mechanism. The micro-structural cracks and delaminations developed during failure initiation often remain hidden due to heterogeneity which ultimately leads to a catastrophic failure when the laminate is subjected to a threshold loading condition. Therefore, prior to design and fabrication of a laminate it is imperative to understand various failure mechanisms and get an insight of how a micro level flaw evolves into a discernible macro-scale failure.

Matrix cracking, fiber breakage and fiber/matrix debonding are some of the common micro scale failure mechanisms that develop in a laminate. Corresponding failure characteristics are inferred from the cohesive strength of the matrix, the ultimate strength of the fiber, and the strength of fiber/matrix interface, respectively. In current investigation our focus is to evaluate the constraint effects on the fiber/matrix interface strength. A single fiber pull-out test configuration is modified and three sets of experiments are conducted. First, the interface shear strength between an unconstrained glass fiber and the epoxy matrix is determined by conducting a single fiber pull-out test. Next, for investigating the effect of constraints due to the surrounding fibers a single fiber is pulled out from the fiber bundle. And finally the whole fiber bundle is tested by conducting the pull-out test. The fiber/matrix interface shear strength is found to be higher when the single fiber is constrained. However, the average interface strength is observed to decrease for the fiber bundle case. This is attributed to the fiber distribution, uneven load distribution in the surrounding fibers and the inter fiber spacing. A two dimensional finite element analysis is conducted to get better insight of the involved failure mechanisms.

Study of effects of free stream turbulence

P. M. Bagade and T. K. Sengupta
Department of Aerospace Engineering, IIT Kanpur 208016, India

In many fluid dynamical applications, the oncoming flow has fluctuations with varying intensities of turbulence. The source of these background noises may be acoustic, vortical or entropic in origin. This calls for accurate modeling of free stream turbulence (FST) to find its effects on flows. In the present work, effects of FST are studied in a square duct flow for Re = 1100 based on centerline velocity and the side of the square. DNS is performed via parallel computing using 128 nodes with high accuracy numerical schemes[1].

Free-stream turbulence modeling

It is noted that solving Navier-Stokes equation (NSE) in the presence of FST, moving average model[2,3] is more realistic than solving stochastic differential equations for flows[4,7]. In a recent DNS, flow in a square duct has been studied with harmonic fluctuations with the oncoming flow to simulate the effects of FST[6]. In the moving average model, from the histogram or probability distribution function (PDF) the central moments are obtained as

$$\mu_n = \frac{\sum [f(x_i - \mu]^n}{\sum f(x_i)} \tag{1}$$

where, *f(x)* represents the histogram and *n* indicates the corresponding moment about the mean, μ. The third moment was considered zero[2] assuming it to be negligibly small and a synthetic time series was generated at the inflow boundary using a standard first order moving average, with the help of second and fourth moments[2,3,7]. Hence, in this model

$$x_t = e_t + \alpha_{fst} e_{t-1} \tag{2}$$

where, e_t is defined by a normal distribution with first central moment as zero and standard deviation defined as σ. Using higher order statistics and approach in [2], the net streamwise disturbance component is obtained as

$$u' = e_t + \alpha_{fst} e_{t-1} + \sum_{j=1}^{N} a_j e^{ik_j(x-ct)} \tag{3}$$

where first two terms are given by the Gaussian distribution aforementioned and the last term shows the low frequency components of the disturbance that is facility- and speed-dependent; *c* being the

phase speed of propagation of these low wavenumber coherent structures. In the final presentation, the modeling of FST and its effects in a 3D channel flow will be reported.

References

1. Sengupta, T. K., High Accuracy Computing Methods, Cambridge University Press, New York, USA, 2013.
2. Sengupta, T. K., De, S., Gupta, K., Effect of free-stream turbulence on flow over aerofoil section at high incidence, J. Flu. Struct. , 15, 2001, 671-690.
3. Fuller, W. A., Introduction to Statistical Time Series, Cambridge Univ. Press, U.K., 1995.
4. Lucor, D., Karniadakis, G. E., Noisy Inflows Cause a Shedding Mode Switching in Flow Past an Oscillating Cylinder, Phys. Rev. Letters ", 92, 2004, 154501-1–54501-4.
5. Sengupta, D. Das, P. Mohanamuraly, V.K. Suman and A. Biswas, Modeling Free-Stream Turbulence based on Wind Tunnel and Flight Data for Instability Studies Int. J. Emerging Multidisciplinary Fluid Sci., 1, 3, 2009, 181-199.
6. Shinn, A. F., Vanka, S. P., Hwu, W. W., Direct numerical simulation of turbulent flow in square duct using a GPU, AIAA paper 2010-5029., 2010.
7. Sengupta, T. K., Instabilities of Flows and Transition to Turbulence, CRC Press, Taylor & Francis Publication, Florida, USA, 2012.
8. Schlichting, H., Boundary Layer Theory, McGraw Hill Publ.,7th edition, 1978.

3-D Impact Angle Constrained missile Engagements

Aditya Varma and Mangal Kothari

Department of Aerospace Engineering, IIT Kanpur 208016, India

This work considers the problem of surface to surface engagement which is impact angle constrained in both azimuth and elevation. This problem is approached by modifying and optimizing Pure Proportional Navigation guidance law. The modified guidance law is validated by simulations taking into consideration first order autopilot lag and maximum acceleration limit.

Experiments on Axisymmetric Boundary Layer at an elevated level of Freestream Turbulence

Eswar Sunder,Balamurugan Ganesan and Alakesh C. Mandal

Department of Aerospace Engineering, IIT Kanpur 208016, India

An experimental study on an axisymmetric boundary layer transition has been carried out. The axisymmetric boundary layer was generated using a circular cylinder with its axis parallel to the freestream. Boundary layer transition was initiated using high level of freestream turbulence generated by passive grid installed at the entrance of the test section. Two cylindrical models with different leading edge shapes were used in this experimental investigation to study the effect of the leading edge shape on axisymmetric boundary layer transition. Both the hotwire and particle image velocimetry (PIV) techniques have been used to characterize the response of the axisymmetric boundary layer to the elevated freestream turbulence. The flow visualization over the circular cylinder model has been carried by injecting kerosene smoke throw a 1 mm slot on the cylinder surface. The measured urms profile shows that the peak urms occurs at about y/d =0.4; here d is the boundary layer thickness. It is also found that the disturbance energy estimated as u2r ms,max/U2 0 grows linearly with the streamwise distance. We also find that the backward and forward-jet like structures of the fluctuating velocities, as obtained from PIV measurements, perhaps corresponds to low- and high-speed streak in the boundary layer, as the flow visualizations on the cylinder surface clearly reveal the existence of low- and high-speed streaks. These results are similar to the results generally seen in the case of bypass transition over a flat plate. To the best of our knowledge, these results are reported for the first time in the case of an axisymmetric boundary layer.

Measuring dynamic properties of epoxy film using laser induced stress waves

S. S. Singh and R. Kitey

Department of Aerospace Engineering, IIT Kanpur 208016, India

Adhesive bonded joints are extensively used in aerospace applications. Several aircraft components are smaller in sizes and/or have complex shapes which cannot be fastened to another structure by riveting, bolting or welding techniques. The reliability of these joints is inevitably governed by the material and interfacial failure properties of the adhesive layer. The failure behavior of multilayer components complicates when they are subjected to extreme dynamic loading conditions. Therefore, prior to design and fabrication of multilayer structures dynamic cohesive, adhesive and fracture characteristics of the adhesive materials should be determined.

While the mechanical behavior of materials under quasi-static loading condition is determined by conducting tension, compression and shear tests, the impact and split-hopkinson bar tests are employed in general to evaluate dynamic characteristics at a strain rate up to the order of 10^5/s. Although the test methods are well established for conducting experiments at macro scale, any such methodologies are failed to evolve for the cases when the material dimensions are constrained to micro and nano levels. Also, the conventional techniques can only provide average macroscopic behavior of material and the microscopic mechanisms, especially in regards to the failure, is largely obscure.

In current investigation we employ laser spallation technique to measure the cohesive and interfacial strengths of epoxy film by subjecting the material to a strain rate of 10^7/s. The failure is initiated by laser induced stress waves in an epoxy film of various thicknesses. Optical micrographs of the failed surfaces are analyzed to infer the failure mechanisms. For a sub optimal film thickness the epoxy film interface is observed to fail. On the contrary a cohesive failure in the material is noticed when the specimen was prepared with a thicker film. Quantitative measurements and computational analysis for understanding the failure mechanisms are under progress.

Effect of adjacent blade oscillation on the forces on a blade of a compressor cascade

M.C. Keerthi and Abhijit Kushari

Department of Aerospace Engineering, IIT Kanpur 208016, India

The current trend of aircraft engines is one towards attaining high isentropic efficiency while minimizing its weight. This leads to a state where the blades are highly loaded and consequently susceptible to vibrations. High cycle fatigue caused as a result of such self-excited flutter or forced vibration due to defects in the air stream are detrimental to the engine. An understanding of the onset of instabilities is essential to predict their occurrences to avoid a catastrophic failure during operation or costly redesign during the development phase. The critical parameters in turbomachine aeroelasticity are the reduced frequency and the inter-blade phase angle. The damping of the system is known to be a function of the phase difference between the blade forces and the blade motion. In the present study, a linear cascade of five blades is considered to understand the effect of harmonically varying boundary conditions. The second and fourth blades of the cascade are subjected to torsional oscillation by an external mechanism. The third blade, considered as the reference, is stationary and instrumented. The unsteady pressure along the reference blade surface is measured simultaneously with the loads acting on the blade. The unsteady pressures are measured using a multi-sensor pressure scanner by multiplexing and the loads are measured using a five-channel strain gage balance. The blade displacement is determined from the integrated accelerometer signal mounted on an oscillating blade. The experiments are conducted at a low-subsonic speed and multiple oscillation frequencies. The cascade is set at zero incidence and four stagger angles. The effect of inter-blade phase angle is included as the oscillation of the walls adjacent to the reference blade. The phase difference between the harmonic motion of the neighboring walls and the pressure and load signals on the reference blade is related to the damping characteristics of the reference blade. The variation in damping is studied for the range of blade motion phase difference angles and reduced frequencies. The effect of the phase difference between the oscillating blades is seen to strongly affect the damping characteristics of the reference airfoil.

Modelling of Spin Stabilized Artillery Shells

Afzal and Abhishek

Department of Aerospace Engineering, IIT Kanpur 208016, India

In this paper, a hybrid combination of ground and body frame model has been used to study the motion of a spin stabilized, conventional artillery projectile. The mathematical model has been used to simulate the flight of the Denel 155mm Assegai M2000 series artillery projectile under the influence of the aerodynamic and gravitational forces imparted by the standard atmospheric model of the earth on

the projectile. The Newtonian equations have been framed assuming the inertial frame to be attached to the center of the earth but not rotating with it. Further, the effect of constant crosswind and range wind on the final range, maximum height attained and lateral deviation has also been studied. The ordinary differential equations have been solved using the Runge–Kutta fourth order method. Another set of equations have further been developed in conjunction to the hybrid model so as to compute the final values of longitude and latitude of the point of impact of the projectile on the curved surface of the earth.

Direct numerical simulation of 2D compressible flow around pitching airfoil

Prasoon Suchandra ,T. K. Sengupta and Dr. Abhishek

Department of Aerospace Engineering, IIT Kanpur 208016, India

Two-dimensional compressible flow past a pitching NACA 0012 airfoil has been computed using direct numerical simulation in a finite difference framework, to study dynamic stall phenomenon on unsteady airfoils. The problem has been formulated in a body-fixed, rotating, non-inertial frame (grid). High accuracy optimized upwind compact schemes (OUCS) have been used for derivatives of convective terms in Navier-Stokes equations (NSE) and optimized third order Runge-Kutta (ORK3) method has been used for time integration. Artificial diffusion model due to Jameson, Schmidt and Turkel has been used. We have computed four cases corresponding to four different parametric combinations from dynamic stall experiments of McCroskey et al. Pressure-based load coefficients and Cp distributions over the airfoil have been calculated. Pressure contours and streamlines around the unsteady airfoil have also been studied in order to investigate vortex-shedding during dynamic stall. Our load calculations are compared with those of McCroskey et al. This is work in progress.

NUMERICAL INVESTIGATION OF TURBULENT LIFTED H2 FLAMES IN A VITIATED CO-FLOW USING RANS-CMC APPROACH

Arpita Dash, Santosh Ansumali and Ashoke De

Department of Aerospace Engineering, IIT Kanpur 208016, India

Mathematical modelling of the turbulent combustion process is becoming increasingly applied in calculations to assist in the design and analysis of practical combustion devices for efficiency-improvement and emission reduction. The necessity for accurate prediction of pollutant emissions in many applications has made it essential to link the turbulent flow calculations and finite-rate chemistry effects. Several methodologies are available for modeling such interactions, with conditional moment closure (CMC) being one such method.

This paper is concerned with investigations of role of turbulence parameter and conditional velocity model (mean, linear and gradient) on turbulent jet diffusion flame. Dependence of lift off height on co-flow temperature is also discussed. The flames issuing from Cabra burner is considered for hydrogen fuel. The first order Conditional Moment Closure (CMC) approach has been employed in this paper. In the present work, a pressure-based, fully elliptic, finite volume formulation with Adams-Bashforth, Predictor-Corrector algorithm in polar cylindrical coordinate system is employed to solve the flow-field with k-ε turbulence model. Fully elliptic CMC equations are solved with detailed chemistry to simulate lifted H2-N2 flame in vitiated co-flow.

The predicted flow and scalar fields are in good agreement with the experimental data. The comparison between the three turbulent parameter set showed that $C\varepsilon 1$ = 1.44 and $C\varepsilon 2$ = 1.92 case predicts the best result. Different conditional velocity models do not exhibit substantial differences in the radial predictions, while in case of lift-off height prediction linear model performs better than other two. The current work correctly predicts the decreasing trend of the lift-off height with increasing co-flow temperature, as observed experimentally.

Wind tunnel testing and aerodynamic study of a large ornithocopter

Joydeep Bhowmik, Saurav Kumar Ghosh and Debopam Das

Department of Aerospace Engineering, IIT Kanpur 208016, India

An experimental work of an ornithopter tethered in a large wind tunnel is presented in this paper. A quasi steady theoretical model have also been developed which estimates the lift and thrust of a flapping wing for a given flying condition and the results are compared with the experiments. The tests have been carried out at the NWTF of IIT Kanpur. The model ornithopter has a wingspan of 1 m, weighs 300 grams. A 3 axis dynamic load cell (ATI mini 40 ™) with a NI DAQ is used to acquire the force data at 1Khz sampling rate. Additionally, a potentiometer is attached to the wing to measure the wing position and the data is acquired using a Aurduino due™board. This setup enables to study the generated forces at every instant of the wing position.

[1] *Keywords*—flapping, flexible wing, force measurement, rigid wing

Figure 1 Back view of the model inside the wind tunnel

In the experimental force data, it can be seen that the actual force is slightly delayed. In a typical Lift generation vs time curve, as the wing starts from the top most position (34 degrees dyhedral), an increase in lift is observed. An additional peak followed by a sustained lift is observed till the end of the stroke reversal (-13 degrees anhedral). Followed by this is a sharp negative peak which occurs when the wing begins the upstroke. The net lift is observed to increase with inclination whereas thrust reduces. Overall trend of the forces remains similar.

1

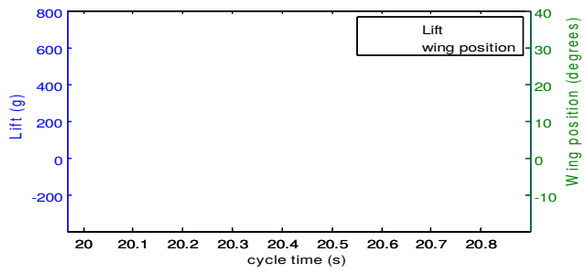

Figure 1 Lift and thrust (filtered) obtained at a flapping frequency 4.5Hz at 6m/s at an inclination of the flapping plane of 2 degrees with the freestream velocity.

The theoretical model used in this study is derived from Bhowmik et al[2]. The effect of wing position and articulation has been implemented. The downstroke and upstroke motion of the wing is separately used to calculate the lift and thrust generated. The wing inclination effects results from the components of the resultant aerodynamic forces as indicated in Phillips et al[3].

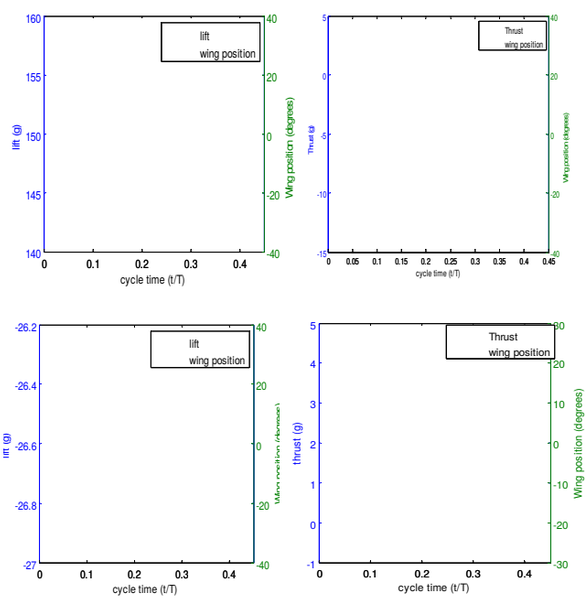

Figure 1 Lift and thrust estimated using quasi-steady lifting line theory for flapping frequency 4.5Hz at 6m/s at an inclination of the flapping plane of 2 degrees with the freestream velocity.

The quasisteady lifting line theory [2] gives a reasonably good estimate of the average lift and thrust over a flapping cycle and is thus useful to design the gearbox and choose appropriate power plant for building a practical flapping wing vehicle. Since it is a quasisteady theory, the lift and thrust generation occurs instantaneously with the wing movement as indicated in Figure 1 and is unable to capture the dynamic effects such as delay in peak lift with respect to wing position.

A different aerodynamic model can be used to model this unsteady phenomena using modified Theodorson'stheory [4] and [1]. But this model is again limited to only harmonic oscillations in pitch

and flap and does not take into effect the twist of the wing which plays a very important aspect in efficient production of lift and thrust in a flapping cycle. This theory predicts the nature of the lift and thrust as shown in Figure 1.

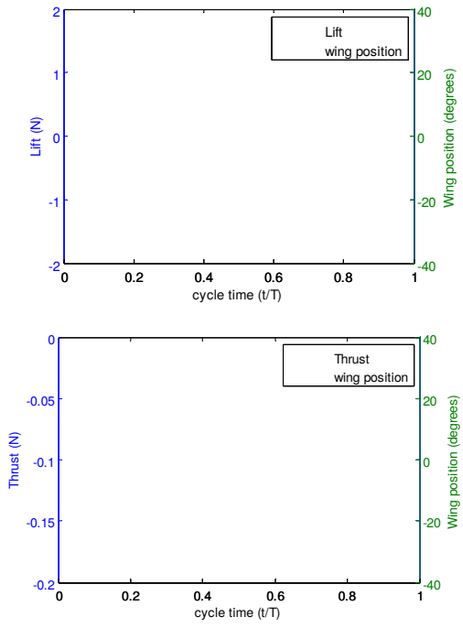

Figure 1 Lift and thrust estimated using [1] for flapping frequency 5Hz at 8m/s at an inclination of the flapping plane of 2 degrees with the freestream velocity.

Acknowledgement

Special thanks to Nidhish Raj for the Aurduino coding and NWTF crew for their support

References

[1] DeLaurier, J. D. (1993). An aerodynamic model for flapping-wing flight. Aeronaut. J. 97, 125–130.

[2] JoydeepBhowmik, Debopam Das, Saurav Kumar Ghosh, (2013) "Aerodynamic modelling of flapping flight using lifting line theory", International Journal of Intelligent Unmanned Systems, Vol. 1 Iss: 1, pp.36 – 61

[3] Phillips, P. J., East, R. A. and Pratt, N. H. (1981). An unsteady lifting line theory of flapping wings with application to the forward flight of birds. J. Fluid Mech. 112, 97–112.

[4] R. T. Jones , Wing Flapping with Minimum Energy, NASA Technical Memorandum 81 174, January 1980

Spatio-temporal wave front – various types possible for Blasius profile and their characteristics.

Akhil Mulloth and Rakesh Mathpal

Department of Aerospace Engineering, IIT Kanpur 208016, India

The presentation will be on the characteristics and types of spatio-temporal wave fronts (STWF) possible for Blasius profile and the basic requirements to capture STWF computationally. Initially reported in Sengupta and Bhaumik [Phys. Rev. Lett., **107**, 154501 (2011)],Bhaumik and Sengupta [Phys. Rev. E., **89**, 043018 (2014)] from the solution of 2D and 3D Navier-Stokes equation (NSE).It is equally important to note that STWF is also supported by linearized NSE with parallel flow approximation, as noted in the solution of Orr-Sommerfeld equation (OSE) in Sengupta, Rao and Venkatasubbaiah [Phys. Rev. Lett., **96**, 224504 (2006)]. In reporting the solution of NSE in the above references, high accuracy methods have been used, which are also known as dispersion relation preservation (DRP) methods. Earlier solutions of NSE in the literature, Fasel [J. Fluid Mech., **78**, 355-383 (1976)] using second order implicit finite difference method did not report any STWF. Even when the authors Fasel, Konzelmann [J. Fluid Mech., **221**, 311-347 (1990)] increased the spatial accuracy to fourth order, STWF was not found. The present study will explain why these references failed and the basic requirements to capture STWF computationally and characterize it.

By varying the frequency and location of the excitation various classes of STWF can be observed in a Blasius boundary layer. The presentation will illustrate these classes and their properties. The importance of including the leading edge of the flat plate while computing the Blasius profile will be illustrated. The presentation will also highlight that the use of an implicit method for time integration is inferior to explicit method for transitional flow, as has been shown in Sengupta *et. al.* [J. Sci. Comput, DOI: 10.1007/s10915-014-9967-1, (2014)].

Vortex induced vibration of a circular cylinder with attached flexible splitter plate

Tulsi Ram Sahu , M.Furquan and Sanjay Mittal

Department of Aerospace Engineering, IIT Kanpur 208016, India

Vortex induced vibration of a circular cylinder with attached flexible splitter plate in laminar flow is studied. The cylinder and plate are additionally mounted on an elastic support modeled by a spring. Both the spring and the flexible plate can deform under the action of fluid forces. A partitioned approach is used to model the interaction between the plate and the flow. The fluid structure interaction is studied for low non-dimensional mass and by systematically varying the reduced velocity and the flexibility of the plate. The Reynolds number of flow is 150. It is discovered that cylinder oscillations can be suppressed by suitable choosing a splitter plate for low reduced speed. However, galloping instability leads to large amplitude oscillations at large values of reduced speed.

Lattice Boltzmann Simulations of coaxial shaped jet flows

Shashank Sharma,Santosh Ansumali and Ashoke De

Department of Aerospace Engineering, IIT Kanpur 208016, India

The coaxial jets, based on a consideration of the velocity ratio λ, represented by Uo/Ui, Uo being the maximum velocity in the outer and Ui being the maximum velocity in the core or inner jet, fall into two main categories: those with velocity ratios not greater than unity ($\lambda<1$) and those greater than it ($\lambda>1$), both of which have been investigated by a number of workers, with majority of the work carried out in the higher Reynolds number regime ($Re>3000$) to capture the turbulent characteristics of the jets. Skampavia, et al. (1995) analyzed a coaxial mixer for use to cytometry at very low Reynolds numbers and obtained results for mixing efficiencies, absorbance and time for complete mixing, the measure of mixing being the uniformity of the product. Kriaa, et al. (2003) studied the influence of nozzle geometry and different velocity ratios on the characteristics of the jet downstream, with the observations that higher nozzle area ratios increasing the acceleration of the fluid and also the on training velocity. The velocity ratio increase also has a similar effect on the jet characteristics. M Ben-Yeoshua in his master's thesis in 1993 studied the effect of swirl on coaxial jets with velocities being in the Reynolds number independent zone and observed the axial and swirl velocity evolution downstream of the jet. Also the studies of axial velocity profiles with Reynolds numbers were carried out by Shaker with significant mixing and evolution differences with different Reynolds numbers.

There have been numerous studies on shaped jets aimed at observation of mixing characteristics of jets that depart significantly from round shape at low Reynolds numbers. Miller et al.(1994) presented a

work on non-circular jets with the shapes restricted to triangles, squares and ellipses of different aspect ratios. The evolution of shape downstream as well as centerline and cross stream velocities show interesting results of formation of shoulders on the minor axis, caused by entrainment of the fluid by the cross stream velocity. Mi, Nathan and Luxton (2000) carried out experiments on nine shaped jets and obtained the centerline velocity decay with axial distance with much difference between the velocities of different shapes, indicating a dependence of velocity decay on the shape of the nozzle. The difference between the centerline velocities at axial locations also suggests interaction of jet flow cross stream velocity with entrained fluid is different for different shapes.

Although work is available for both the conditions separately, the work on interaction of shaped jets with annular jet is scarcely available. Xu and Pollard (2001) carried out studies on square annular jets at laminar conditions and presented the evolution of centerline velocity and vorticity with details focused around the corners. In the present work we try to bring out the interaction of shaped jets in annulus of the other shape jets. As the mixing characteristics of annular jets are much different from single jets and shaped jets are very different from round jets, the characteristics of the flow are expected to be very different from round single jets.

The development of the entropy based Lattice Boltzmann Method (ELBM) was done in order to simulate the coaxial shaped jet flow. The double relaxation time in the LBM code was utilized to simulate the flow at higher Reynolds numbers. The KIDA-PELZ flow was validated for the working of the code. Also, validations of Reynolds number 100 and 437 were done to check the correctness of the code. The grid independence test will be done with the nozzle length of 5 jetdiameters and domain size of 40 jetdiameters X 8 jetdiameters X 8 jetdiameters for jetdiameters varying from 64 to 128 and Reynolds numbers 100 and 437. Additionally, Reynolds number 100 grid independence test will be done with jetdiameter of 30. Before moving further, simulations of square jet and triangular jet would be done so as to study the mixing in the two flows separately

Effect of Global Acceleration in the Response Function of Premixed Flame

Harikant and Sathesh Mariappan

Department of Aerospace Engineering, IIT Kanpur 208016, India

The aim of this work is to analysis experimentally the effect of global acceleration in the response function of premixed flame. In this experiment, the response function is calculated for two cases. In the first case, response function is obtained in the absence of acoustic duct. While in second case, response function is obtained in the presence of acoustic duct. In this study, the air flow rate

varies from 5 lpm to 10 lpm and fuel flow rate varies from 0.1 standard liters per minute to 0.3 standard liters per minute this corresponds to the a variation of the equivalence (Φ) ratio from 0.5 to 0.9.Global acceleration is defined as additional term in momentum equation, governing the dynamics of heat source [1]. This global acceleration term occurs due to the acoustic field in a duct.

In this experiment, four loud speakers are used to excite the flame at a particular frequency. In the first case, The response function is defined as R (ω) = $G(\omega)/F(\omega)$. While in the second case, the response function R' (ω) = $G(\omega)/F(\omega)[3]$. Here we defined terms, G (ω) = $q'/\acute{q}$ and F (ω) = $u'/\acute{u}$. Here (q') is the amplitude of fluctuating heat release rate and $\acute{q}$ is the amplitude of mean heat release rate, u' is the amplitude of fluctuating velocity and $\acute{u}$ is the amplitude of mean velocity measured upstream of flame.

In premixed combustion process heat release rate is proportional to the flame chemiluminescence intensity. The intensity (I) of CH* chemiluminescence is often used as a measure for the heat release rate in the premixed flame [2]. The intensity of CH* radicals is captured by photo multiple tube technique (PMT) with a band pass filter having a wavelength is 430 nm $\pm 10\,nm$. The ratio of fluctuating heat release rate to mean heat release rate is equal to the ratio of the amplitude of voltage fluctuating to amplitude of mean voltage. The equation is

$$q'/\acute{q}=v'/\acute{v}$$

The velocity fluctuation u'(ω) is measured by two microphone technique and the mean velocity $\acute{u}$ is measured using rot meter both cases. The formula used for the measurement of velocity fluctuation by two micro phones technique is

$$\text{¿}\,u'(\omega)=\text{¿}\,\frac{1/\square}{\Delta x*\acute{\rho}*\omega}$$

Here Δx is distance between two microphones, $\acute{\rho}$ is the mean density of air fuel mixer and $P_2 \wedge P_1$ are acoustic pressures amplitude from two microphones.

References

1. **Mariappan S., and R. I. Sujith, (2011).** Modelling nonlinear thermosacoustic instability in an electrically heated Rijke tube. J. Fluid Mech. 680,511-533
2. **Schuller, T., D. Durox, and S. Candel, (2003).** Self-induced combustion oscillations of laminar premixed flame stabilized on annular burners. Combustion and Flame, 135(4), 525-537.
3. **Preetham, S. H. and T. Lieuwen (2008).** Dynamics of laminar premixed flames forced by harmonic velocity disturbances. Journal of propulsion and power, 24(6), 1390-1402.

Linear stability analysis of measured inflectional velocity profiles in separated boundary layer flows

Vaibhav Dabaria and A. C. Mandal

Department of Aerospace Engineering, IIT Kanpur 208016, India

A linear instability analysis of separated flows over a flat plate has been carried out based on the measured velocity profiles. Since the measured data often show inflection velocity profiles, both the Orr-Sommerfeld and the Rayleigh equations were numerically solved using spectral method with spatial stability approach for investigating their local stability characteristics. The spectral codes for solving these equations were developed in house. The Orr-Sommerfeld code was validated using the published data in the literature for Blasius boundary layer. Similarly, the Rayleigh code was validated using the published data for the hyperbolic tangent velocity profile. Based on the linear stability analysis of the measured velocity profiles in a pressure driven separation bubble, Diwan & Ramesh (J. Fluid Mech. Vol. 629, 2009) proposed a new scaling relation for the most amplified frequency based on the height of the inflection point, and the vorticity thickness. In this work, we have verified their proposal based on a different curve fitting on the measured velocity data obtained from particle image velocimetry (PIV) measurements. We then applied the scaling relation for a geometry induced separation bubble and for a pressure induced separation bubble forced with elevated free-stream turbulence. It is found that the scaling relation holds good for both the cases with a small shift from their proposed line in the case of geometry induced separation bubble.

Computation of Supersonic Flow Past Backward Facing Step in OpenFOAM

Rahul Kumar Soni, Nitish Arya and Ashoke De

Department of Aerospace Engineering, IIT Kanpur 208016, India

The understanding of flow field associated with Scramjet engine is crucial in the efficient design of high speed propulsion system. The flow physics is often complex and experimental study with prototypes are quite expensive. This immediately poses the need for robust and customizable

solver that could be utilized to investigate detailed flow structure. In the present work, systematic validation of density based solver (rhoCentralFoam) available in OpenFOAM, for supersonic flow, is carried out. The test case investigated here is backward-facing step at Mach 2, reasonable agreement was observed in numerical and experimental results.

1. Introduction

Scramjet engines are found efficient air-breathing propulsion system in high speed flow regime. However, due to intrinsic difficulties associated with the combustion mechanism, there exists need for development in the area of fuel mixing and flame holding. At higher supersonic speed, due to very small residence time (O~ms), achievement of efficient fuel mixing and flame holding is of paramount importance. In recent years backward-facing step being conventional geometric configuration to establish subsonic recirculation zone, has been studied widely by many researchers [1-3]. They also found out that low recirculation zone established behind the step prolongs the residence time. Huang et al [3] put forward that vortices generated at the step corner enhance the fuel and air mixing which is reflected in the improved combustion and mixing efficiency.

The objective of the present work is to evaluate the capability of OpenFOAM framework, an open source computational fluid dynamics class library based on C++ [4], for high speed computation. The density based solver, that utilizes Kurganov and Tadmor schemes is chosen and modified as per our needs to simulate the flow physics over backward-facing step at Mach 2. The test case investigated here represents the experimental investigation of McDaniel et al [2].

2. Formulation

Numerical results were obtained by employing density based solver (rhoCentralFoam) available in OpenFOAM. Compressible, unsteady mass averaged, Reynolds-averaged Navier-Stokes equation are solved with cell-centered finite volume scheme [5]. Turbulence is represented through one equation Spalart-Allmaras (SA) and two equation models like, k-ε, RNG k-ε and SST k-ω. A comparative study of these turbulence models is also performed as part of current work. Apart for RANS calculation, Large Eddy simulation (LES) methodology where the dynamic Smagorinsky model is used for SGS modeling. The molecular dynamic viscosity was evaluated by Sutherland's law, turbulent and molecular Prandlt number is 1 and 0.7 respectively. The convection terms are discretized using monotone preserving schemes and diffusion terms are discretized using central difference scheme, temporal discretization is obtained through second order backward scheme.

At inlet boundary uniform flow properties, i.e. Mach number (M_∞= 3), free stream velocity (V=520 m/s), static pressure (P_∞ = 35 KPa) and static temperature (T_∞ = 167 K) are specified. Adiabatic, no-slip boundary conditions were enforced at the top and bottom wall along with condition that noraml pressure gradient vanishes at wall

3. Result

The numerical results computed through various RANS models and dynamic LES are validated against the experimental results. The static pressure profiles at x/h = 6.66, is compared with experimental data, presented in Figure 1. It is observed that numerically predicted results match well with experimental data at both the locations for almost all the turbulence models, except in the region of y/h < 1, which is consistent with the observations of Huang et al. [3].

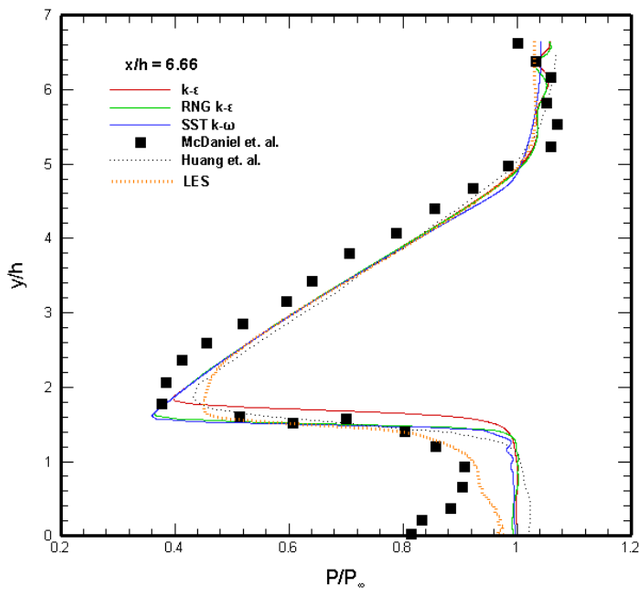

Figure 1: Comparison of numerical results with experimental observations at x/h = 6.66

4. Conclusion

Systematic validation of OpenFOAM for supersonic flow has been successfully attempted. Flow at Mach 2 over backward-facing step was studied and validated. Reasonable accuracy was observed, except for pressure profile in the region y/h < 1, due poor performance of most of the RANS based turbulence models in the separation region. The LES result however, has greater accuracy in near wall region with slight over prediction. Overall OpenFOAM, appear to be competitive computational fluid dynamics tool for the investigation of supersonic flow physic

5. References

[1] Takahashi, S., Yamano, G., Wakai, K., Tsue, M., &Kono, M., (2000). Self-ignition and transition to flame-holding in a rectangular scramjet combustor with a backward step. Proc. Combust Inst., 28, 705–712.

[2] McDaniel, J. C., Fletcher, D, G., &Hartfield, R.J., Jr., (1992). Staged transverse injection into Mach 2 flow behind a rearward-facing-step: a 3D compressible flow test case for hypersonic combustor CFD validation: AIAA Paper, 1992-0827.

[3] Huang, W., Pourkashanian, M., Ma, L., Ingham, D. B., Luo, S. B., & Wang, Z. W., (2011). Investigation on the flameholding mechanisms in supersonic flows: backward-facing step and cavity flameholder. J. Visualization, 14, 63–74.

[4] Weller, H. G., Tabor, G., Jasak, H., and Fureby, C., "A tensorial approach to computational continuum mechanics using object-oriented techniques," Comput. Physics, Vol. 12, No. 6, 1998, pp.620–631.

[5] Christopher JG, Henry GW, Luca G, Reese JM. Implementation of semi-discrete, non-staggered central schemes in a collocated, polyhedral, finite volume framework, for high-speed viscous flows. Int. J. Numer. Methods Fluid 2010; 63(1):1–21.

Inverse Simulation Method for Prediction of Helicopter Blade Loads

Raghavendra Prasad and Abhishek

Department of Aerospace Engineering, IIT Kanpur 208016, India

Predicting time histories of control angle and vehicle attitude would be very useful in main rotor and control system sizing for Helicopter. Control time history can be derived from flight tests but the associated cost and risk necessitate the development of a computational alternative. The development of an algorithm for inverse flight dynamic simulation for rotorcraft application would be discussed in this work. Control time history for various manoeuvres would be predicted and validated for Westland Lynx Helicopter. Further the developed algorithm would be implemented for simulation of 2.1g pull-up manoeuvre for UH-60A helicopter and obtained control time history would be used to predict rotor blade loads.

Keywords: Inverse simulation, Helicopter, Controls, Flight dynamics

Experimental investigations of the effects of increased free-stream turbulence level on a pressure induced separation bubble

Abhishek Dhiman and A.C. Mandal

Department of Aerospace Engineering, IIT Kanpur 208016, India

The effects of increased free-stream turbulence (FST) have been examined on a pressure induced separation bubble at low Reynolds number. The bubble was generated on a flat plate mounted horizontally at the mid-section of a wind tunnel test section by imposing adverse pressure gradient. A contoured ceiling above the flat plate was used to generate adverse pressure gradient in the boundary layer. The FST level was increased using a grid placed upstream of the plate and at the entrance of the test section. Both the hotwire anemometry and the particle image velocimetry (PIV) techniques were used for data acquisition. PIV measurements in the separation bubble indicate that the bubble size reduces at an increased freestream turbulence level. The hotwire measurements at the shear layer reveal that the shedding frequency of the bubble reduces in the case of high freestream turbulence. Proper orthogonal decomposition (POD) of the PIV data has also been carried out to get a better insight of the flow structures with and without freestream turbulence.

Comparison of Heat Release Rate Measurement By Chemiluminescence and Two Microphone Technique

Rajbir and Sathesh Mariappan

Department of Aerospace Engineering, IIT Kanpur 208016, India

In combustion reaction zone some electronically excited species are formed such as CH*, OH* etc. During de-excitation these radicals emit electromagnetic radiation of certain wavelength. This process is called chemiluminescence. It is used to measure heat release rate from premixed flames. This technique is well established but has certain limitations. In lean mixtures whole fuel is burnt due

to sufficient air, whereas in fuel rich condition some fuel remains unreacted. The heated carbon emits black body radiation which also contribute to heat release rate measured by photo multiplier tube (PMT).So chemiluminescence technique is not valid for fuel rich conditions. Furthermore radiation from one radical can be corrupted by, radiation from other radicals like HCO and radiation from black body radiation sources such as, combustor walls etc. [1]. Moreover this technique is valid only for premixed flames. It cannot be used for diffusion flames and partially premixed flames. Chemiluminescence occurs only in the presence of flame. Therefore one cannot measure heat release rate by chemiluminescence technique for heat sources where there is no flame, for example electrically heated wire [2]. These limitations can be overcome by using two microphone technique. In this technique we relate velocity jump across heat source to measure heat release rate. We use four microphones in total. Two microphones we put upstream of flame and two in downstream of flame.

References:

1-Venkata Nori[1] and Jerry Seitzman[2] "Evaluation of Chemiluminescence as a Combustion Diagnostic under Varying Operating condition" 46th AIAA Aerospace Sciences Meeting and Exhibit 7 - 10 January 2008

2-Sathesh Mariappan and R. I. Sujith " Experimental Investigation of Non-normality of Thermoacoustic Interaction in an Electrically Heated Rijke Tube"accepted for publication in the international journal of Spray & Combustion Dynamics.

Information Rich Formation

Ajit Kumar and Mangal Kothari

Department of Aerospace Engineering, IIT Kanpur 208016, India

This work considers the problem of maximizing, the information in tracking a stationary target using multiple Unmanned Aerial Vehicles (UAVs). Using Fisher-information theory, the condition to maximize information is derived, which requires to maintain a bi-circular formation around a target. Next, using the idea of cyclic pursuit, a formation controller is developed to keep UAVs in the desired formation. The efficacy of the proposed algorithm is shown through simulations. It is also shown by keeping the desired formation, the information is maximized.

Control of Laminar Separation Bubble over M-300 Airfoil at Low Reynolds numbers Using Passive Blowing and Suction

Shabeeb and K. Poddar

Department of Aerospace Engineering, IIT Kanpur 208016, India

An experimental study is done to examine the laminar separation bubble on M – 300 Airfoil. Wind tunnel tests were conducted at chord Reynolds numbers 10^5, $2x10^5$, $3\ x10^5$ and $4\ x10^5$ with angle of attack ranging from -10^0 to 30^0. Pressure measurement is done on the surface and in the wake of the airfoil and coefficient of lift, drag and moment is calculated. A passive flow control technique is developed by interconnecting the pressure ports on the airfoil based on the bubble location at different angle of attack. The coefficient of lift and stall angle has increased and the coefficient of drag has reduced after implementing this technique.

A Cooperative Pursuit-Evasion Game of a High Speed Evader

Ramana M V and Mangal Kothari

Department of Aerospace Engineering, IIT Kanpur 208016, India

This work studies a pursuit-evasion game of multiple pursuers and a high speed evader for holonomic systems. A group of pursuers uses the idea of perfectly encircled formation to capture an evader. The perfectly encircled formation is a formation in which an evader does not have an instantaneous escape path and it is defined using the concept of Apollonius circle. The work initially presents the conditions required to create a perfectly encircled formation. It is then argued that if a perfectly encircled formation shrinks over time by maintaining the connectivity between the

Apollonius circles, then capture is guaranteed. This work shows that it is not possible to maintain the connectivity while shrinking with minimum number of pursuers. Hence in this case, the capture cannot be guaranteed. An escape strategy is suggested that enables an evader to escape from the perfectly encircled formation. Two special cases of three and four pursues are also analyzed. The proposed escape strategy is evaluated through numerical simulations and future directions are discussed.

Stiffness Predictions of a 3D Random Chopped Fiber Reinforced Composites using Random Sequential Adsorption

Babu, P. M. Mohite and C.S. Upadhyay

Department of Aerospace Engineering, IIT Kanpur 208016, India

Fiber reinforced composites have found numerous applications in aerospace industry for their high specific strength and specific stiffness. However, the cost of traditional composite materials is also considerable.Random chopped fiber reinforced composites have emerged as alternative materials for lightweight structures due to their low cost and mass production capabilities. Nevertheless, understanding and modelling of their mechanical properties are still in research, yet to be exhausted.

The overall elastic properties of random fiber reinforced composites were obtained using a Mathematical homogenization procedure based on micromechanics of a RVE. A Random Sequential Algorithm has been developed to generate the complex geometry of a random fiber composite. The elastic material properties of the composite were obtained using FEM for continuum micro-mechanical analysis.

Filler shape and volume fraction effect on the fracture behavior of epoxy composites

Yesgat Admassu L. and R. Kitey

Department of Aerospace Engineering, IIT Kanpur 208016, India

The effect of filler shape and volume fraction on the fracture behaviour of epoxy composite is investigated. The composites are prepared by reinforcing glass spheres and milled glass fibers into a DGEBA/TETA (Diglycidy Ether of Bisphenol A / Tri ethylene tetra amine) epoxy system. Total five filler volume fractions, ranging from 0 to 10% at a step of 2.5%, are considered in this study. The spherical glass particles are of 34 μm average diameters whereas the cylindrical fiber strands have average dimensions of 16 μm diameter and 200 μm length. A process is developed to uniformly

disperse the fillers into the matrix material. The filler distribution uniformity is confirmed by optically imaging the polished test samples. The elastic and fracture properties are determined by conducting experiments at quasi-static loading conditions. The density, elastic modulus and fracture toughness of composites monotonically increase with increasing filler volume fraction. For the same volume fraction the fracture toughness value of milled fiber composite is consistently higher compared to the spherical particle case. Optical and finite element analysis is performed to study the toughening mechanisms involved in the fracture process. Crack bridging is evident from the fracture surface micrographs of short fiber composites. A two dimensional finite element study indicates significant crack-tip shielding and crack-leg bridging due to the secondary phase fillers, especially when the filler is located in the vicinity of the crack-tip. It is demonstrated that the skewed orientation of milled fibers with respect to crack orientation develop significant mode mixities even if the loading is symmetric. Better fracture behaviour of milled fiber composites is attributed to the non-symmetric bridging which increases the effective fracture toughness.

NUMERICAL PREDICTION OF TRANSITIONAL FLOW OVER THICK AIRFOILS MODELS

Saravana Kumar L, Alok M , Ashok De

Department of Aerospace Engineering, IIT Kanpur 208016, India

A numerical study has been performed over thick airfoils to investigate the prediction capabilities of newly developed transition model, based on laminar kinetic energy called K-KL-Omega through open source code (OpenFOAM). Two different airfoils NACA 0021 & NACA 65-021 are considered for this study and two dimensional (2D) unsteady simulations were performed at Re 120,000 for a range of angle of attacks. The performance of this model is assessed through aerodynamic lift, drag and pressure coefficients. The simulated results are also compared with the experimental measurements and XFOIL results as well. In this present study, it is found that the considered transition model is capable of predicting the complex laminar to turbulent transition phenomena to some extent. And for afore mentioned airfoils, at low angle of attacks the results are well matched with experiments and some discrepancies are found at higher angles. Nevertheless, the overall prediction characteristics are found satisfactory.

Keywords: Laminar Separation Bubble, K-KL-Omega, OpenFOAM, URANS. Transition flow, Thick Airfoils.

Reduced order modeling for flow past cylinder using the Enstrophy-based POD

S. I. Haider and T.K. Sengupta

Department of Aerospace Engineering, IIT Kanpur 208016, India

A reduced order model (ROM) is developed using Enstrophy based proper Orthogonal decomposition (POD) of DNS data performed for flow past a circular cylinder. Presence of multiple instability modes for the flow field is accounted for by eigenfunction approach using the Stuart-Landau Eckhaus (SLE) equation.The main aim here is to replace the solution of Navier-Stokes equation (PDE) by pairs of ODE for the amplitude and phase of the instability modes of vorticity. This reconstruction requires obtaining optimal initial conditions for the instability modes. These optimal initial conditions for the modes has been obtained for a set of Reynolds numbers .The present development is based on a dynamical system approach [T. K. Sengupta, N. Singh and V. K. Suman, J. Fluid Mech.,656,82–115, 2010] of flow past a circular cylinder.

Conceptual Design Studies of a Coaxial Mono Tiltrotor / Tiltwing

Naman Rawal and Abhishek
Department of Aerospace Engineering, IIT Kanpur 208016, India

Conventional helicopters suffer a major disadvantage over fixed wing aircrafts in forward flight. While providing the capability to vertically take off and land, helicopters are slower than their fixed wing counterparts. This also affects the range of such vehicles adversely. Keeping these drawbacks in mind, a mono coaxial tilt-rotor/tilt-wing design is proposed and analyzed in this paper. A cartoon of the design is shown in Fig. 1. The design is characterized by a coaxial prop-rotor system that is capable of converting from a lifter to a propulsor between vertical and forward flight. Half of the wing is also tilted along with the rotor, while the remaining out-board half remains in horizontal position at all times. It should be noted that the proposed design is different from the coaxial mono tilt-rotor concept with folding wings studies by

Preator et al. [1,2] as the present design has simpler blade tilting instead of mechanically complex blade folding.

This project discusses the performance study and conceptual design of a hybrid tilt-rotor / tilt-wing vehicle, which consists of a tilting coaxial rotor system. In addition, the portion of the wing in the downwash of the rotors also tilt along with the rotors. This rotor system allows the vehicle to vertically take off and then transition into a fixed wing aircraft. Current design with coaxial rotor promises to offer several advantages over conventional tandem rotor tilt-rotor systems: 1) superior lateral stability in hover, compared to conventional tilt-rotor designs, 2) lower actuator forces and moments for tilting the rotors as coaxial rotors tend to have significantly lower gyroscopic loads compared to single rotors during transition, and 3) tilting of the wing portion below the rotor minimizes the hover losses due to aerodynamic download and interference of the rotor wake and the wing. The hover performance analysis would be carried out using Blade Element Momentum Theory (BEMT). The forward flight analysis would be carried out using a coaxial rotor dynamics analysis in which the blades would be modelled as rigid blades with hinge offset and root spring with only flap degree of freedom. Aerodynamic loads would be estimated using Blade Element Theory coupled to Drees inflow model. The objective is develop the analysis for systematic performance prediction of the proposed design and compare it with a conventional coaxial helicopter system. Impact of the proposed design changes on the key performance parameters such as speed and range would be discussed in detail.

Experimental study on leading edge effects in bypass transition

U. Yasar Arafath and A.C. Mandal

Department of Aerospace Engineering, IIT Kanpur 208016, India

In the presence of high freestream turbulence (FST), boundary layer transition does not follow the growth mechanism of Tollmien-Schilchting (TS) waves. Instead, transition that occurs rapidly, via some mechanisms yet to be fully understood, is often called bypass transition. Although there have been many analytical, experimental, and numerical studies of bypass transition, the path to turbulence while changing the leading edge shape is yet to be experimentally investigated, to the best of our knowledge. In this study, we, therefore, experimentally investigate the effect of leading edge shape on bypass boundary layer transition. Three different types of blunt nosed flat plates have been fabricated to carry out this experimental investigation. The experiments have been carried out in the low speed wind tunnel available at the low-speed laboratory. Our initial measurements reveal that there exist lambda structures in the boundary layer. Although these structures are generated near the leading edge, eventually low- and high-speed streaks appear in the flow. We speculate that these low- and high-speed

streaks are generated due to the stretching of the lambda structures generated at the initial stage of boundary layer transition. This is our ongoing study, and we need more number of experiments to establish this speculation.

Numerical Investigation of Combustion Acoustic Instability in atmospheric can combustor

Sudharsan K, Dinesh Kumar S J, Ashoke De, and Abhijitkushari

Department of Aerospace Engineering, IIT Kanpur 208016, India

The present work deals with numerical investigation of combustion acoustics in an atmospheric can combustor. For this study, Detached Eddy Simulation (DES) model is used for turbulence modeling and non–premixed steady flamelet model is used for turbulence-chemistry interaction. Coupled CFD – CAA solver is used for acoustic analysis. Initially, non–reacting calculation is carried out to study flow characteristics in the can combustor, where recirculation zones and acoustic frequency of the combustor are accurately captured. For reacting cases, two conditions are studied: one corresponds to best hooting condition and the other corresponds to worst hooting condition. Fluctuating pressure and heat release rate, correlating frequency and overall sound pressure level are computed. The result shows that changing mass flow rate of air in various inlets modifies the overall sound pressure level.

Keywords: Combustion instability, Acoustics, Detached Eddy simulation (DES), Computational Aero Acoustics (CAA), Non – premixed steady flamelet.

An experimental investigation of a geometry-induced separation bubble

V. S. Caesar and A. C. Mandal

Department of Aerospace Engineering, IIT Kanpur 208016, India

A geometry induced separation bubble has been investigated in this present experimental study. A blunt flat plate with right angled corners has been used to generate the bubble. The plate with chord-to-thickness ratio, c/t = 47, was horizontally mounted at the mid-plane of a low-speed low-turbulence

wind tunnel. The experiments were carried out at the Reynolds number, Re = 8.5 × 102 ; Re is defined based on the freestream velocity and the thickness of the plate. Hot-wire anemometry and time-resolved particle image velocimetry (TR-PIV) systems were used for data acquisition. The TR-PIV measurements clearly show the vortex shedding features of the separated shear layer, and unsteady reattachment on the surface of the plate owing to high c/t ratio. The shedding frequency, which is estimated from the hot-wire signal, is found to be due to the Kelvin-Helmholtz instability. Proper orthogonal decomposition (POD) of the wallnormal velocity data clearly reveal the signature of the underlying instability.

Planform Optimization of a Finite Wing

Varun Bhatt, Sanjay Mittal

Indian Institute of Technology, Kanpur, UP 208016, India

Planform optimization is performed for a finite wing with rounded wingtip. The objective is to find an optimal planform with maximum aerodynamic efficiency. The 3D computations are done for a finite wing at Re = 1000 and angle of attack 4o. The wing semi-span is kept fixed to 5 units. The cross-section of the wing is NACA 0012 throughout. In order to obtain optimal planform, chords at different spanwise stations are allowed to move along the streamwise direction and chord-lengths are allowed to vary. Two different strategy referred to as symmetrical constrained optimization and unsymmetrical constrained optimization are considered. Different planforms i.e. rectangular planform and elliptical planforms are considered for the optimization as initial guesses. The optimal planform obtained via the unsymmetrical constrained optimization for elliptical planform performs the best, leading to 19% increase in aerodynamic efficiency compared to the elliptical planform.

Key Words: *optimization, planform*

REFERENCES

[1] Srinath D., Mittal S., 2010. An adjoint method for shape optimization in unsteady viscous flows, *Journal of Computational Physics*, **229**, 1994-2008.

[2] Srinath D., Mittal S., 2007, A stabilized finite element method for shape optimization in low Reynolds number flows, *International Journal for Numerical Methods in Fluids*, **54**, 1451-1471.