MVJ COLLEGE OF ENGINEERING
Since 1982
Engineering A Better Tomorrow
An Autonomous Institute
Approved by AICTE, New Delhi
Affiliated to Visvesvaraya Technological University, Belagavi
Recognized by UGC with 2(f) and 12(B) status
Accredited by NBA and NAAC

:

Three-Day Faculty Development Programme (FDP) on "Application of CFD in Thermal and Fatigue Analysis"

Department of Mechanical Engineering – Event Report

The **Department of Mechanical Engineering** organised a three-day FDP titled **Recent Application of CFD in Thermal and Fatigue Analysis** from **16-03-2026 to 18-03-2026** from **10.00 AM to 04.00 PM**, in **Seminar Hall 02**.

Session-1
Topic: From Simulation to Sustainability: CFD Solutions for Fatigue and Thermal Assessment.
Time: 16.03.2026 FN
Resource Person: Shri Alok Hegde

The Session 1 Day 1 "From Simulation to Sustainability: CFD Solutions for Fatigue and Thermal Assessment" by **M**r. **Alok Hegde,** Scientist-D Central Manufacturing Technology Institute, Bangalore, provided an in-depth overview of how Computational Fluid Dynamics (CFD) has evolved into a powerful tool that bridges the gap between advanced simulation and sustainable engineering practices. By enabling precise modeling of fluid flow, heat transfer, and structural interactions, CFD provides engineers with deep insights into how systems behave under real-world operating conditions.

One of its critical applications lies in fatigue assessment, where CFD helps predict stress concentrations and cyclic loading effects on materials. Through accurate simulation of flow-induced vibrations and pressure fluctuations, engineers can identify potential failure points early in the design stage, thereby extending component life and reducing maintenance costs.

Similarly, thermal assessment benefits greatly from CFD's ability to model heat distribution, cooling efficiency, and thermal stresses. This ensures that designs not only meet performance requirements but also minimise energy consumption and environmental impact. By optimising cooling strategies and material selection, CFD contributes directly to sustainable engineering solutions.

In essence, CFD empowers engineers to move beyond simulation as a mere analytical tool—transforming it into a cornerstone of sustainable innovation, where durability and efficiency converge to create greener, more resilient technologies.

Objectives of the Event

- Leverage CFD for fatigue analysis: Predict stress concentrations, cyclic loading effects, and flow-induced vibrations to enhance component durability.
- Enable accurate thermal assessment: Model heat transfer, cooling efficiency, and thermal stresses to optimise energy use and material performance.
- Promote sustainability in design: Reduce resource wastage and energy consumption by integrating predictive simulations into engineering workflows.
- Improve reliability and safety: Identify potential failure points early, minimising maintenance costs and extending product life cycles.
- Support data-driven innovation: Transition from trial-and-error methods to predictive, simulation-based approaches for sustainable engineering solutions.
- Integrate multidisciplinary insights: Combine fluid dynamics, structural mechanics, and thermal modeling for holistic system evaluation.

Event Overview

The Department of Mechanical Engineering organised a Three-Day FDP on the topic “Recent Application of CFD in Thermal and Fatigue Analysis”. The resource person showed that integration of CFD into fatigue and thermal analysis represents a shift from traditional trial-and-error approaches to a data-driven, predictive methodology. This transition enhances reliability, reduces resource wastage, and supports the broader goal of sustainability in engineering design and manufacturing.

The resource person highlighted bridging advanced simulation with sustainability goals in engineering. Fatigue assessment through CFD modeling of stress cycles, vibrations, and flow-induced loads. Thermal assessment for heat transfer, cooling strategies, and energy optimisation. Demonstrate how predictive simulation reduces resource wastage, enhances reliability, and supports sustainable innovation. Participants gain insight into integrating CFD into design workflows for improved product life cycles, reduced maintenance costs, and greener technologies.

Significance: The event underscores the transition from traditional trial-and-error methods to data-driven, simulation-based engineering, aligning with global sustainability initiatives.

Photographs from the Event

As shown in Figure 1, Dr. Arun Kumar K, the in-charge and head of the Department of Mechanical Engineering, welcomed the chief guest and participants to the guest lecture.

Figure 1: Session delivery by the resource person

Figure 2 shows the photographs of the students listening eagerly to the resource person.

(a) Faculty members, resource person, and students gathering during the guest lecturer

(b) Dr. Arun Kumar K delivering the vote of thanks

Schedule of the Event

Day	Time	Session Details
Monday, March 16, 2026	10.00 am - 10.10 am 10.10 am - 10.20 am	Welcome Address: Introduction about the Resource Person
Friday, February 16, 2026	10.30 am - 12.20 pm 12.20 pm - 12.30 pm	Session Delivery Vote of Thanks

Table 1: Schedule of the Event

Table 1 gives the schedule of the event. The session began with the introduction of the resource person by the Head of the Department, followed by the overview of the guest lecture by Dr. Arun Kumar K, Associate Professor, Department of Mechanical Engineering and Coordinator of the Three-Day FDP.

Outcomes and Impact

- **Enhanced Understanding of CFD Applications**: Participants gain clarity on how Computational Fluid Dynamics (CFD) can be applied beyond traditional flow simulations, specifically in **thermal management** and **fatigue life prediction**.
- **Integration of Sustainability Goals**: The session highlights how CFD-driven design reduces material wastage, optimises energy usage, and supports sustainable engineering practices.
- **Skill Development**: Attendees acquire practical insights into linking simulation outputs with real-world fatigue and thermal performance, strengthening their ability to apply CFD in industry and research.
- **Cross-Disciplinary Relevance**: Demonstrates how CFD bridges mechanical design, materials engineering, and sustainability, encouraging participants to adopt holistic approaches in problem-solving.
- **Industry Readiness**: Exposure to case studies and workflows prepares participants to implement CFD tools in **automotive, aerospace, and energy sectors**, where fatigue and thermal challenges are critical.

Conclusions

By demonstrating how CFD seamlessly connects simulation with sustainability, this session establishes that accurate thermal and fatigue assessments not only extend product life and enhance safety but also drive resource efficiency, reduce environmental impact, and empower engineers to design solutions that are both technically robust and aligned with global sustainability goals.

.

Report by: Dr. Rajesh Kumar P

Affiliation: Faculty in the Department of Mechanical Engineering, MVJ

College of Engineering

Session-2
Topic: Design and CFD analysis of a turbine stage
Time: 16.03.2026 AN
Resource Person: Dr. Vimala Narayan

Session 2 Day 1: The afternoon session focused on the design methodology and CFD-based performance evaluation of turbine stages, a critical component in power generation and aerospace applications. **Dr. Vimala Narayan** emphasised how Computational Fluid Dynamics (CFD) enables engineers to analyse aerodynamic efficiency, pressure distribution, and thermal stresses in turbine blades under realistic operating conditions.

Objectives of the Event

- Apply CFD tools to turbine stage design for improved aerodynamic efficiency.
- Demonstrate fatigue and thermal analysis of turbine blades under operational stresses.
- Highlight the role of predictive simulations in reducing trial-and-error design cycles.
- Showcase industry relevance in aerospace, energy, and power plant engineering.

Event Overview

The afternoon session focused on the design methodology and CFD-based performance evaluation of turbine stages, a critical component in power generation and aerospace applications. Dr. Vimala Narayan emphasised how Computational Fluid Dynamics (CFD) enables engineers to analyse aerodynamic efficiency, pressure distribution, and thermal stresses in turbine blades under realistic operating conditions.

The lecture highlighted:

- Design principles of turbine stages, including blade geometry, flow passage optimisation, and material considerations.
- CFD applications in predicting flow behavior, turbulence, and heat transfer within turbine channels.
- Performance assessment through simulation of efficiency, losses, and fatigue life under cyclic thermal and mechanical loading.
- Sustainability aspects by optimising turbine designs to reduce fuel consumption and emissions.

Photographs from the Event

Figure 1: **Dr. Vimala Narayanan,** Former Scientist G and Technical Director, Gas Turbine Research Establishment (GTRE), Bangalore, delivering a talk on “Design and CFD analysis of a turbine stage”

Figure 1: Session delivery by the resource person

Figure 2: Photographs of the resource person and students' participation in the activity

Schedule of the Event

Table 1 outlines the event schedule. The session began with an introduction to the resource person by the Head of the Department, followed by an overview of the guest lecture by Dr.

Arun Kumar K, Associate Professor in the Department of Mechanical Engineering and Coordinator of the Three-Day FDP.

Day	Time	Session Details
Monday, March 16, 2026	1.30 pm - 1.35 am 1.35 pm - 1.40 pm	Welcome Address: Introduction about the Resource Person
Friday, February 16, 2026	1:45 pm - 3.20 pm 3.20 pm - 3.30 pm	Session Delivery Vote of Thanks

Table 1: Schedule of the Event

Outcomes and Impact

- Enhanced Knowledge: Participants developed a deeper understanding of turbine stage design and CFD-based analysis.
- Skill Development: Exposure to advanced simulation techniques prepared attendees for industry challenges.
- Sustainability Integration: The session reinforced the importance of energy-efficient turbine designs aligned with global sustainability goals.
- Industry Readiness: The insights provided prepare participants for roles in aerospace propulsion, energy systems, and high-performance machinery design.

Conclusions

The session by Dr. Vimala Narayan successfully demonstrated how CFD transforms turbine stage design into a data-driven, sustainable, and reliable engineering process. By bridging simulation with real-world performance, engineers can achieve higher efficiency, longer component life, and reduced environmental impact.

.

Report by: Dr. Arunkumar K

Affiliation: Faculty in the Department of Mechanical Engineering, MVJ College of Engineering

Session-3
Topic : Conjugate heat transfer.
Time: 17.03.2026 FN
Resource Person: Shri Dinesh S

The FDP session on "Conjugate Heat Transfer" by Dinesh S provided a comprehensive exploration of how coupled thermal-fluid simulations have become essential in modern engineering. Conjugate Heat Transfer (CHT) integrates fluid dynamics with solid heat conduction, enabling engineers to analyse how heat moves simultaneously through fluids and solid structures. This dual-domain modeling ensures accurate predictions of system performance under realistic operating conditions. The resource person, Shri Dinesh, emphasised that CHT is not merely a computational exercise but a critical design tool for industries ranging from aerospace and automotive to electronics cooling and energy systems. By capturing the interaction between fluid flow and solid conduction, CHT helps optimise cooling strategies, material selection, and overall thermal management. A total of 25 faculty and students from the 2nd and 3rd years were present for the session.

Objectives of the Event

- **Enable precise thermal modeling:** Simulate heat transfer across both fluid and solid domains for accurate system-level analysis.
- **Optimize cooling strategies:** Evaluate cooling efficiency in engines, turbines, and electronic devices to ensure reliability and performance.
- **Promote sustainability in design:** Reduce energy consumption and material wastage by integrating predictive CHT simulations into the workflow.
- **Improve durability and safety:** Identify thermal hotspots and stress points early in the design stage to extend component life cycles.
- **Support data-driven innovation:** Transition from empirical trial-and-error methods to predictive, simulation-based engineering approaches.
- **Integrate multidisciplinary insights:** Combine fluid dynamics, heat conduction, and structural mechanics for holistic evaluation of thermal systems.

On Day 2 of the Three-Day FDP on *"Recent Applications of CFD in Thermal and Fatigue Analysis."*, the focus shifted to **Conjugate Heat Transfer (CHT)**, where Shri Dinesh demonstrated from the basics of CFD and heat transfer and transitioned to CFD simulations.

The session highlighted:

- Bridging advanced simulation with sustainability goals in thermal management.
- CHT modeling of heat conduction in solids and convective transfer in fluids.
- Applications in cooling of electronics, aerospace propulsion systems, and energy-efficient manufacturing.
- How predictive simulation reduces energy wastage, enhances reliability, and supports sustainable innovation.
- Participants gained insights into integrating CHT into design workflows for improved product reliability, reduced maintenance costs, and greener technologies.

The event underscored the importance of **Conjugate Heat Transfer** as a cornerstone of modern engineering design. By moving away from isolated thermal or fluid analyses, CHT provides a unified, data-driven approach that aligns with global sustainability initiatives. It ensures that designs are not only efficient but also resilient, supporting the broader vision of sustainable innovation in engineering.

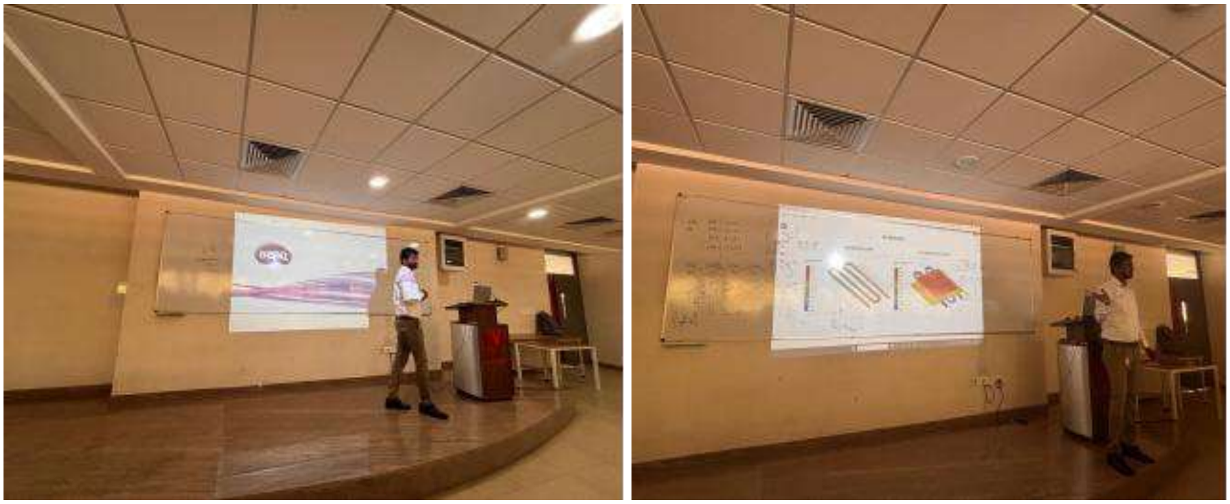

Figure 1: Mr. Dinesh S delivery lecture to the faculties and students

Figure 2: Vote of thanks by Dr. Shrinivas L Gombi, HoD, ME

Session-4

Topic: Recent trends in CFD modelling of Thermal and Fluid Sciences. Time: 17.03.2026 AN

Resource Person: Dr. Ninge Gowda

The Department of Mechanical Engineering organised a three-day FDP titled Recent Application of CFD in Thermal and Fatigue Analysis on 17-03-2026 from 1.30 PM to 3.30 PM, in Seminar Hall 02.

Session 2, Day 2, The Session on “Thermal Performance and Flow Distribution Analysis of Compact Heat Exchangers for Aircraft Environmental Control System using CFD” provided a deep dive into the thermal performance and flow distribution analysis of compact heat exchangers used in Aircraft Environmental Control Systems (ECS). This session addressed the critical engineering challenge of managing high-temperature bleed air within the strict weight and volume constraints of modern aircraft to ensure passenger safety and comfort. By utilizing Computational Fluid Dynamics (CFD), researchers can identify flow maldistribution and analyse conjugate heat transfer, which are essential for optimising the efficiency of these complex thermal components. Expert resource persons from prestigious organizations like the Gas Turbine Research Establishment (GTRE) and the Aeronautical Development Agency (ADA) shared practical approaches to resolving these aerospace challenges.

The program emphasised the transition from traditional simulation to sustainable engineering by focusing on CFD solutions for energy efficiency and thermal assessment. Participants gained insights into advanced topics such as multiphase flow, electronic cooling, and the integration of Artificial Intelligence to accelerate the design iteration process for mechanical applications. This comprehensive analysis fosters cross-disciplinary collaboration among faculty in mechanical, aerospace, and civil engineering, aligning academic research with cutting-edge industry trends. Ultimately, the FDP equipped attendees with the technical capabilities to integrate these sophisticated CFD methodologies into both their teaching and applied research projects.

Key Technical Themes

The session delved into several specialised areas of CFD modeling:

- **Flow Distribution Analysis:** Examining how air moves through the heat exchanger core to ensure uniform cooling and prevent thermal stresses.
- **Thermal Performance:** Evaluating the heat rejection capabilities of compact designs under varying flight conditions.

- **Advanced Modeling:** The integration of multifluid modeling and the potential for AI-enhanced CFD to optimise aerospace research.
- **System Sustainability:** Aligning CFD solutions with recent trends in energy efficiency and sustainable engineering practices.

In essence, CFD transforms the way aircraft heat exchangers are engineered shifting from traditional empirical design practices to a predictive, simulation-driven methodology. This empowers engineers to create compact exchangers that are lighter, more energy-efficient, and more reliable, directly contributing to safer, high-performance aircraft systems.

Objectives of the Event

The primary goal was to enhance faculty and researcher expertise in advanced CFD techniques, specifically for mechanical and aerospace applications. The session aimed to provide:

- **Theoretical Insights:** Understanding the fluid dynamics and heat transfer principles governing aircraft ECS.
- **Practical Approaches:** Demonstrating how CFD is used to solve real-world engineering challenges in high-performance cooling systems.
- **Performance Metrics:** Analyzing flow maldistribution and thermal effectiveness to improve system reliability

Event Overview

The Department of Mechanical Engineering organised a specialised session within the three-day Faculty Development Programme (FDP) focusing on the **"Thermal Performance and Flow Distribution Analysis of Compact Heat Ex-changer for Aircraft Environmental Control System using CFD"**. The session emphasised how the integration of Computational Fluid Dynamics (CFD) into the design of Environmental Control Systems (ECS) marks a transformative shift from traditional empirical methods to a high-fidelity, simulation-driven approach. By enabling the precise evaluation of complex **Conjugate Heat Transfer**, flow uniformity, and pressure drop characteristics, CFD enhances the reliability of critical aerospace components while significantly reducing experimental costs and development lead times.

The expert resource persons highlighted how advanced modeling bridges the gap between theoretical fluid mechanics and the increasing demand for high-performance thermal management in modern aviation. Through detailed simulation of coolant flow paths, turbulence behavior, and heat transfer effectiveness, engineers can optimise **Compact Heat Exchanger** geometries to ensure stable operation across diverse flight profiles. The session demonstrated how predictive modeling strengthens system durability, improves operational safety, and supports sustainability by minimizing energy waste. Participants gained valuable insights into incorporating CFD tools into real-world aerospace workflows—leading to improved product life cycles, reduced maintenance requirements, and more energy-efficient aircraft systems.

Significance:

The event reinforces the analysis of compact heat exchangers for Aircraft Environmental Control Systems (ECS), as it directly impacts passenger safety and comfort by managing high-temperature bleed air within the cabin. Using CFD allows engineers to optimise thermal performance and identify flow maldistribution, which are critical for maintaining system reliability under extreme flight conditions. This topic is strategically significant for research, as it aligns mechanical engineering practices with global sustainability goals by reducing the fuel penalty associated with aircraft cooling. Furthermore, the session provided faculty with advanced insights into conjugate heat transfer and multiphase flow, bridging the gap between academic theory and complex aerospace applications. By integrating expert knowledge from organizations like GTRE and ADA, the topic fosters cross-disciplinary innovation necessary for modern aviation and defense sectors.

Photographs from the Event :

As shown in Figure 1, Dr. Ninge Gowda, Director and Technical Lead, SS Integrated Tech Solutions, Bangalore, addressed the gathering and initiated the session by sharing valuable insights with the participants.

Figure 1: Session delivery by the resource person

Figure Prof. Vinothkumar G delivering vote of thanks

Schedule of the Event

The Table 1 gives the schedule of the event. The session began with the introduction of the resource person by the Head of the Department followed by the overview of the guest lecture by Prof. Shivakumar H D, Assistant Professor, Department of Mechanical Engineering.

Day	Time	Session Details
Tuesday, March 17, 2026	1.30 pm - 1.40 pm 1.40 pm - 1.50 pm	Welcome Address: Introduction about Resource Person
Wednesday, March 17, 2026	2.00 pm - 3.50 pm 3.50 pm - 4.00 pm	Session Delivery Vote of Thanks

Table 1: Schedule of the Event

Outcomes and Impact:

- **Enhanced Understanding of CFD-Based Heat Exchanger Analysis:** Participants gained clarity on how CFD can be applied beyond basic flow simulation to evaluate heat-transfer performance, flow maldistribution, and pressure-drop characteristics in compact aerospace heat exchangers.
- **Integration of Efficiency-Driven Design Goals:** The session demonstrated how CFD-guided optimisation reduces thermal losses, improves cooling uniformity, and supports energy-efficient, sustainable thermal-management solutions for aircraft systems.
- **Skill Development in Simulation Interpretation:** Attendees acquired practical insights into connecting CFD results—such as temperature fields, velocity profiles, and turbulence effects with real-world heat-exchanger behavior and operational performance.
- **Cross-Disciplinary Relevance in Aerospace Engineering:** The discussions highlighted how CFD links fluid dynamics, heat-transfer physics, and system-level aerospace requirements, encouraging participants to adopt integrated approaches in solving thermal-control challenges.
- **Industry Readiness for Advanced Thermal Systems:** Exposure to case studies and modeling workflows prepared participants to apply CFD tools effectively in aerospace

cooling systems, improving reliability, lowering maintenance costs, and meeting stringent aviation standards.

Conclusions

With the expert guidance of leaders from the Gas Turbine Research Establishment (GTRE) and the Aeronautical Development Agency (ADA), participants gained the technical depth required to resolve high-stakes challenges in conjugate heat transfer and multiphase flow. Ultimately, this FDP has fostered a collaborative environment for cross-disciplinary innovation, ensuring that the academic community remains at the forefront of advanced aerospace thermal management solutions.

Report by: Prof. Shivakumar H D

Affiliation: Faculty in the Department of Mechanical Engineering,

MVJ College of Engineering

Session-5

Topic: Thermal Performance and Flow Distribution Analysis of Compact Heat Exchanger for Aircraft Environmental Control System using CFD. Time: 18.03.2026 FN

Resource Person: Dr. Ramanamurthy

Session 1 Day 3: The Session on "Thermal Performance and Flow Distribution of Compact Heat Exchangers for Aircraft Control Systems using CFD" provided a comprehensive insight into how modern Computational Fluid Dynamics (CFD) acts as a transformative analytical tool in aerospace thermal management. Compact heat exchangers play a critical role in maintaining optimal operating temperatures across aircraft control systems, where reliability, precision, and efficiency are non-negotiable.

By simulating complex fluid–thermal interactions, CFD allows accurate prediction of temperature gradients, pressure drops, and flow maldistribution—factors that directly influence exchanger efficiency and component life. Engineers can visualize how air or coolant navigates intricate passage geometries, identify regions of recirculation or hot spots, and refine designs to boost heat transfer while minimising energy losses.

A key application of CFD in this context is optimising flow distribution across the exchanger core. Uneven flow can severely degrade performance, reduce heat transfer rates, and accelerate fatigue. By evaluating how geometry, orifice sizing, inlet conditions, and operating pressures affect coolant pathways, CFD supports the development of highly compact yet thermally robust designs suitable for modern aircraft architectures.

In essence, CFD transforms the way aircraft heat exchangers are engineered, shifting from traditional empirical design practices to a predictive, simulation-driven methodology. This empowers engineers to create compact exchangers that are lighter, more energy-efficient, and more reliable, directly contributing to safer, high-performance aircraft systems. Dr. Ramanamurthy, Scientist 'F', Deputy Project Director, Aeronautical Development Agency, Bangalore.

Objectives of the Event

- To utilise CFD for evaluating thermal performance and optimising flow distribution within compact aircraft heat exchangers, ensuring efficient heat transfer.
- To analyse temperature gradients, cooling effectiveness, and thermal stresses using CFD to enhance operational reliability of aircraft control systems.
- To improve flow uniformity by identifying recirculation zones, maldistribution, and boundary-layer effects through high-fidelity CFD simulations.
- To incorporate predictive, simulation-driven design practices that replace traditional trial-and-error thermal modeling in aerospace applications.
- To integrate multidisciplinary knowledge from fluid dynamics, thermodynamics, and aerospace engineering for holistic evaluation of heat-exchanger performance.

Event Overview

The Department of Mechanical Engineering organized a three-day FDP on the topic “Thermal Performance and Flow Distribution of Compact Heat Exchangers for Aircraft Control Systems using CFD.” The resource person emphasised how the integration of CFD into heat-exchanger design marks a major shift from conventional trial-and-error approaches to a fully predictive, simulation-driven methodology. By enabling accurate evaluation of temperature gradients, flow uniformity, and pressure drop characteristics, CFD enhances the reliability of aircraft thermal-control components while significantly reducing development time and resource wastage.

The expert highlighted how CFD bridges advanced simulation with the growing demand for efficient aerospace thermal-management solutions. Through detailed modeling of coolant flow paths, turbulence behavior, and heat-transfer performance, engineers can optimise exchanger geometries, improve cooling strategies, and ensure stable operation under varying flight conditions. The session demonstrated how predictive simulation strengthens system durability, enhances operational safety, and supports innovation by minimising uncertainties in design. Participants gained valuable insight into incorporating CFD tools into real-world aerospace workflows—leading to improved product life cycles, reduced maintenance requirements, and more energy-efficient aircraft systems.

Significance:

The event reinforces the transition from traditional experimental methods to data-driven, simulation-based engineering, aligning with global goals of reliability, efficiency, and sustainable aerospace technology development.

Photographs from the Event

As shown in Figure 1, Dr. Ramanamurthy, Scientist ‘F’ and Deputy Project Director at the Aeronautical Development Agency, Bangalore, addressed the gathering and initiated the session by sharing valuable insights with the participants.

Figure 1: Session delivery by the resource person

Schedule of the Event

The Table 1 gives the schedule of the event. The session began with the introduction of the resource person by the Head of the Department followed by the overview of the guest lecture by Prof Vinoth Kumar G, Assistant Professor, Department of Mechanical Engineering.

Day	Time	Session Details
Wednesday, March 18, 2026	10.00 am - 10.10 am 10.10 am - 10.20 am	Welcome Address: Introduction about the Resource Person
Wednesday, March 18, 2026	10.30 am - 12.20 pm 12.20 pm - 12.30 pm	Session Delivery Vote of Thanks

Table 1: Schedule of the Event

Outcomes and Impact

- **Enhanced Understanding of CFD-Based Heat Exchanger Analysis:** Participants gained clarity on how CFD can be applied beyond basic flow simulation to evaluate heat-transfer performance, flow maldistribution, and pressure-drop characteristics in compact aerospace heat exchangers.
- **Integration of Efficiency-Driven Design Goals:** The session demonstrated how CFD-guided optimisation reduces thermal losses, improves cooling uniformity, and supports energy-efficient, sustainable thermal-management solutions for aircraft systems.
- **Skill Development in Simulation Interpretation:** Attendees acquired practical insights into connecting CFD results—such as temperature fields, velocity profiles, and turbulence effects with real-world heat-exchanger behavior and operational performance.
- **Cross-Disciplinary Relevance in Aerospace Engineering:** The discussions highlighted how CFD links fluid dynamics, heat-transfer physics, and system-level aerospace requirements, encouraging participants to adopt integrated approaches in solving thermal-control challenges.
- **Industry Readiness for Advanced Thermal Systems:** Exposure to case studies and modeling workflows prepared participants to apply CFD tools effectively in aerospace cooling systems, improving reliability, lowering maintenance costs, and meeting stringent aviation standards.

Conclusions

By demonstrating how CFD accurately captures flow distribution and thermal behavior within compact heat exchangers, this session establishes that simulation-driven assessment not only enhances cooling efficiency and operational safety but also improves resource utilisation, reduces thermal losses, and enables engineers to develop aerospace thermal-management solutions that are both technically robust and aligned with sustainable engineering practices.

Report by: Prof Vinoth Kumar G

Affiliation: Faculty in the Department of Mechanical Engineering, MVJ

College of Engineering

Session-6

Topic : CFD Applications in Electronic Cooling, Busbar Thermal Analysis, and Multiphase Flow.

Time: 17.03.2026 FN

Resource Person: Dr. Thejaraju

The Session 6 Day 3 "**CFD Applications in Electronic Cooling, Busbar Thermal Analysis, and Multiphase Flow**." Presented by **Mr. Tejaraju, Head of Modeling and Simulation, Fusion Minds,** Bangalore.

Computational Fluid Dynamics (CFD) plays a transformative role in driving innovation and excellence across diverse industry sectors by enabling engineers to analyse complex fluid flow and heat transfer phenomena with high accuracy. Through virtual simulation, CFD allows industries to explore multiple design alternatives, predict product performance, and identify potential issues early in the development cycle. This simulation-driven approach significantly reduces development time, cost, and dependence on physical prototypes while ensuring higher efficiency and reliability.

Across sectors such as automotive, aerospace, energy, manufacturing, electronics, healthcare, and consumer products, CFD supports optimized designs that enhance performance, safety, and sustainability. From improving vehicle aerodynamics and thermal comfort to enhancing cooling of electronic devices and optimising airflow in HVAC systems, CFD enables data-driven decision-making aligned with real-world operating conditions. As industries increasingly pursue energy efficiency and environmental responsibility, CFD becomes a critical tool in achieving regulatory compliance and sustainable innovation.

By integrating advanced computational techniques with practical engineering applications, CFD continues to elevate industry standards and foster technological excellence. Its widespread adoption empowers organisations to remain competitive in a rapidly evolving global landscape, fostering innovation not only in high-end industrial systems but also in everyday products that influence quality of life.

Objectives of the Event

- Introduce the fundamental concepts of Computational Fluid Dynamics (CFD) relevant to thermal and fluid flow analysis in engineering systems
- Develop understanding of CFD-based thermal management techniques used in electronic cooling applications

- Analyse heat dissipation, airflow patterns, and temperature distribution in electronic components and enclosures
- Study busbar thermal behavior under electrical loading conditions using CFD and coupled electrothermal analysis
- Evaluate temperature rise, current density effects, and cooling strategies for busbar systems to ensure safety and reliability
- Provide insight into multiphase flow phenomena such as liquid–gas, liquid–solid, and boiling flows encountered in industrial applications
- Demonstrate practical CFD modeling approaches for multiphase systems used in energy, chemical, and process industries
- Bridge theoretical concepts with real-world industrial case studies and simulation practices
- Enhance participants' ability to apply CFD tools for design optimization, performance improvement, and failure prevention
- Encourage the integration of advanced CFD applications into teaching, research, and industry-oriented problem solving

Event Overview

The Faculty Development Program (FDP) on *Computational Fluid Dynamics (CFD): Driving Innovation and Excellence for All Industry Sectors* is structured to provide a comprehensive understanding of CFD and its practical relevance in engineering applications. The event focuses on explaining how numerical simulations help analyse fluid flow and heat transfer phenomena encountered in everyday products and industrial systems. By blending theoretical foundations with application-oriented discussions, the program emphasises the growing importance of CFD as a core tool in modern product design and development.

The event covers real-world applications of CFD in sectors such as consumer appliances, automotive and transportation, HVAC systems, electronics cooling, energy systems, and healthcare devices. Through expert lectures, case studies, and practical insights, participants gain exposure to simulation-driven problem-solving adopted by the industry. Overall, the FDP serves as a platform for knowledge enhancement, academic enrichment, and alignment of teaching and research practices with industry standards and emerging technological needs.

Photographs from the Event

As shown in Figure 1, Dr. Shrinivas L Gombi, the head of the Department of Mechanical Engineering, welcomed the chief guest and the participants to the guest lecture.

Figure 1, Dr. Shrinivas L. Gombi formally welcomed the Chief Guest

Figure 2: Session delivery by the resource person

Figure 3 shows the photographs of the students listening eagerly to the resource person.

b. Mr Irfan Khan delivering the vote of thanks

Schedule of the Event

Day	Time	Session Details
Wednesday, March 18, 2026	1.00 pm - 1.10 pm 1.10 pm - 1.20 pm	Welcome Address: Introduction about the Resource Person
Wednesday, March 18, 2026	1.20 pm - 3.20 pm 3.20 pm - 3.30 pm	Session Delivery Vote of Thanks

Table 1: Schedule of the Event

Table 1 gives the schedule of the event. The session began with the introduction of the resource person by the Head of the Department, followed by the overview of the guest lecture by Mr. Irfan Khan, Assistant Professor, Department of Mechanical Engineering

Outcomes and Impact

The event successfully enhanced participants' awareness and understanding of the pivotal role Computational Fluid Dynamics (CFD) plays in driving innovation across diverse industry sectors. Participants gained practical insights into how CFD supports simulation-driven design, enabling accurate prediction of fluid flow, heat transfer, and performance characteristics in real-world products. This understanding reinforced the importance of CFD as a decision-making and optimization tool in modern engineering practice.

The program strengthened participants' ability to connect theoretical concepts with industrial applications in sectors such as consumer products, automotive systems, energy, HVAC, electronics cooling, and healthcare technologies. Through application-focused discussions and case studies, attendees developed a broader perspective on improving product efficiency, reliability, sustainability, and cost-effectiveness using CFD-based approaches. Overall, the event fostered academic and professional growth by equipping participants with industry-relevant knowledge and confidence to integrate CFD concepts into teaching, research, and consultancy. It motivated faculty members to adopt advanced simulation techniques, encourage innovation in problem solving, and contribute to excellence in engineering education and industrial collaboration across multiple sectors.

Conclusions

The event successfully enhanced participants' awareness and understanding of the pivotal role Computational Fluid Dynamics (CFD) plays in driving innovation across diverse industry sectors. Participants gained practical insights into how CFD supports simulation-driven design, enabling accurate prediction of fluid flow, heat transfer, and performance characteristics in real-world products. This understanding reinforced the importance of CFD as a decision-making and optimisation tool in modern engineering practice.

Overall, the event fostered academic and professional growth by equipping participants with industry-relevant knowledge and confidence to integrate CFD concepts into teaching, research, and consultancy. It motivated faculty members to adopt advanced simulation techniques, encourage innovation in problem solving, and contribute to excellence in engineering education and industrial collaboration across multiple sectors.

Report by: Mr. Irfan Khan
Affiliation: Faculty in the Department of Mechanical Engineering,
MVJ College of Engineering