Niharika
Institute of Computational Engineering

"Knowing is not enough; we must apply "

"Learn CFD analysis with complex Industry Projects"

ANSYS® FLUENT® CFX®

Temperature

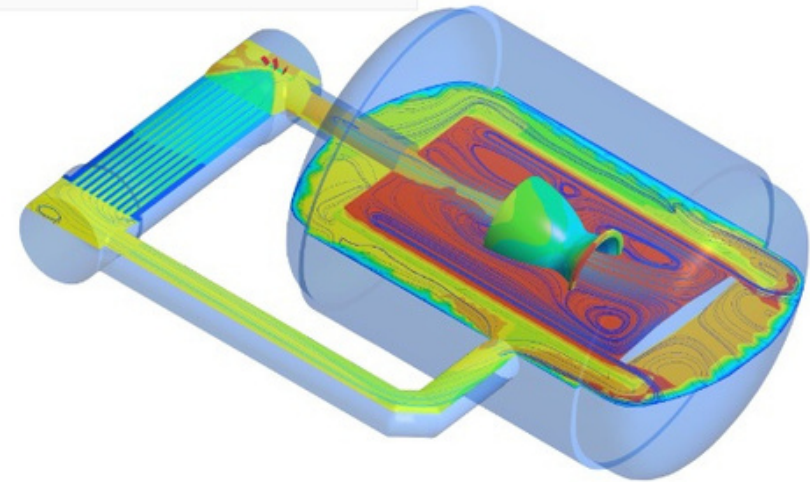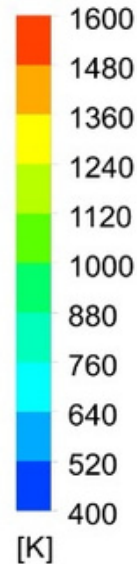|	
	1600
	1480
	1360
	1240
	1120
	1000
	880
	760
	640
	520
	400

[K]

Image courtesy: NCES Pvt Ltd

Contact us

Ground Floor, JC Plaza, #35, 42nd Cross, Off 12th Main, 3rd Block, Rajajinagar, Bengaluru. Karnataka - 560010, India
Mobile: **+91 9916266179**
Email: **rudresh@nicecfd.com**

Certification course on CFD analysis - Ansys Fluent, CFX

Join Now

www.nicecfd.com

About Us

Started in the year 2010, we are the pioneers in Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA) technologies.
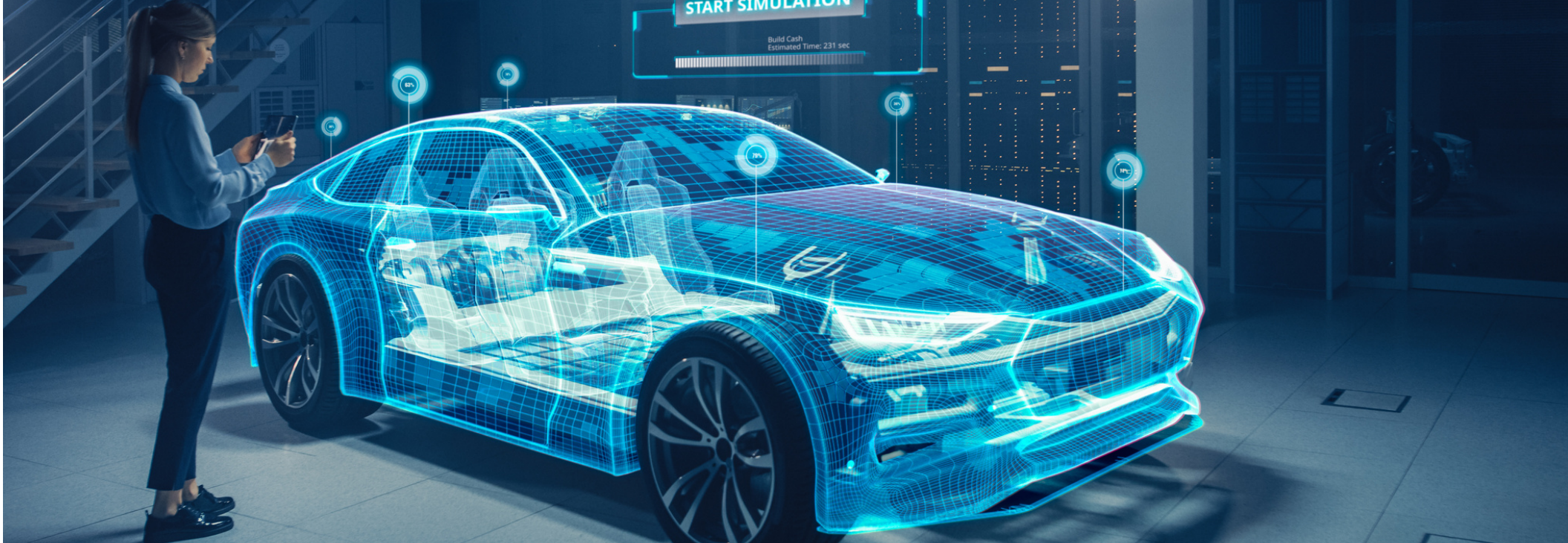
We offer online and offline training programs on CFD and FEA. Our courses are completely Industry oriented and till date we have trained more than 5000 Engineers in these niche engineering technologies.

Syllabus

- Computational Fluid Dynamics (CFD)
- Fluid Dynamics, Gas Dynamics
- Heat Transfer
- Aerodynamics
- Training on Meshing tools
- CFD analysis projects using ANSYS Fluent, CFX (Genuine, Licensed version)

Be a potential Engineer

- Always look beyond Tutorials and simple projects
- Gain thorough theoretical knowledge and start with complex Industry projects
- Industry projects are always complex and challenging, solving them will enable you to create good Impression in Interview and chances of selection shall be high

What is CFD analysis ?

CFD analysis is a technology which uses computer based numerical simulation methods, to solve the engineering problems Involving Fluid flow, Heat transfer, Combustion and Multi phase flows.

CFD is extensively used to validate engineering designs (In lieu of time consuming prototype tests) in almost all Industry verticals covering Automotive, Aerospace, Chemical, Industrial Engineering , Medical devices , Semiconductor etc.

Taught with Licensed versions of ANSYS Fluent, ANSYS CFX software

What Skills Will You Learn?

In this course you will learn

- Basics of CFD, mathematical background of CFD, Finite Volume method, modelling of boundary layer and Turbulence modelling.
- Essential Theoretical Fundamentals covering Fluid Dynamics, Gas dynamics, heat transfer and Aerodynamics from Industrial application point of view.
- Meshing skills
- Solve 4 to 5 complex Industry problems using ANSYS Fluent , CFX CFD analysis software.

Who Should Take This Course?

Pre requisites for the course

- Bachelor's / Master's degree in Mechanical/ Aero/Chemical/Industrial Engineering stream with exposure to subjects like Fluid Mechanics, Thermo dynamics, heat transfer etc.
- Attitude to learn and excel in career is the most desirable pre requisite at Niharika Institute of Computational Engineering

Highlights

- **Strong focus on theoretical fundamentals**

- **Industry projects**

- **Taught with Licensed Software**

- **Faculty with 20 years Industry experience**

- **Transparent and ethical business practices**

Main Faculty

Mr.Rudresh Ganganna
Former Scientist
Thermal Design and Analysis Division
Indian Space Research Organization
Bangalore, India

For more details refer our website

www.nicecfd.com

Cost: 53,100 Indian Rupees

(Inclusive of all Taxes)

☑ ENROLL
☑ EDUCATE
☑ SUCCESS