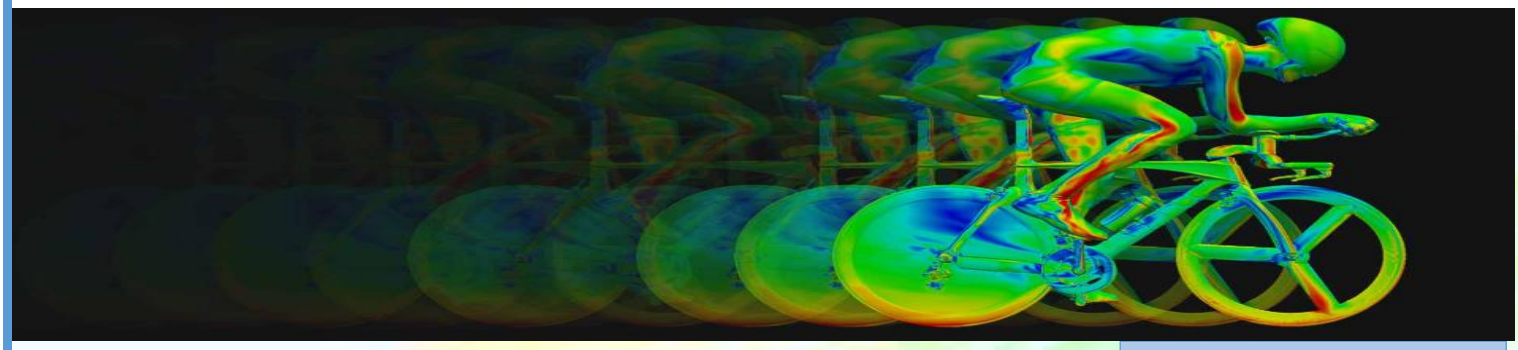
CAE FlowXplore

Let's create value to your design

TRAINING BROCHURE

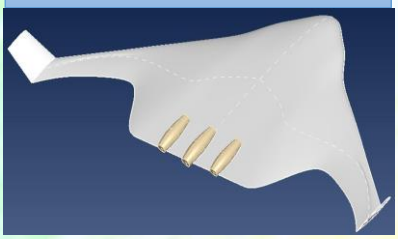


What is CFD?

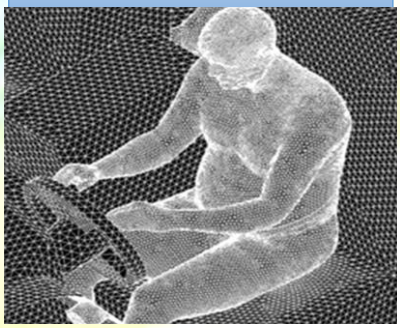
Computational Fluid Dynamics or Simply CFD is concerned with obtaining numerical solution to fluid flow, thermal problems using computer. It is a ***virtual validation technology*** that forms an alternate to "PROTO-BASES EXPERIMENTAL TECHNOLOGY". The advent of high speed and large memory computers has enabled CFD to obtain solutions to many flow problems including those that compressible or incompressible, laminar or turbulent, chemically reacting or non-reacting

Computational Fluid Dynamics constitutes a new "third approach" in philosophical study and development of the whole discipline of Fluid Dynamics. In 17th century, the foundations for experimental CFD were laid. The 18th and 19th century saw the gradual development of theoretical Fluid Dynamics. As a result, throughout most of 20th century, the study practice of Fluid Dynamics (indeed, all physical science and engineering) involved the use of pure theory on one hand and pure experiments on the other hand.

CAD



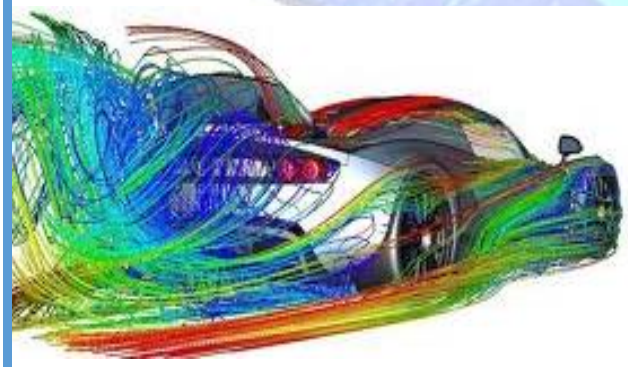
Mesh

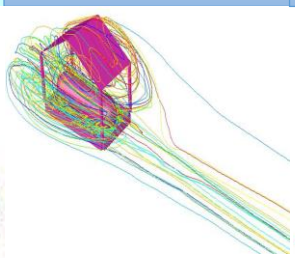


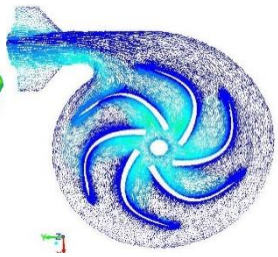
Need of R&D

- **Enhancing product quality**
- **Reducing Design time & Design Cost**
- **Getting the Insight & foresight**

Path lines | Contours | Vectors







#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

Methodologies to approach a fluid problem

EFD- Experimental fluid dynamics

TFD- Theoretical fluid dynamics

CFD- Computational fluid dynamics

EFD

PROCESSS

-protype model

-set the flow area

-providing the initial conditions

-obtain results

ADVANTAGES

-deals with the actual physical system

-desiered quantity is measured with instruments

DIS ADVANTAGES

-expensive

-time consuming

-loss of insight and foresight

-impossible for complex models

-single purposed

-sequential

TFD

Deriving the exact, continuous mathematical solution

ADVANTAGES

-Fast and inexpensive.

-Results obtained are subject to the accuracy of the assumptions, approximations, and idealizations made in the analysis.

DIS ADVANTAGES

-Meant for only simple problems.

-Manually finding the solution of equation is complex job.

-Assumption made in illogic manner, leads to entire analysis to violate the nature of the problem

experiment

analytical

Why CFD?

- ✓ Insight and fore sight
- ✓ Can capture complex physics
- ✓ Multi-purpose
- ✓ Quality
- ✓ Reduce the time consumption

Overcomes the difficulties and disadvantages of both Experimental and analytical method

To find solution for a physics ???

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

How CFD works

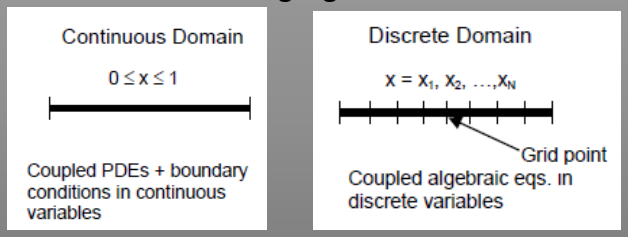
Physics

Problem identification

Mathematical representation of fluid physics in PDE

- ✓ Continuity equation
- ✓ N-S equation
- ✓ Energy equation

Replacing the continuous problem domain with a discrete domain using a grid.



Converting the PDE into Algebraic equations

Discretization

- ✓ Finite difference method
- ✓ Finite volume method

Numerical methods

- ✓ Gauss seidel
- ✓ Gauss Jordan
- ✓ Gauss elimination

Digital computers

- ✓ Solving iteratively
- ✓ Interpreting the results

PDE form (continous representation)

$$\frac{du}{dx} + u^m = 0;\ 0 \le x \le 1;\ u(0) = 1$$

ALGEBRAIC form (discrete representation using **Taylors Expansion)**

$$\left(\frac{du}{dx}\right)_i = \frac{u_i - u_{i-1}}{\Delta x} + O(\Delta x^2)$$

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

Governing equations

Conservation of mass (Continuity Equation)

$$\partial\rho/\partial t+\text{div}(\rho u) =0$$

Conservation of momentum (N-S Equation)

X-momentum

$$\partial(\rho u)/\partial t+\text{div}(\rho uu)= -\partial p/\partial x+\text{div}(\mu \text{ grad } u) +SMx$$

Y-momentum

$$\partial(\rho v)/\partial t+\text{div}(\rho vu)= -\partial p/\partial y+\text{div}(\mu \text{ grad } v) +SMy$$

Z-momentum

$$\partial(\rho w)/\partial t+\text{div}(\rho wu)= -\partial p/\partial z+\text{div}(\mu \text{ grad } w) +SMz$$

Conservation of Energy

$$\partial(\rho i)/\partial t+\text{div}(\rho iu)= -p \text{ div } u+\text{div}(k \text{ grad } T) +Si + ø$$

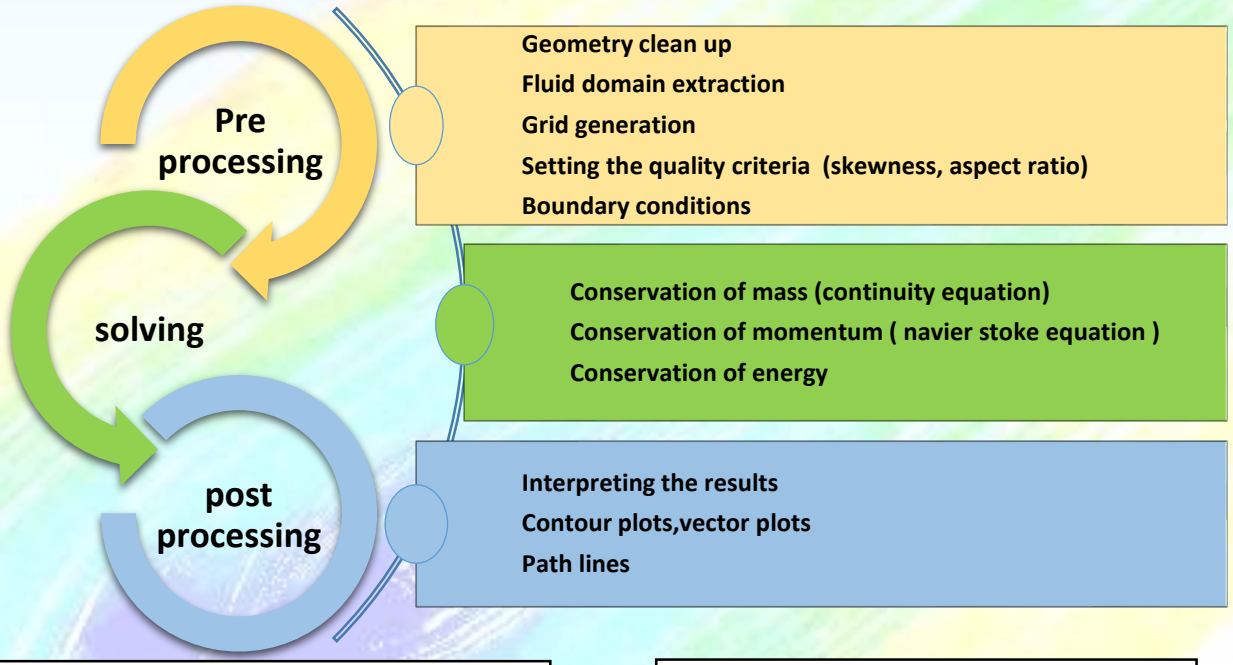
ρ- Fluid density (kg/m^3), p- Pressure (Pa)

U, V, W –fluid velocity (m/s), T-temperature (K), K-thermal conductivity (W/m-K)

μ - Dynamic viscosity (Ns/m^2) S_{Mx}, S_{My}, S_{Mz} – momentum source / momentum sink term

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

Methodology of CFD



Equiangular skew

It is a non-dimensional parameter calculated using the normalized angle deviation method, and is defined as,

$$\max\left[\frac{q_{\max}-q_e}{180-q_e}, \frac{q_e-q_{\min}}{q_e}\right]$$

Where,

$q_{\max}$ = largest angle in the face or cell

$q_{\min}$ = smallest angle in the face or cell

q_e = angle for an equiangular face or cell (e.g., 60 for a triangle and 90 for a square)

Equivolume skew

It is a non-dimensional parameter calculated using the volume deviation method, and is defined as,

$$\frac{\text{optimal-cell-size} - \text{cell-size}}{\text{optimal-cell-size}}$$

A value of 0 indicates a best case equilateral cell and a value of 1 indicates a completely degenerate cell.

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

Applications of CFD

Over the past few years CFD has become a vital tool in many industries. Because of its ability to predict flow features correctly, it has become a regular activity in design and R&D departments of many organisations.

Following are the typical applications of CFD,

Aerospace

Combustion – External Aerodynamics – Inlets and Nozzles Instrumentation – Missile systems - Propulsion

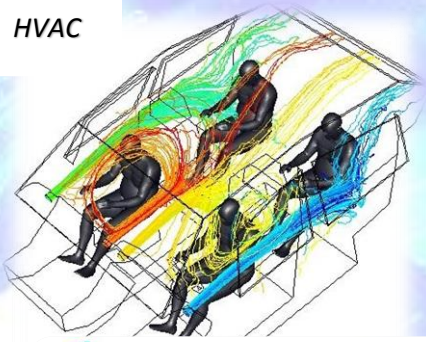
Automotive

External Aerodynamics – HVAC – Engine Cooling – Inlet and Exhaust manifolds – Under– hood Analysis – Brake Cooling – In-cylinder combustion – Pollution control.

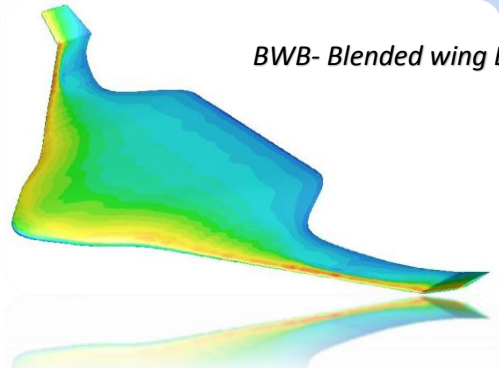
Turbo machinery

Axial and Centrifugal Pumps – Fans and Blowers – Gas and Steam Turbines – Wind Turbines – Cavitation analysis.

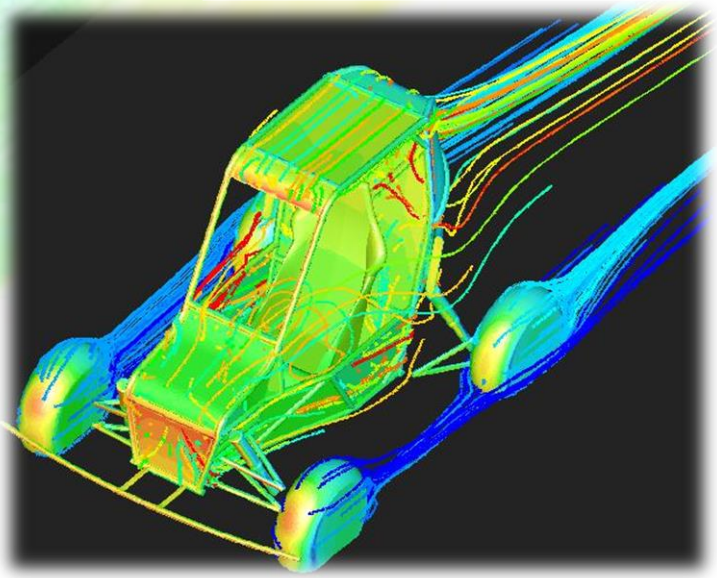
HVAC



BWB- Blended wing Body



External aerodynamics –SAE BAJA



#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

 # FXTR-CFD 005

Heat Exchangers

Shell and Tube Heat Exchangers - Heat Pipes and Pumps - Spiral and Compact Heat Exchangers - Energy Recovery Systems

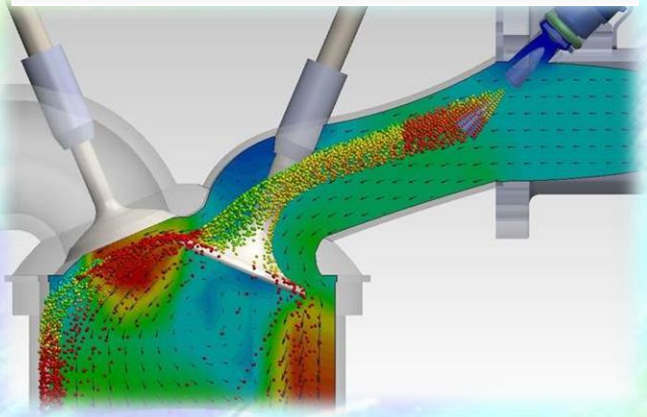
Chemical

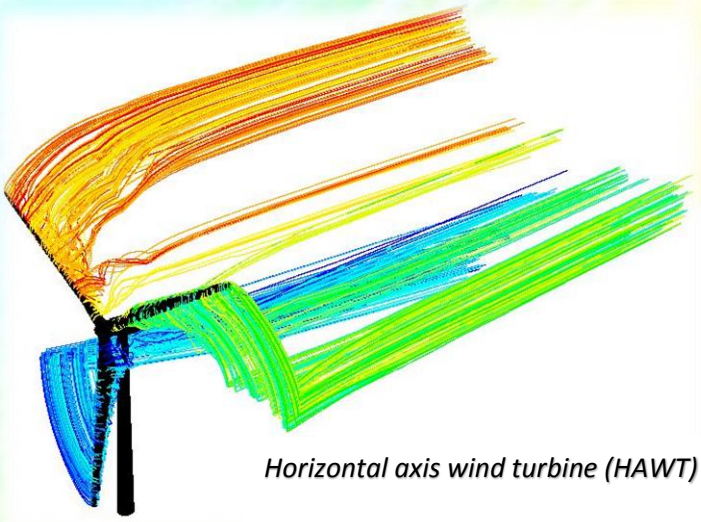
Drying technology – Filtration - Heat and Mass Transfer - Mixing simulation - Chemical reactors

Bio-medical

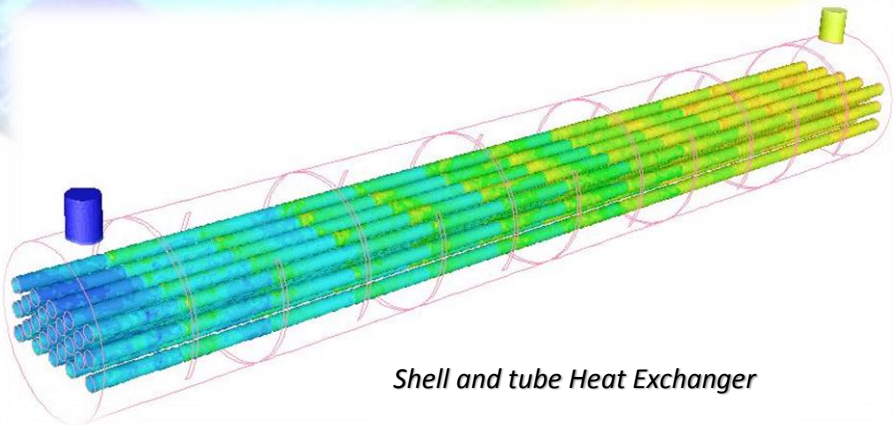
Blood handling equipments - Surgical analysis - Sterilization equipment - Blood separators

Four stroke engine





Horizontal axis wind turbine (HAWT)



Shell and tube Heat Exchanger

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

Flow Control

Control Valves – Medical Equipments – Oil and Gas – Manifolds – HVAC systems

Electronics and Semi-conductors

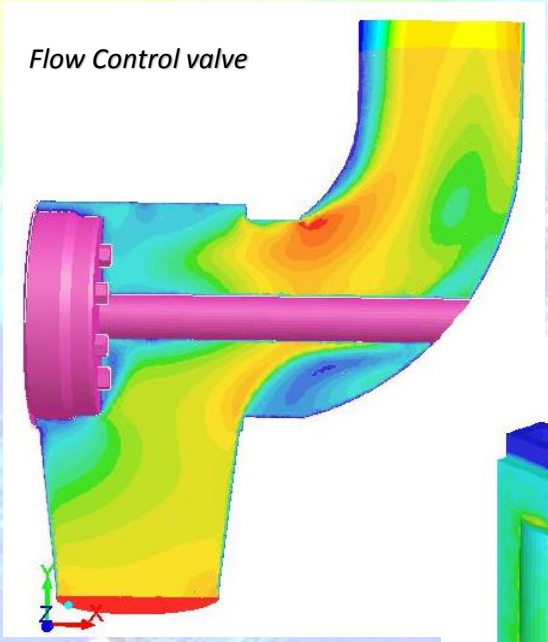
CPU cabinet cooling - Chip cooling system - Heat sink design - Cooling fan design –

Heating and Refrigeration

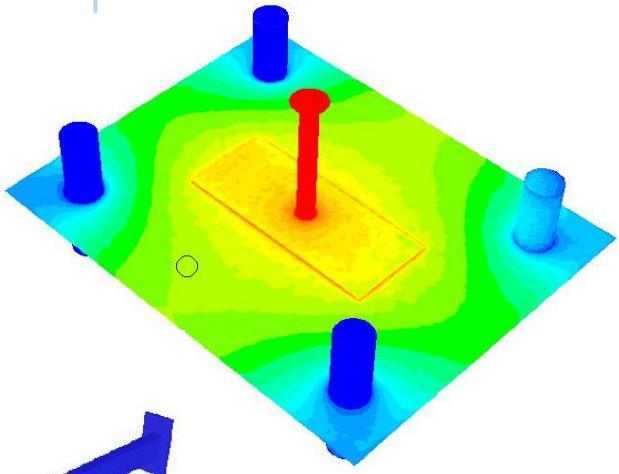
Air flow around buildings - Fan noise - Environmental control systems - Heating system design - Ducts, Hoods, Fittings, Valves - Room flow distribution

Heat Transfer Analysis

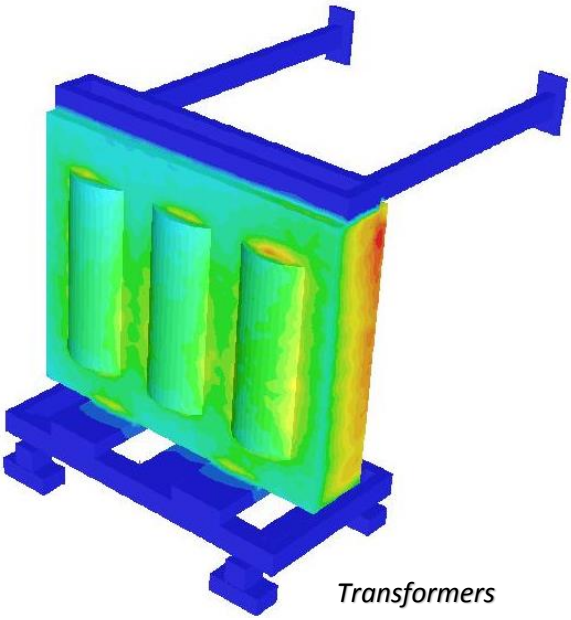
Conductive Heat transfer – Convective Heat Transfer – Radiative and Mixed Heat transfer.



Flow Control valve



Injection molding



Transformers

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

Career in CFD

India is identified as the world's fastest growing technical hub, and many more have started their CFD design centre. The main stumbling block today is not technology but the skilled professional who can make use of the technology to solve industrial problems. CFD industry is lacking quality man power. DACFD is a unique professional which bridges the gap between YOU and CFD industry.

CFD opportunities in Industries

OEMs

Airbus Engineering Centre India
Allied Cosmic Technologies

BOSCH
Cummins in India
Delphi
EADS (Airbus Engineering Centre India)
GE Energy
General Motors
Honeywell
Intel (Electronics cooling)
Rolls Royce Operations India Pvt Ltd
SAFRAN Aerospace India Pvt Ltd (SNECMA)

Government Establishments

ADE
BARC
CFEES (Centre for Fire, Environment & Explosive Safety)
DRDL
DRDO
GTRE
ISRO
NAL
NPCIL

Consulting and Service organisations

CADES
CCTech
EDS
HCL
Infosys
InfoTech
Mahindra Engineering Services (MES)
Quest
Satyam
Tata Technologies Ltd (TTL),
TCE
TCS
Transoft

CFD codes and Services Industries

ACRi InfoTech Pvt Ltd (www.acricfd.com)
ALTAIR Engineering
CD-ADAPCO
COMSOL India
CSM (Start CD)
EA Technologies (CFX ANSYS)
ESI Group
Flomerics
Fluent (FLUENT/ANSYS)
NFOTEC (Numaca-India)

Course Brochure

Is the course suitable for you?

Are you an UG/PG graduate or working in an engineering firm?

Are you interested in Analysis?

Do you want to shift your professional as an Analyst?

*Would you like to be a part of fastest growing CAE INDUSTRY?*if the answer for any of the above question is YES, you're the right candidate for the course.*

Salient Features of the course?

- ✓ *This is a Certified Course*
- ✓ *Well-designed course with theoretical foundation and software training*
- ✓ *Course involves industry standard problems*
- ✓ *Continuous assessment throughout the course*
- ✓ *Direct placement for top rankers*
- ✓ *Course material*

Fluid Dynamics

Introduction and Basic concepts

- Introduction
- No-slip condition
- Classification of fluid flows
- System and control volume
- Properties of fluids
- Density and Specific gravity
- Velocity and Pressure
- Viscosity and Vortices
- Non – Dimensional numbers

Fluid Kinematics

Introduction and Basic concepts

- Langragian and Eulerian approach
- Streamlines, Streaklines, Timelines and Path lines
- Profile plots, contour plot and vector plot
- Reynolds Transport Theorem

Fluid Flow equations

- Mass, Bernoulli and Energy equations
- Momentum analysis of flow systems
- Navier-stokes equation
- Dimensional and differential analysis of fluid flow
- Internal and External flow
- Laminar and turbulent
- Drag and Lift
- Flow separation

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

Heat Transfer

- Heat and other forms of energy
- 1D heat conduction equations and general heat conduction equations
- Boundary and initial conditions
- Steady and transient heat conduction
- Heat transfer by convection
- Velocity and Thermal Boundary layer
- External forced convection, Internal forced convection and natural convection
- Heat transfer radiation Conjugate Heat Transfer (CHT)

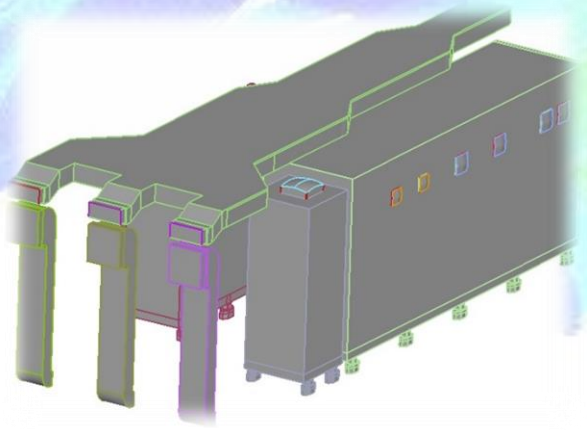
Pre processing

Geometric modelling

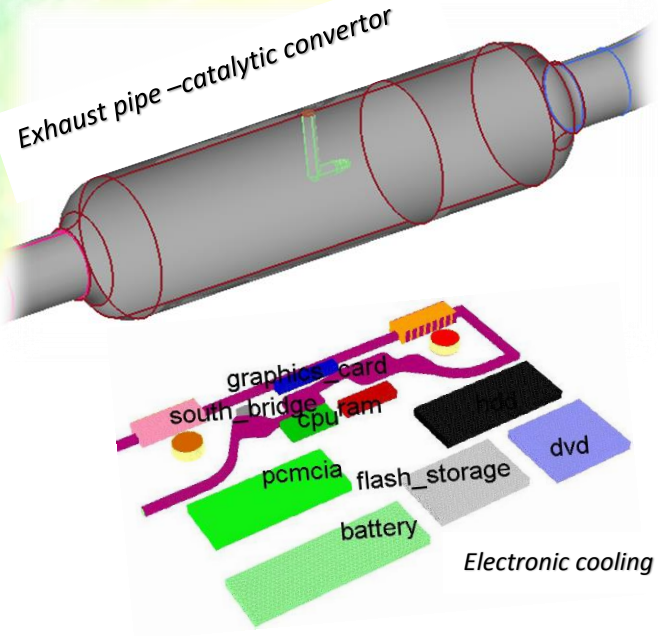
- Geometric transformations
- File formats and translators
- Concept of topology
- Surface modelling
- Faceted models
- Solid modelling

Fluid domain extractions

- Creation of water tight geometry
- Faceted Boolean operations
- Dependent and independent CAD errors
- Healing the geometry using surface knitting
- Surface projection methods



Radar transport system



Electronic cooling

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

Pre processing

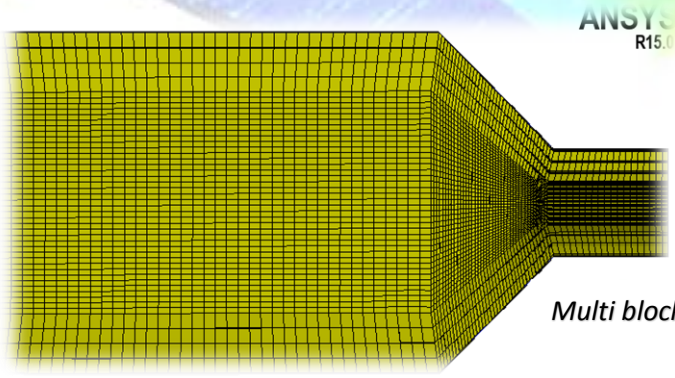
Structured grid generation

- Linear interpolation
- Transfinite interpolations
- Laplace grid generations
- Poisson grid generations
- Map and sub map
- Sweeping and surface grid generations
- Mono block and multi block
- Hierarchical multi block
- Moving and sliding multi block
- Grid clustering and grid enhancement

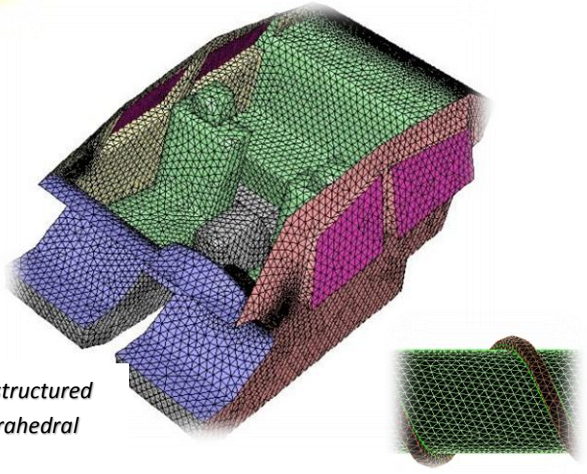
Pre processing

Unstructured grid generation

- Basic theory of unstructured grid generation
- Advancing front
- Delaunay triangulation and various points insertion methods
- Un-structured quad and Hex generation
- Grid based methods
- Quad tree and Octree based methods
- Concept of medial axis and medial surface
- Various models in unstructured grids
- Surface mesh generation
- Surface mesh repair
- Volume grid generation
- Volume mesh improvement
- Mesh smoothing algorithms
- Grid clustering and quality checks for volume mesh



Multi blocking



Unstructured Tetrahedral

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

 # FXTR-CFD 005

CFD Solver

Introduction to CFD

Philosophy of CFD

- Governing equations of fluid dynamics and their physical meaning
- Mathematical behaviour of governing equations and their impact
- Simple CFD techniques

Numerical methods in CFD

- Finite Difference method
- Finite Volume method
- Upwind and Downwind schemes
- Implicit and Explicit approach
- Convergence and stability

Visualization: Post Processing

- Contour plot, Vector plot and scatter plot
- Shaded and Transparent surfaces
- Particle and Path-line trajectories
- Iso-surface, Animation and Movies
- Exploration and Analysis of data

Advanced Modules

Turbulence modelling

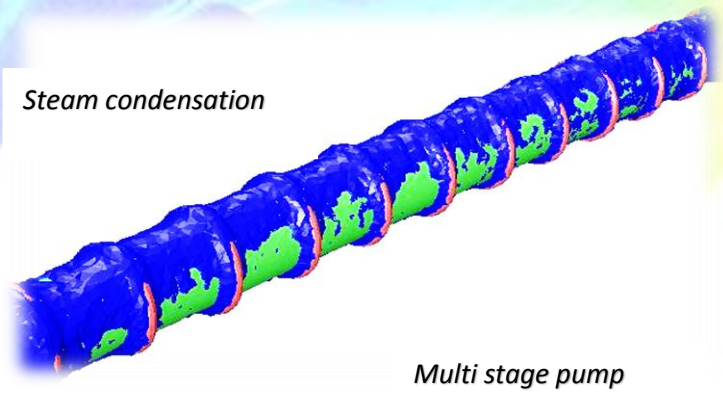
- Theory of Turbulence modelling
- Closure problem
- Reynolds Averaged Navier Stokes Equation (RANS)
- Eddy viscosity model
- Algebraic model
- One and two equation models
- Near wall treatment
- Large Eddy Simulation (LES)
- Direct Numerical Simulation (DNS)

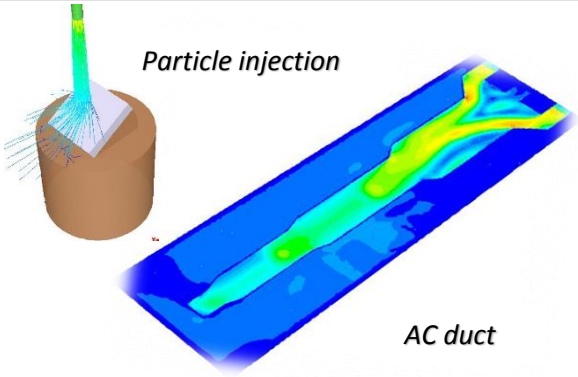
Multi – Phase Modelling

- Fundamentals of multi – phase flows
- Eulerian – Lagragian approach (ELAG)
- Eulerian – Eulerian approach (E2P)
- Volume – of – Fraction (VOF)
- Problem solving

Conjugate Heat Transfer

- Introduction to CHT
- Fluid Boundary Condition
- CHT solid boundary conditions
- CHT interface conditions





#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

Industry Specific Modules

Turbo – Machinery

- Introduction to Turbo – machinery terminology
- Quasi steady rotor – stator interaction
- CFD study of rotor – stator interaction
- CFD simulation of Turbo – machinery problems

Aerospace Simulation

- Introduction to various terminologies in Aerodynamics
- Computation of Drag, lift and moment coefficient
- Supersonic and subsonic simulation
- Lift and drag on aeroplanes, missiles and rocket

Automobile Simulation

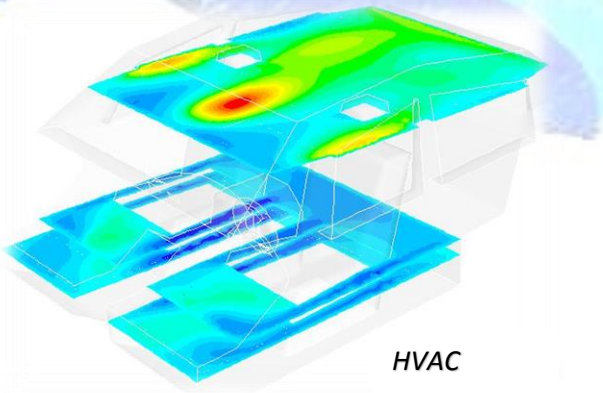
- CFD analysis of Ahmed Body
- Transient and steady state analysis
- Under – hood analysis
- External flow
- HVAC for car and passenger comfort

Pre – processing

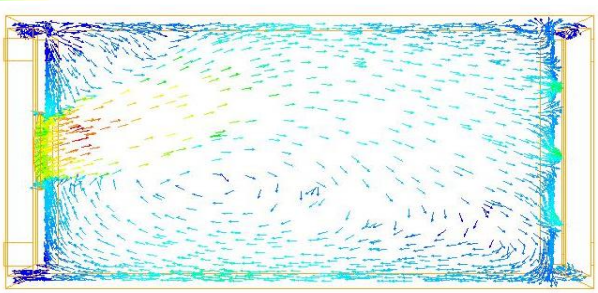
- **ANSYS – ICEM CFD**
- **ANSA**
- **Hypermesh**
- **Gambit**
- **ANSYS T – Grid**

Solver & post processing

- ✓ **Fluent**
- ✓ **CFX**
- ✓ **Star – CCM+**
- ✓ **CFD post**



HVAC



Fan flow inside a panel

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

Why FlowXplore?

As one of the seeds of IIT, Mumbai we distinguish ourselves as a team of FlowXplore with Professional Engineers has unique vision in taking the simulation industry into the future by making high level design technology available and affordable to every Industry and Individual.

Our Vision is implemented by applying principals that,

- ✓ **Focusing more on fundamentals of fluid dynamics and heat transfer theories**
- ✓ **Enabling students to work on real world industrial projects**
- ✓ **Training students in multiple commercial software packages**
- ✓ **Proving an opportunity for students to get into IIT, NIT as SRF & JRF**
- ✓ **Innovative Simulating Techniques, Responsive Technical support, Comprehensive Documentation and Professional Training.**

PLACEMENTS

- ✓ Placed around sixty students in CFD and FEA in several MNCs which is highly meant for from IITs / NITs

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com

Recent Placements

Prabhukumar	IIT,Chennai- Ph.D. Scholar
Anantha Narayanan	IIT, Chennai - Jr. Research Assistant
Karthik	IIT, Chennai - Sr. Research Fellow
Anandh Raj	ISRO
Sugumaran	Simgrosis
Arun	Cummins India.
Fedrick	Cummins India.
Nijanthan	Cummins India.
Prasana Ramanan	Cummins India.
Rubesh	Cummins India.
Tamilselvan	Cummins India.
Boopathi	Daimler Chrysler
Saravanan	Daimler Chrysler
Mani Kandan	Daimler Chrysler
Athiannan	Daimler Chrysler
Suganth	CAE Analyser Solutions
Thennarasu	CAE Analyser Solutions
Venketesh	CAE Analyser Solutions
Hariharan	CAE Analyser Solutions
Hari Kribahar	CAE Analyser Solutions
Joe Lawerence	CAE Analyser Solutions
Rajesh	CAE Analyser Solutions
Gokula Krishnan	CAE Analyser Solutions
Muthu Kumar	Renault Nissan

#1054/18-1, 2nd Floor, Gowthem Centre Annex,(Behind Jet Airways Office), Avanashi Road, Coimbatore, TN-641 018
Ph: +91-9952205914 E mail: info@flowxplore.com ; consulting@flowxplore.com