|  |  |
| --- | --- |
| Indian Institute of Technology Madras  Department of Aerospace Engineering  Chennai, Tamil Nadu 600036  India | sghosh1@iitm.ac.in  Phone: +91-44-22574031  Mobile: +91 8056253470  Website: https://sites.google.com/site/iitmsghosh/ |

Santanu Ghosh, Ph.D.

Education

|  |  |
| --- | --- |
| *Aug 2006 – May 2010* | **North Carolina State University**  Doctor of Philosophy, Aerospace Engineering  Raleigh, North Carolina, United States |
| *Aug 2004 – May 2006* | **University of Texas at Arlington**  Master of Science, Mechanical Engineering  Arlington, Texas, United States |
| *Aug 1997 – May 2001* | **Jadavpur University**  Bachelor of Technology, Mechanical Engineering  Kolkata, West Bengal, India |

Professional Experience

|  |  |
| --- | --- |
| *Sep 2019 – Present* | **Associate Professor**  Indian Institute of Technology Madras, Department of Aerospace Engineering  Chennai, India |
| *Mar 2012 – Sep 2019* | **Assistant Professor**  Indian Institute of Technology Madras, Department of Aerospace Engineering  Chennai, India |
| *Jun 2010 – Mar 2012* | **Post-Doctoral Researcher**  North Carolina State University, Department of Mechanical and Aerospace Engineering  Raleigh, North Carolina, United States |

Research Statistics/Profiles

Scopus ID: 55478977500

Research Gate: Santanu Ghosh

Orcid: 0000-0002-5543-0388

Awards & Grants

*Awards*:

Title: Dean’s Fellowship

Year: 2004-2006

Awarded by: University of Texas an Arlington

Inducted into honor society Phi Kappa Pi in Fall 2007.

Inducted into engineering honor society Tau Beta Pi in Fall 2005.

*Grants*:

|  |  |
| --- | --- |
| *Title* | Numerical investigation of boundary-layer separation control in airfoils using surface porosity |
| *Duration* | 29-08-2017 to 28-08-2019 |
| *Sponsor* | ARDB (Aeronautical Research and Development Board) |
| *Role* | PI |
| *Value:* | INR 22,71,449 |
|  |  |
| *Title* | Numerical and Experimental Investigation of Flow-separation Control Using Passive Mechanisms in High-speed Flows |
| *Period* | 20-05-2019 to 19-05-2021 |
| *Sponsor* | MHRD (Ministry of Human Resource Development) |
| *Role* | PI (Indian) |
| *Value:* | INR 47,96,353 |
|  |  |
| *Title* | Variable camber morphing wing |
| *Period* | 01-06-2017 to 29-05-2022 |
| *Sponsor* | DRDO (Defense Research and Development Organization) |
| *Role* | Co-PI |
| *Value:* | INR 128610000 |
|  |  |
| *Title* | Experimental and Numerical Study of flow-separation control in Impinging-Shock/Boundary-Layer Interaction with Permeable Walls |
| *Period* | 24-01-2020 to 23-01-2021 |
| *Sponsor* | ICSR, IITM |
| *Role* | PI |
| *Value* | 8,21,000 |

Expertise & Activities

|  |  |
| --- | --- |
| *Expertise* | Fluid Mechanics, Flow Control, Turbulence, Computational Fluid Dynamics |
| *Reviewer* | Physics of Fluids,  AIAA Journal,  Chinese Journal of Aeronautics,  Journal of Aircraft,  Journal of Flow Visualization  Defense Science Journal  Fluid Mechanics and Fluid Power Conference (2014, 2018) |
| *Institute Memberships* | AIAA |
| *Conference/Workshop Organized* | SAROD (29th Nov – 1st Dec 2018): Member of Organizing Committee  Pointwise IIT Madras Meet (14th Dec 2018): Organizer |
| *Invited Lectures* | Title: Finite Volume Methods for Compressible Flows  Venue: Eashwari College of Engineering  Event: Faculty Development Program  Date: 12th June 2013  Title: Finite Volume Methods  Venue: Department of Civil Engineering, IIT Madras  Event: AICTE Short-term course  Date: 16th February 2017  Title: Immersed-boundary Methods for Compressible Flows  Venue: Department of Aerospace Engineering, IIT Madras  Event: AICTE Short-term course  Date: 13th March 2019  Title: Immersed-boundary Methods for Compressible Flows: Development and Applications  Venue: NAL, Bangalore  Event: 21st AeSI CFD conference  Date: 8th August 2019 |

Publications

Journal Publications (accepted/published)

1. ﻿﻿Bharadwaj, A. S., and Ghosh, S. “Numerical Investigation of Lift Enhancement in Flapping Hover Flight.” Physics of Fluids, Vol. 32, No. 5, 2020, p. 051901. https://doi.org/10.1063/5.0004021.
2. Sandhu, J., Girdhar, A., Ramakrishnan, R., Teja, R., and Ghosh, S. “FEST-3D: Finite-Volume Explicit STructured 3-Dimensional Solver.” Journal of Open Source Software, Vol. 5, No. 46, 2020, p. 1555. https://doi.org/10.21105/joss.01555.
3. Bharadwaj, A., and Ghosh, S. (2019). “Data Reconstruction at Surface in Immersed-Boundary Methods”, Computers and Fluids, *196*, 104236.
4. Roy, S., Subramaniam, K., and Ghosh, S. (2020). “Positioning of Normal Shock in a Novel Constant-Area Test Section: a Numerical Study”, AIAA Journal, *57*(12), 5582-5587.
5. Chakravarthy, R. V. K., Nair, V., Muruganandam, T. M., & Ghosh, S. (2018). Analytical and numerical study of normal shock response in a uniform duct. *Physics of Fluids*, *30*(8), 086-101. https://doi.org/10.1063/1.5027903
6. Sandhu, J. P. S., Ghosh, S., Subramanian, S., & Sharma, P. (2018). Evaluation of Ramp-Type Micro Vortex Generators Using Swirl Center Tracking. *AIAA Journal*, *56*(9), 3449–3459. https://doi.org/10.2514/1.j056796
7. Varma, D., & Ghosh, S. (2017). Flow Control in Mach 4.0 Inlet by Slotted Wedge-Shaped Vortex Generators. *Journal of Propulsion and Power*, *33*(6), 1428–1438. https://doi.org/10.2514/1.B36314
8. Sharma, P., Varma, D., & Ghosh, S. (2016). Novel Vortex Generator for Mitigation of Shock-Induced Flow Separation. *Journal of Propulsion and Power*, *32*(5), 1264–1274. https://doi.org/10.2514/1.B35962
9. Ghosh, S., Edwards, J. R., & Choi, Y.-I. (2012). Numerical Simulation of the Effects of Mesoflaps in Controlling Shock/Boundary-Layer Interactions. *Journal of Propulsion and Power*, *28*(5), 955–970. https://doi.org/10.2514/1.B34297
10. Ghosh, S., Choi, J.-I., & Edwards, J. R. (2010). Numerical Simulations of Effects of Micro Vortex Generators Using Immersed-Boundary Methods. *AIAA Journal*, *48*(1), 92–103. https://doi.org/10.2514/1.40049
11. Ghosh, S., Choi, J.-I., & Edwards, J. R. (2010). Simulation of Shock/Boundary-Layer Interactions with Bleed Using Immersed-Boundary Methods. *Journal of Propulsion and Power*, *26*(2), 203–214. https://doi.org/10.2514/1.45297
12. Ghosh S., Dennis B.H., Han Z-X, (2008) "Numerical Investigation of Moisture Diffusion Effects on Underfill within Flip-Chip Packages," Int. Review of Mechanical Engineering, *2*(3), 357-363.

Book Chapter (Invited):

Ghosh S., Anand Bharadwaj S (2020) Development and Application of Immersed Boundary Methods for Compressible Flows. In: Roy S., De A., Balaras E. (eds) Immersed Boundary Method. Computational Methods in Engineering & the Sciences. Springer, Singapore

Conference Papers / Proceedings

[1] A. Bharadwaj S and S. Ghosh, “Second-order Interpolation Techniques for Accurate Surface Data Estimation in Immersed-Boundary Methods,” in *2018 AIAA Applied Aerodynamics Conference*, 2018.

[2] A. Sahoo, S. Roy, and S. Ghosh, “Numerical Investigation of Transonic Flow over Porous Medium Using Immersed Boundary Method,” in *2018 AIAA Applied Aerodynamics Conference*, 2018.

[3] J. P. Singh Sandhu, A. Girdhar, R. Ramakrishnan, R. D. Teja, and S. Ghosh, “A convergence study of solutions using two two-equation RANS turbulence models on a finite volume solver for structured grids,” in *2018 AIAA Fluid Dynamics Conference*, 2018.

[4] Aditya Madabushi, Santanu Ghosh, “COMPARATIVE ASSESSMENT OF TWO ALL-SPEED FLOW ALGORITHMS USING EXPLICIT TIME MARCHING,*”* *in ASCHT 2017 6th Asian Symposium on Computational Heat Transfer and Fluid Flow*, 2017.

[5] S. Roy, K. Subramaniam, and S. Ghosh, “Passive Control of Normal-shock-wave/Boundary-layer Interaction Using Porous Medium: Computational Study,” in *35th AIAA Applied Aerodynamics Conference*, 2017.

[6] A. Bharadwaj S, S. Ghosh, and C. Joseph, “Interpolation Techniques for Data Reconstruction at Surface in Immersed Boundary Method,” in *55th AIAA Aerospace Sciences Meeting*, 2017.

[7] J. P. S. Sandhu, S. Subramanian, S. Ghosh, and P. Sharma, “Evaluation of some wedge-shaped vortex generators using swirl center tracking,” in *8th AIAA Flow Control Conference*, 2016.

[8] Dheepak Nandkishore Khatri, Rajesh Gopalapillai, Santanu Ghosh, Anil Kumar Pasam, “An investigation of hysteresis phenomenon in reflection of asymmetric shock waves,*”* i*n 22nd international shock interaction symposium*, 2016.

[9] R. Ramakrishnan, A. Girdhar, and S. Ghosh, “IMMERSED BOUNDARY METHODS FOR COMPRESSIBLE LAMINAR FLOWS,” in *ECCOMAS Congress VII European Congress on Computational Methods in Applied Sciences and Engineering*, 2016.

[10] D. Varma, S. Saurav, and S. Ghosh, “Flow Control in a Mach 4.0 Inlet by Slotted Wedge-shaped Vortex Generators,” in *33rd AIAA Applied Aerodynamics Conference*, 2015.

[11] P. Sharma and S. Ghosh, “A Novel Vortex Generator for Mitigation of Shock-Induced Separation,” in *52nd Aerospace Sciences Meeting*, 2014.

[12] S. Ghosh, J.-I. Choi, and J. Edwards, “Numerical Simulation of the Effects of Mesoflaps in Controlling Shock / Boundary Layer Interactions,” in *40th AIAA Fluid Dynamics Conference and Exhibit*, 2010.

[13] J. R. Edwards, J.-I. Choi, S. Ghosh, D. A. Gieseking, and J. D. Eischen, “An immersed boundary method for general flow applications,” in *American Society of Mechanical Engineers, Fluids Engineering Division (Publication) FEDSM*, 2010, vol. 1, no. PARTS A, B AND C.

[14] S. Ghosh, J.-I. Choi, and J. R. Edwards, “Simulations of shock/boundary layer interactions with bleed using immersed boundary methods,” in *47th AIAA Aerosp. Sci. Meet.*, 2009.

[15] S. Ghosh, J.-I. Choi, and J. Edwards, “RANS and Hybrid LES/RANS Simulation of the Effects of Micro Vortex Generators Using Immersed Boundary Methods,” in *38th AIAA Fluid Dynamics Conference and Exhibit*, 2008.

Research

Research at University of Texas at Arlington, Arlington, TX, USA, 2004-2006

***Analysis of hygroscopic and thermal stresses in flip-chip ball-grid array packages***

During this period I had worked on simulating the thermal and hygroscopic stresses in flip-chip ball-grid array packages during the reflow process using the ANSYS software. I also investigated the effects of varying amounts of underfill material used in such packages on the moisture related stresses.

Research at North Carolina State University, Raleigh, NC, USA, 2006-2012

***Development and application of immersed-boundary method for compressible turbulent flows***

In the course of my doctoral research, I was involved in the development of an immersed-boundary technique suitable for high speed flows and applied the method to simulate the effects induced by boundary-layer control devices used in the control of shock/boundary-layer interactions. This method was developed as an extension of an existing immersed-boundary method, which was developed earlier at our lab for incompressible flows at all, speeds. The immersed-boundary method is a non-body-conforming methodology in which the effects on the flow due to an object embedded in the domain are mimicked by use of proper conditions near the boundary of the immersed object. This can reduce to a great extent the complexity involved in grid-generation or even grid-adaptation in simulating flows around complicated objects, especially when these objects are moving. Thus, the potential advantages of an IB method in the simulation of boundary layer control devices include significant economy in the number of mesh points required to render the control device (compared to body-fitted meshes), ease with which different types of control devices can be interchanged and their effects assessed, and the ability to model moving control devices without mesh adaptation. I performed simulations of three different boundary-layer control devices – wedge shaped vortex generator(s), array of 90⁰ bleed holes, and aero-elastically deflecting flaps (mesoflaps) - which are used in the control of shock/boundary-layer interactions which occur in mixed compression high speed inlets. Each of these cases had its own challenges, and working on these different cases enriched my understanding of the capabilities and limitations of the IB method and made me work towards its improvement. Also, for simulating the control device mesoflap I had to solve a fluid-structure interaction problem. To do this, I integrated a finite difference based structural solver –– which involved modeling of the mesoflaps as Euler-Bernoulli beams –– with the immersed-boundary method I had worked with, to develop a coupled system.

I worked as a post-doctoral researcher with Dr. Jack R. Edwards at North Carolina State University, Raleigh, NC from June 2010 to March 2012. During this period I was involved in the simulation of a Mach 4 flow through a turbine-based combined-cycle inlet using the solver REACTMB, which is a finite volume based code developed at our lab for high-speed turbulent flows. My work was based on experiments conducted at NASA Glenn Research Center (GRC) [1] in their 1 ft x 1 ft supersonic wind tunnel with a small-scale model of the afore-mentioned intake. In the experiments, the turbine flowpath consisted of nine bleed regions in order to control the adverse effects of shock / boundary-layer interaction and help achieve stable operation of the inlet and smooth mode transition. The boundary layer suction through the bleed regions play an important role in stabilizing the normal shock positioning within the turbine flowpath and prevent unstarts. As such, accurate prediction of bleed rates and bleed patterns is important for the computation of inlet flows. Conventional CFD methods use models for determining the bleed rates through the bleed surfaces in place of resolving the flow through the individual bleed holes. In general, the primary obstacle in resolving the flow through the individual grid holes is the complexity of the grid generation through the multiple holes and the associated high grid volume (number of cells), which would make the computations expensive. We used the immersed-boundary approach developed during my PhD to reduce the computational cost involved.

Research at Indian Institute of Technology Madras, Chennai, India, 2012-Present

***Development of FEST3D solver***:

Finite-volume Explicit STructured 3 Dimensional (FEST3D) is a Navier-Stokes solver for laminar / turbulent flows. This code is parallel and suitable for inviscid, laminar and turbulent flow simulations at low (subsonic) and high (supersonic) speeds. We have been verifying the code against benchmark solutions provided on NASA’s validation and verification website. Salient features of the solver are listed below.

* Highly modular code
* Uses MPICH for multi-block parallel processing.
* CMAKE implementation for making compilation platform independent.
* Multiple options for:
  + Higher-order spatial reconstruction
    - MUSCL, PPM, WENO
  + Temporal reconstruction (implict and explicit)
    - RK2, RK4, TVD RK2, TVD RK3, LU-SGS
  + Turbulence model
    - SA, Menter’s SST, k-kl
  + Transition model
    - 1-EQN γ model, SA-BC model

**The details about the solver (with references for the methods used) are available at the following website:** [**http://fest3d.github.io/**](http://fest3d.github.io/)

***Data reconstruction at surfaces in immersed-boundary methods***:

Immersed-boundary methods are a special class of CFD (computational fluid dynamics) methods, which do not require grid conformity at the surfaces of immersed objects. These are aimed to make the grid-generation process, an integral part (and in many cases a bottleneck for CFD simulations) easy, and are as such suitable for simulating flow filed past objects of intricate shape and / or moving boundaries. In our work we specifically use upwind-biased inverse-distance based weights for reconstructing the pressure and shear stress on the surface of the immersed object using neighboring fluid cell data. The immersed surface is represented as a cloud of points with additional information about the local shape stored as (local) normal (perpendicular) vectors. The pressure and shear stress (reconstructed at the immersed surface) is integrated using information about the normal vectors to obtain the forces acting on the immersed object.

**Practical applications may include design studies of multi-component airfoils to increase lift and reduce drag and investigations of time varying lift/drag characteristics for flow past pitching / plunging airfoils.**

***Flow control with ramp-type micro vortex generators***:

Vortex generators are roughness elements, which introduce vortex pairs in the wake downstream of the device. These vortices are considered to energize the near-surface flow, making it resistant to separation in the face of adverse pressure gradients. We have compared the flow-field downstream of four different ramp-type micro (height less than local boundary layer) vortex generators using RANS (Reynolds-averaged Navier-Stokes) simulations couple to an immersed-boundary method, which renders the devices. The devices considered are the ramp, split-ramp, slotted-ramp and slotted-ramp with taper The performance of the different vortex generators is compared using metrics like near-surface streamwise velocity contours, and outgoing boundary-layer integral properties (displacement thickness, momentum thickness and pressure recovery factor). It is observed that the presence of a secondary pair of streamwise vortices, which are closer to the wall and the mid-plane of the device compared to the primary vortices, have an effect on the trajectory of the primary vortices. This in turn affects the flow-field and performance of the vortex generators.

**The practical use of these micro-vortex generators can be on wings and in the intakes of jet aircrafts to reduce flow-separation and associated detrimental effects like buffeting (wings) and unstart (inlets).**

***Investigations of flow-separation control using porous medium:***

We plan investigate the flow past a porous airfoil surface at transonic speeds to assess its effectiveness in reducing flow separation due to existence of slip velocities at the interface. The porous surface is modeled as a cavity with circular cylinders (representing rod like structures) running across its span. An immersed-boundary technique is to used for rendering the cylinders. We would also like to subsequently extend this work to 3D wherein attempts will be made to model the porous regions as a 3D structure, having better fidelity with actual structure of porous medium.

***This work is part of the work proposed under the sanctioned SPARC proposal “Numerical and Experimental Investigation of Flow-separation Control Using Passive Mechanisms in High-speed Flows” (20.05.2019-19.05.2021).***

***Development of a zero-equations transition model***:

We attempt the development of a local intermittency-based zero-equation transition model that derives from the 1-equation transition model of Menter. The motivation in deriving such a model is that it does not require us to solve a separation transport or algebraic equation for the intermittency, but combines it with the transport equation of the turbulence kinetic energy. We define a new quantity –– the intermittent turbulent kinetic energy –– and solve its transport equation. This equation has new source terms, which are modeled using known quantities and flow physics. The resultant transport equation is similar in appearance to the turbulent kinetic energy transport equation, with some additional source terms, which makes it very easy to implement in existing source codes, which solves the transport of turbulent kinetic energy.

**Applications of the new transition model can be in airfoil drag prediction, especially for natural laminar flow airfoils, and aircraft wings, among others.**

***Aerodynamic analysis of morphing wing in UAV***:

This research is aimed at investigation of the quasi-steady and unsteady aerodynamics of morphing wings using OpenFOAM and immersed-boundary methods. The motivation of airfoil morphing stems from reducing the drag during two distinct flight regimes –– loiter and dash –– with prescribed values of lift coefficient. The morphing of the airfoil section will be achieved using camber morphing. Aerodynamic characteristics of the morphing wing, assuming both quasi-steady and unsteady approaches, will be analysed. The results from these investigations will help in understanding the advantages of using morphing technologies from the standpoint of aerodynamics and help in developing morphing wing technologies for integration with UAVs.

**This work is part of the work proposed under the sanctioned CoPT (Centre for Propulsion Technologies) proposal “Variable camber wing morphing” (01.06.2017-29.05.2022).**

Work Experience

|  |  |
| --- | --- |
| *Mar 2012 – present* | Indian Institute of Technology Madras, Department of Aerospace Engineering  Chennai, India (**Assistant Professor, Associate Professor**)  *Research*: My research activities include development of solver for turbulent fluid flows (FEST-3D), devising algorithms for immersed-boundary methods in fluid flows, application of immersed-boundary methods for flow-control, devising laminar-turbulence transition models etc.  *Teaching*:   * AS 5020: Gas Dynamics and Propulsion (2012) (0.5 load) * AS 2030: Gas Dynamics (2013) * AS 1020: Fluid Mechanics (2017) * AS 2040: Flight Dynamics I (2016, 2018) * AS 5010: Aerodynamics (2014) * AS 5040: Flight Performance and Control (2015) * AS 5210: Aircraft Design (2016, 2017) * AS 5330: Computational Aerodynamics (2014, 2015, 2016) * **AS 5460: Finite Volume Methods for Hyperbolic PDEs** (2017, 2018, 2019) * AS 5410: Grid Generation (2013, 2015) * AS 6520: Mathematics for Aerospace Engineers (2014) (0.5 load) * AS 6000: Fundamentals of Aerospace Engineering (2015, 2019) (Part load)   New course: **Finite Volume Methods for Hyperbolic PDEs**  The course is aimed to serve as an introduction for graduate students (and 6th semester UG/DD) to CFD using primarily finite volume methods. The students will be offered an insight into the solution methodology for problems related to fluid flows that are governed by hyperbolic PDEs. The students will learn algorithms and write programs to numerically solve flows with discontinuities like shock waves and contact waves that occur in high-speed compressible flows.  *Guidance*:   * Research guidance:   + MS – 2 completed, 1 in progress   + PhD – 1 completed, 4 in progress (1 co-guided, 1 external). * Project guidance   + M.Tech – 5 completed, 0 in progress   + DD – 5 completed, 0 in progress   Other *Activities*   * Advising of B.Tech students. * Course-syllabus development: AS5011 (Compressible Fluid Flows). * MS/PhD shortlisting and interviewing of candidates. * Summer Fellowship shortlisting |
| *Jun 2010 – Mar 2012* | North Carolina State University, Department of Mechanical and Aerospace Engineering (**Post-Doctoral Researcher**)  Raleigh, North Carolina, United States  *Research:* Simulation of super-critical flow through dual-mode inlet at Mach 4.0 with flow control by suction using immersed-boundary methods. |