Computational Fluid Dynamics

CHAITANYA GAURKAR
Student,M. Tech.(CAD/CAM)
(Production Engineering Department,
Shri Guru Gobind Singhji Institute of Engineering and Technology, Vishnupuri, Nanded)

ABSTRACT

Computational Fluid Dynamics (CFD) is a powerful simulation tool to predict flow patterns, pressures, temperatures and concentrations in a vast range of applications. It can simulate behavior with the right fluids at the right scale and operating conditions. Unlike correlations and "black box" models, it gives a full 3-D view inside or around your equipment. Because it is a simulation technology, it is safe, clean and nearly always cheaper and faster than experimentation**.**

As experimental investigation is becoming costlier and computer resources are increasing, industries have started adapting numerical analysis as an alternate approach for their Research and Development (R&D). With this approach industries can carry out analysis on virtual prototypes saving project time and cost significantly. Many of the Industrial applications involve fluid dynamics and CFD is the computational tool which helps engineers to carry out numerical analysis.

1. INTRODUCTION

Recent advancements in CFD include Computational Aero Acoustics (CAA), Computational Electro Magnetism (CEM), and Fluid Structure Interaction (FSI). Due to this, it has a wide applicability in different engineering domains.

Computational Fluid Dynamics (CFD) is the science of determining a numerical solution to the governing equations of fluid flow whilst advancing the solution through space and time to obtain a numerical description of the complete flow field of interest. As a developing science, CFD has received extensive attention throughout the international community since the advent of the digital computer. The attraction of the subject is twofold. Firstly, the desire to be able to model physical fluid phenomena that cannot be easily simulated or measured with a physical experiment, for example weather systems or hypersonic aerospace vehicles. Secondly, the desire to be able to investigate physical fluid systems more cost effectively and more rapidly than with experimental procedures.

There has been considerable growth in the development and application of CFD to all aspects of fluid dynamics. In design and development, CFD programs are now considered to be standard numerical tools, widely utilized within industry. As a consequence there is a considerable demand for specialists in the subject, to apply and develop CFD methods throughout engineering companies and research organizations. The current status of CFD within industry may be likened to that of structural analysis a decade ago, when it too was rapidly maturing. At that time the typical company marketing Finite Element structural analysis programs had a turnover an order of magnitude greater than the largest CFD vendor. Finite Element programs are now considered to be part of the routine design and analysis cycle within industry and are available on almost every computing platform. In a similar manner Computational Fluid Dynamics has become a standard industry tool and is now finding its place alongside CAD and FE packages. CFD is utilized as a design analysis tool within both industry and research organizations. The course provides a solid background for graduates to be able to apply, in an educated manner, CFD as a design tool for engineering applications.

Radiator is one of the most important operating components of our automobile. Responsible for keeping our automobile's engine at a safe operating temperature, a malfunctioning automotive radiator could mean big trouble for us if we do not seek professional mechanical attention immediately. Auto radiator and truck radiator problems can often develop without we even knowing it, with small particles of dirt and rust clogging up the essential elements, preventing our car radiator from being able to cool our engine properly. If this happens, our vehicle will over heat, potentially leaving us stranded.

2. Governing equations of Computational Fluid Dynamics

The physical aspects of any fluid flow are governed by the following three fundamental principles: Continuity, Momentum and Energy equations.

(1) Mass is conserved;

(2) $F =$ ma (Newton's second law); and

(3) Energy is conserved.

These fundamental principles can be expressed in terms of mathematical equations, which in their most general form are usually partial differential equations. Computational fluid dynamics is, in part, the art of replacing the governing partial differential equations of fluid flow with *numbers*, and advancing these numbers in space and/or time to obtain a final numerical description of the complete flow field of interest. This is not an all-inclusive definition of CFD; there are some problems which allow the immediate solution of the flow field without advancing in time or space, and there are some applications which involve integral equations rather than partial differential equations. In any event, all such problems involve the manipulation of, and the solution for, *numbers*.

The end product of CFD is indeed a collection of numbers, in contrast to a closed-form analytical solution.

CFD solutions generally require the repetitive manipulation of thousands, or even millions, of numbers—a task that is humanly impossible without the aid of a computer. Therefore, advances in CFD, and its application to problems of more and more detail and sophistication, are intimately related to advances in computer hardware, particularly in regard to storage and execution speed.

Continuity equations

(Non-conservation form)

$$\frac{D\rho}{Dt} + \rho\nabla \cdot \vec{V} = 0$$

Conservation form

$$\frac{\partial\rho}{\partial t} + \nabla \cdot (\rho\vec{V}) = 0$$

Momentum equations

(Non-conservation form)

$$x\text{-component}: \quad \rho\frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{\mathrm{xx}}}{\partial x} + \frac{\partial \tau_{\mathrm{yx}}}{\partial y} + \frac{\partial \tau_{\mathrm{zx}}}{\partial z} + \rho f_{\mathrm{x}}$$

$$y\text{-component}: \quad \rho\frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{\mathrm{xy}}}{\partial x} + \frac{\partial \tau_{\mathrm{yy}}}{\partial y} + \frac{\partial \tau_{\mathrm{zy}}}{\partial z} + \rho f_{\mathrm{y}}$$

$$z\text{-component}: \quad \rho\frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{\mathrm{xz}}}{\partial x} + \frac{\partial \tau_{\mathrm{yz}}}{\partial y} + \frac{\partial \tau_{\mathrm{zz}}}{\partial z} + \rho f_{\mathrm{z}}$$

Conservation form

$$x\text{-component}: \quad \frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u\vec{V}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{\mathrm{xx}}}{\partial x} + \frac{\partial \tau_{\mathrm{yx}}}{\partial y} + \frac{\partial \tau_{\mathrm{zx}}}{\partial z} + \rho f_{\mathrm{x}}$$

$$y\text{-component}: \quad \frac{\partial(\rho v)}{\partial t} + \nabla \cdot (\rho v\vec{V}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{\mathrm{xy}}}{\partial x} + \frac{\partial \tau_{\mathrm{yy}}}{\partial y} + \frac{\partial \tau_{\mathrm{zy}}}{\partial z} + \rho f_{\mathrm{y}}$$

$$z\text{-component}: \quad \frac{\partial(\rho w)}{\partial t} + \nabla \cdot (\rho w\vec{V}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{\mathrm{xz}}}{\partial x} + \frac{\partial \tau_{\mathrm{yz}}}{\partial y} + \frac{\partial \tau_{\mathrm{zz}}}{\partial z} + \rho f_{\mathrm{z}}$$

Energy equation

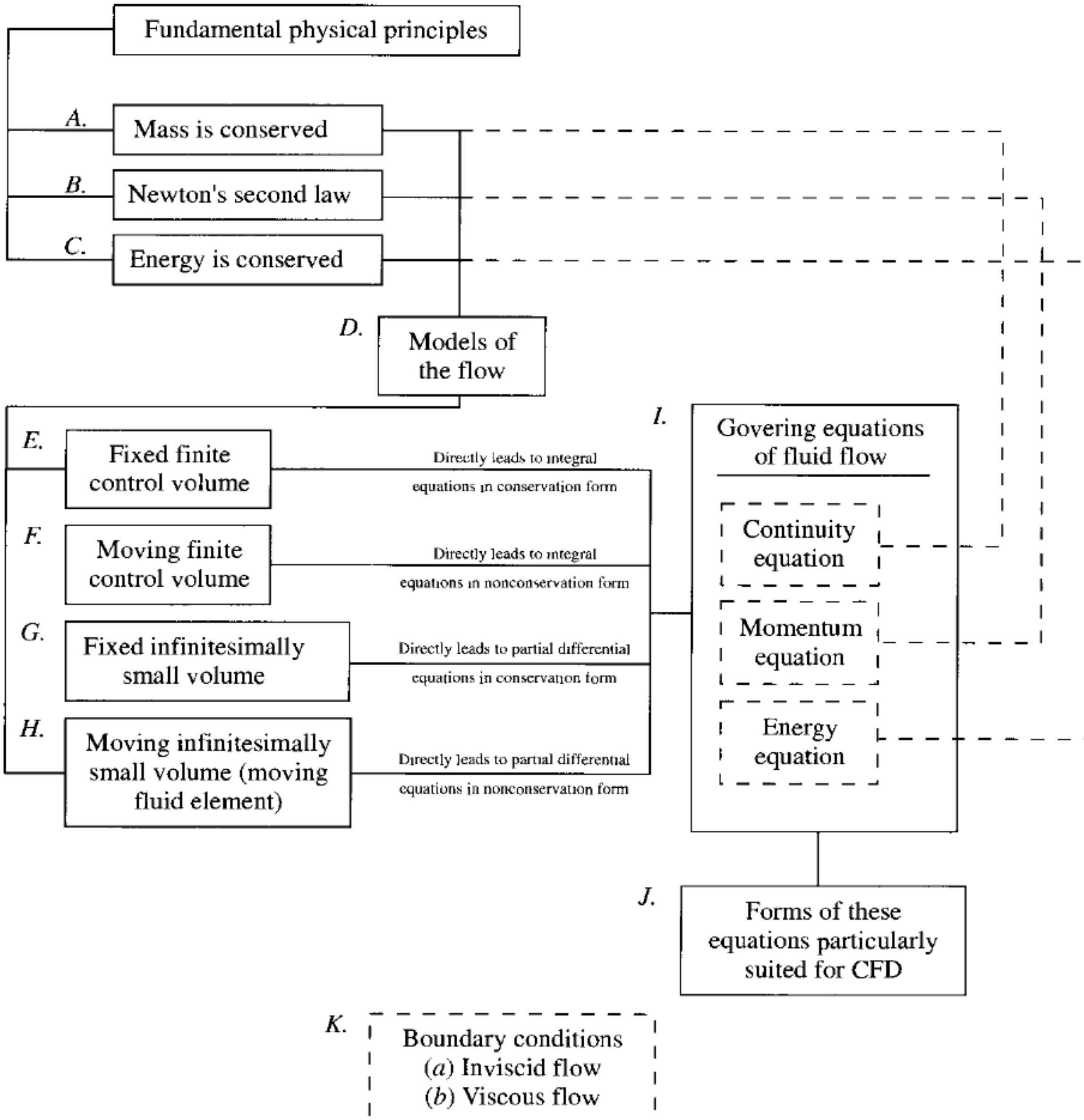
(Non-conservation form)

$$\rho\frac{D}{Dt}\left(e+\frac{V^2}{2}\right)=\rho\dot{q}+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)$$
$$-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}+\frac{\partial(u\tau_{\mathrm{xx}})}{\partial x}$$
$$+\frac{\partial(u\tau_{\mathrm{yx}})}{\partial y}+\frac{\partial(u\tau_{\mathrm{zx}})}{\partial z}+\frac{\partial(v\tau_{\mathrm{xy}})}{\partial x}+\frac{\partial(v\tau_{\mathrm{yy}})}{\partial y}$$
$$+\frac{\partial(v\tau_{\mathrm{zy}})}{\partial z}+\frac{\partial(w\tau_{\mathrm{xz}})}{\partial x}+\frac{\partial(w\tau_{\mathrm{yz}})}{\partial y}+\frac{\partial(w\tau_{\mathrm{zz}})}{\partial z}+\rho\vec{f}\cdot\vec{V}$$

Conservation form

$$\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right]+\nabla\cdot\left[\rho\left(e+\frac{V^2}{2}\vec{V}\right)\right]$$
$$=\ \rho\dot{q}+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)$$
$$+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}+\frac{\partial(u\tau_{\mathrm{xx}})}{\partial x}$$
$$+\frac{\partial(u\tau_{\mathrm{yx}})}{\partial y}+\frac{\partial(u\tau_{\mathrm{zx}})}{\partial z}+\frac{\partial(v\tau_{\mathrm{xy}})}{\partial x}+\frac{\partial(v\tau_{\mathrm{yy}})}{\partial y}$$
$$+\frac{\partial(v\tau_{\mathrm{zy}})}{\partial z}+\frac{\partial(w\tau_{\mathrm{xz}})}{\partial x}+\frac{\partial(w\tau_{\mathrm{yz}})}{\partial y}+\frac{\partial(w\tau_{\mathrm{zz}})}{\partial z}+\rho\vec{f}\cdot\vec{V}$$

Following Chart defines types of governing equations depending on conservation of mass, energy and Newton's second law of motion. Also defines different methods to solve these equations.



3. THE NAVIER–STOKES EQUATIONS

The governing equation used in CFD for fluid flows is the Navier–Stokes equations. The equations of viscous, incompressible fluid flow, known as the Navier–Stokes (N.–S.) equations after the Frenchman (Claude Louis Marie Henri Navier) and Englishman (George Gabriel Stokes) who proposed them in the early to mid 19th Century, can be expressed as

$$\rho \frac{D\boldsymbol{u}}{Dt} = -\nabla p + \mu \Delta \boldsymbol{u} + \boldsymbol{F}_B$$

$$\nabla \cdot \boldsymbol{u} = 0 \,,$$

Where ρ is density of the fluid (taken to be a known constant); u ≡ (u1, u2, u3)T is the velocity vector(which we will often write as (u, v,w)T); p is fluid pressure; μ is viscosity, and FB is a body force. D/Dt is the substantial derivative expressing the Lagrangian, or total, acceleration of a fluid parcel in terms of a convenient laboratory-fixed Eulerian reference frame; ∇ is the gradient operator; _ is

the Laplacian, and ∇· is the divergence operator. We remind the reader that the first of these equations (which is a three component

vector equation) is just Newton's second law of motion applied to a fluid parcel—the left-hand side is mass (per unit volume) times acceleration, while the right-hand side is the sum of forces acting on the fluid element. Equation is simply conservation of mass in the context of constant-density flow.

Comments on the Governing Equations

Surveying the above governing equations, several comments and observations can

be made.

(1) They are a coupled system of non-linear partial differential equations, and hence are very difficult to solve analytically. To date, there is no general closed-form solution to these equations.

(2) For the momentum and energy equations, the difference between the non conservation and conservation forms of the equations is just the left-hand side.

The right-hand side of the equations in the two different forms is the same.

(3) Note that the conservation form of the equations contain terms on the left-hand side which include the divergence of some quantity, such as $\Delta \cdot (\rho\_V)$, $\Delta \cdot (\rho u\_V)$, etc. For this reason, the conservation form of the governing equations is sometimes called the *divergence form*.

(4) The normal and shear stress terms in these equations are functions of the velocity gradients.

(5) The system contains five equations in terms of six unknown flow-field variables, ρ, p, u, v, w, e. In aerodynamics, it is generally reasonable to assume the gas is a perfect gas (which assumes that intermolecular forces are negligible).

For a perfect gas, the equation of state is

$p = \rho RT$

where R is the specific gas constant. This provides a sixth equation, but it also introduces a seventh unknown, namely temperature, T. A seventh equation to close the entire system must be a thermodynamic relation between state variables.

For example,

$e = e(T, p)$

For a calorically perfect gas (constant specific heats), this relation would be

$e = c\mathrm{v}T$

where cv is the specific heat at constant volume.

(6) The momentum equations for a viscous flow were identified as the *Navier–Stokes equations*, which is historically accurate. However, in the modern CFD literature, this terminology has been expanded to include the *entire system* of flow equations for the solution of a viscous flow—continuity and energy as well as momentum. Therefore, when the computational fluid dynamic literature discusses a numerical solution to the 'complete Navier–Stokes equations', it is usually referring to a numerical solution of the *complete system of equations*.

In this sense, in the CFD literature, a 'Navier–Stokes solution' simply means a

solution of a *viscous flow* problem using the *full governing equations*.

Discretization of Partial Differential Equations

Analytical solutions of partial differential equations involve closed-form expressions which give the variation of the dependent variables *continuously* throughout the domain. In contrast, numerical solutions can give answers at only *discrete points* in the domain, called *grid points*.

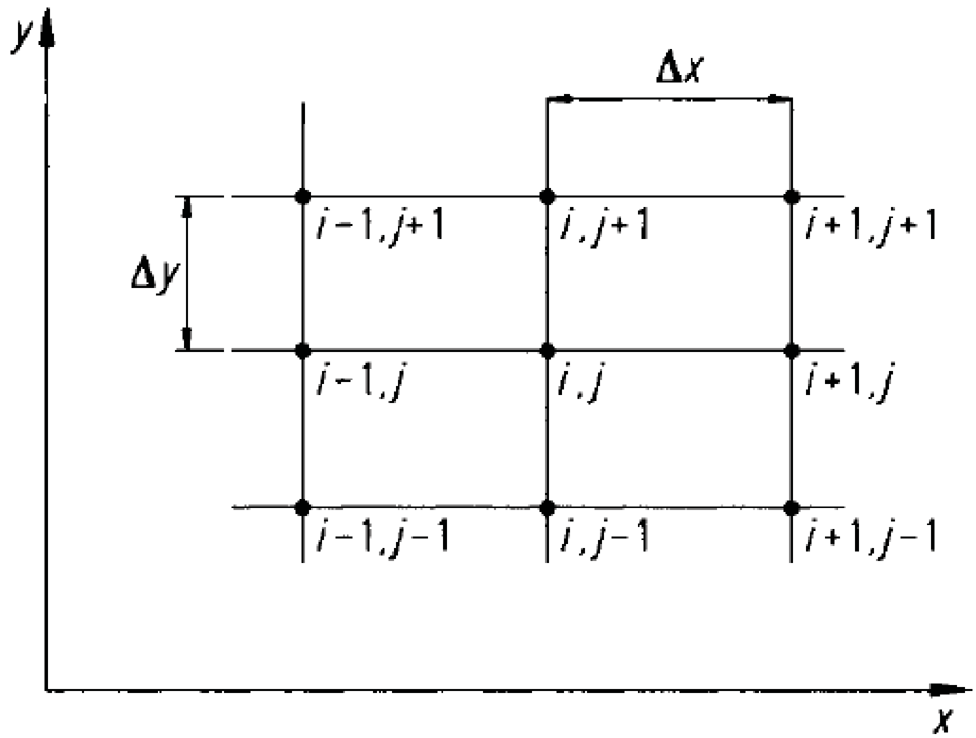


Figure: Discrete grid points

For example, consider above Figure, which shows a section of a discrete grid in the *xy*-plane. For convenience, let us assume that the spacing of the grid points in the *x*-direction is uniform, and given by Δx, and that the spacing of the points in the *y*-direction is also uniform, and given by Δy, as shown in figure.

In general, Δx and Δy are different. Indeed, it is not absolutely necessary that Δx or Δy be uniform; we could deal with totally unequal spacing in both directions, where Δx is a different value between each successive pairs of grid

points, and similarly for Δy. However, the vast majority of CFD applications involve numerical solutions on a grid which involves uniform spacing in each direction, because this greatly simplifies the programming of the solution, saves storage space and usually results in greater accuracy. This uniform spacing does not have to occur in the physical *xy* space; as is frequently done in CFD, the numerical calculations are carried out in a transformed computational space which has uniform spacing in the transformed independent variables, but which corresponds to non-uniform spacing in the physical plane.

Returning to Figure, the grid points are identified by an index *i* which runs in the *x*-direction, and an index *j* which runs in the *y*-direction. Hence, if $(i,\ j)$ is the index for point *P* in Figure then the point immediately to the right of *P* is labeled as $(i+1,\ j)$, the immediately to the left is $(i-1,\ j)$, the point directly above is $(i,\ j+1)$, and the point directly below is $(i,\ j-1)$.

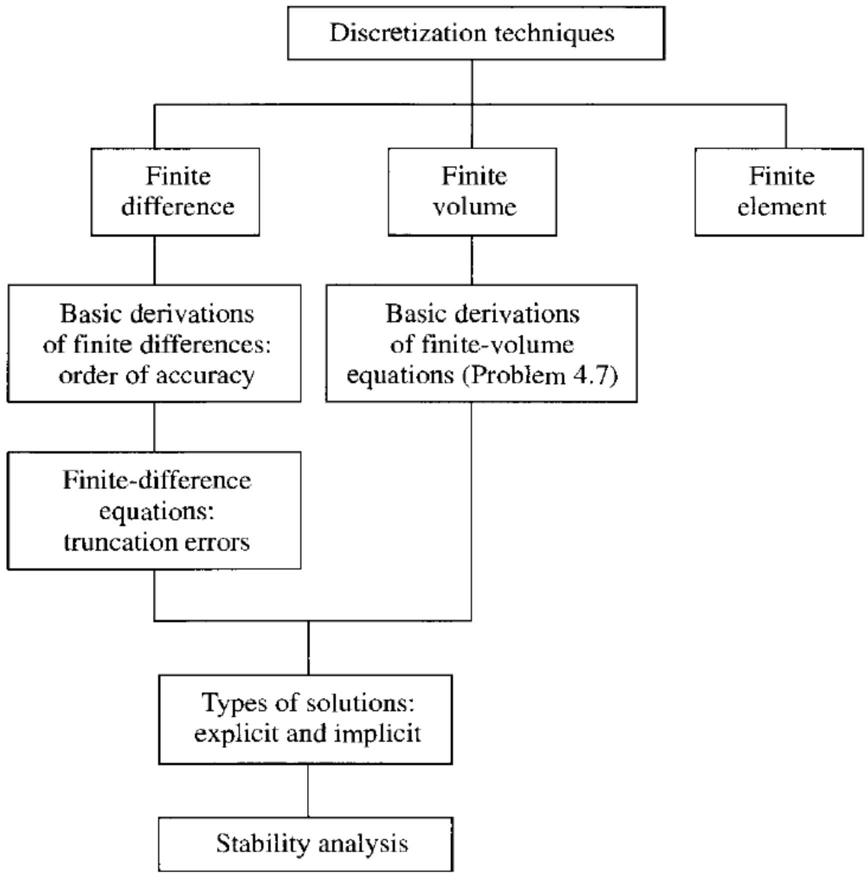
The *method of finite-differences* is widely used in CFD

The philosophy of finite difference methods is to replace the partial derivatives appearing in the governing equations of fluid dynamics with algebraic difference quotients, yielding a system of *algebraic equations* which can be solved for the flow-field variables at the specific, discrete grid points in the flow.

It is clear that finite-difference solutions appear to be philosophically straightforward; just replace the partial derivatives in the governing equations with algebraic difference quotients, and grind away to obtain solutions of these algebraic equations at each grid point. However, this impression is misleading. For any given application, there is no guarantee that such calculations will be accurate, or even stable, under all conditions. Moreover, the boundary conditions for a given problem dictate the solution, and therefore the proper treatment of boundary conditions within the framework of a particular finite-difference technique is vitally important.

For these reasons, finite-difference solutions of various aerodynamic flow fields are by no means routine. Indeed, much of computational fluid dynamics today is still more of an art than a science; each different problem usually requires thought and originality in its solution. However, a great deal of research in applied mathematics is now being devoted to CFD, and the next decade should see a major expansion in our understanding of the discipline, as well as the development of more improved, efficient algorithms.

Flow chart of Discretization Technique:



The objective of computational fluid dynamics is to calculate an entire flow field either around an arbitrary obstacle or through a channel of any shape. The flow may be unsteady, three dimensional, compressible and turbulent. At hypersonic speeds, regions of reacting flow (dissociation, ionization, etc.) might also be considered. The equations to describe this task, are the Navier–Stokes equation, the energy equation, the global and partial continuity equations and other closure model equations describing turbulence and reacting gas effects. It can easily be shown that, at present, no computer could provide either the capacity or the necessary calculation speed to fulfil this task.

The basic laws of fluid dynamics are conservation laws. They are statements that express the conservation of mass, momentum and energy in a volume closed by a surface. Only with the supplementary requirement of sufficient regularity of the solution can these laws be converted into partial differential equations. Sufficient regularity cannot always be guaranteed. Shocks form the most typical example of a discontinuous flow field. In case discontinuities occur, the solution of the partial differential equations is to be interpreted in a weak form, i.e. as a solution of the integral form of the equations. For example, the laws governing the flow through a shock, i.e. the Hugoniot-Rankine laws, are combinations of the conservation laws in integral form. For a correct representation of shocks, also in a numerical method, these laws have to be respected.

There are additional situations where an accurate representation of the conservation laws is important in a numerical method. A second example is the slip line which occurs behind an airfoil or a blade if the entropy production is different on streamlines on both sides of the profile. In this case, a tangential discontinuity occurs.

Another example is incompressible flow where the imposition of incompressibility, as a conservation law for mass, determines the pressure field. In the cases cited above, it is important that the conservation laws in their integral form are represented accurately. The most natural method to accomplish this is to *discretize the integral form of the equations* and not the differential form. This is the basis of a *finite volume method*. Further, in cases where strong conservation in integral form is not absolutely necessary, it is still physically appealing to use the

basic laws in their most primitive form.

The flow field or *domain* is subdivided, as in the finite element method, into a set of *non-overlapping cells* that *cover the whole domain*. In the finite volume method (FVM) the term *cell* is used instead of the term *element* used in the finite element method (FEM). The conservation laws are applied to determine the flow variables in some discrete points of the cells, called *nodes*. As in the FEM, these nodes are at typical locations of the cells, such as cell-centres, cell-vertices or midsides. Obviously, there is considerable freedom in the choice of the cells and the

nodes. Cells can be *triangular, quadrilateral*, etc. They can form a *structured grid* or an *unstructured grid*. The whole geometrical freedom of the FEM can be used in the FVM.

A CFD solution involves the following basic steps:

- Creation of the geometry (or import of the geometry from a CAD package)

- Grid generation

- Choice of the models

- Application of the boundary conditions

- Flow field computation

- Postprocessing

The first step is the creation of the geometry. Usually this is done with a separate CAD package. However, since the grid generator has some specific demands on the imported geometry, the imported geometry often has to be 'cleaned up'. Most CFD packages provide a CAD tool together with their grid generator. The geometry created

with this embedded CAD tool is directly suitable for the grid generator. However, design engineers are using specific CAD packages for their needs and therefore the most common way to obtain the geometry in the grid generation package is the import from a CAD package.

The next phase is the grid generation process. A choice has to be made as to which kind of grid will be used: structured, block structured, unstructured, hybrid. For viscous calculations, a boundary layer mesh also has to be constructed. For turbulent flow calculations, the distance to the wall of the first cell in the boundary layer

mesh depends on the near-wall treatment of the turbulence model. In cases where the grid is not optimal for an accurate solution of the flow field, grid adaptation can be used in order to adapt the grid to the computed flow field features, such as shocks, slip lines, etc. The choice of the models depends on the kind of flow to be computed, and

will have an impact on the grid generation process. The flow can be two- or three dimensional, steady or unsteady, incompressible or compressible, laminar, turbulent or both and heat transfer can be important. These are the models used in the examples in this lesson.

The next step is the application of the boundary conditions. Since the flow field is only computed in the region of interest, adequate boundary conditions have to be provided at the boundaries of the computed region. Frequently used boundary conditions are inlet, outlet and wall boundary conditions. More complex boundary conditions can be defined through user-written routines. The computation of the flow field with the solver becomes of less and less concern to standard users of a commercial CFD software package. So, the user can focus on the fluid dynamics without caring too much of the numerics behind it. However, the more experienced user who intends to write user routines that can be coupled with the software package needs to have a basic understanding of the underlying algorithms of the discretization and solution techniques.

Once the flow field is computed, it can be analyzed in the postprocessing phase. Many postprocessing means are available today. If the user is not satisfied with the solution, a grid adaptation step can be performed as mentioned before. More complex flow calculations e.g. with moving meshes and fluid-structure interactions

can also be performed these days and will have an influence on the different

steps outlined above.

This provides an introduction to the concepts required for developing discretized forms of the governing equations and a discussion of the solution of the resulting algebraic equations. For the most part, we adopt the viewpoint of solving equilibrium (elliptic) problems. This is in contrast to the more frequent emphasis on solving hyperbolic systems.

Although the basic idea of CFD appears straightforward, once again we find that a successful numerical method depends on considerable analysis to formulate an accurate, robust, and efficient solution method. The classification of the mathematical type of the governing equations plays an important role in the development of the numerical methods. Although we adopt finite difference/finite volume methods to solve nonlinear equations, to establish the basic ideas we consider only linear equations.

3.1 Data export

Neutral formats like IGES, STEP, DXF

Direct interfaces to major CAD- and CAE/CFD-systems

Special export format

Gridgen is a high-quality preprocessor for computational fluid dynamics.

4. ANSYS CFD-Post

ANSYS CFD-Post software incorporates many advanced features to generate powerful graphical renderings of fluid flow solutions

Computational fluid dynamics (CFD) simulations don't end with the fluid flow prediction. Benefiting from the prediction requires post-processing that gives users complete insight into their fluid dynamics simulation results. ANSYS CFD-Post software, the common post-processor for all ANSYS fluid dynamics products, gives users everything they need to visualize and analyze their results. Within a modern and intuitive user interface, ANSYS CFD-Post software sets no limits on creativity when generating powerful images to illustrate the flow in any desired level of detail. From vector plots and streamlines to vortex cores and flow animations, ANSYS CFD-Post software provides users all the tools they need to produce insightful solution visualizations, including 3-D images. These high-quality visuals are invaluable in communicating results to colleagues and customers by helping to explain and provide an understanding of complex flow phenomena. High-end graphical post-processing is only one part of ANSYS CFD-Post technology; its quantitative post-processing abilities are at least as important and powerful. With the functions of ANSYS CFD-Post at their fingertips, users can quickly get all the data they need out of their calculation; weighted averages, mass flows, forces, maximum/minimum values and many more functions enable precise analysis of the CFD results. with automatic report generation, and there is virtually no limit to the post-processing possibilities with the ANSYS CFD-Post solution. The final touch is automation. Users can easily create macros — for example by recording the interactive steps in a session file for replay and reuse with similar cases — and create images, charts, tables and reports automatically for different simulation results. CFD-Post software can also run completely in batch mode for fully automated processes and integration in optimization. Beyond a plethora of individual features and options, ANSYS CFD-Post technology provides key functionality that allows users to get full value from their CFD simulation:

5. Results Comparison

ANSYS CFD-Post software allows multiple solution datasets to be loaded simultaneously, significantly easing the comparison of different design alternatives or operating conditions. Results, including those for fluid structure interaction (FSI), can be examined side-by-side with synchronized views, as well as with synchronized time for transient simulations. Additionally, differences between two results can be computed and analyzed both visually and quantitatively.

5.1 3-D Images

All images created in the ANSYS CFD-Post software can, of course, be saved in standard 2-D image formats like JPG and PNG. However, it is often difficult to find the right 2-D views to effectively communicate results to managers, customers and colleagues. For these situations, ANSYS CFD-Post technology provides the ability to write 3-D image files that anybody can view with a freely distributable 3-D viewer available from ANSYS. It is even possible to embed these 3-D images in applications like Microsoft® PowerPoint® to add a new dimension to presentations

5.2 Custom Reports

Each session in ANSYS CFD-Post software includes a standard template for report generation .Simple selection and de selection of items to include in the report allows users to customize the content with user-defined text, images, charts or tables, as well as with the company logo at the top. The report is dynamic, updating automatically with new datasets. The final report can then be exported to HTML, optionally with 3-D images.

5.3 Flow Animation

Whether simulations are steady or transient, animations help bring CFD results to life. Animations can be quickly defined, including powerful key frame settings, and the resulting animation saved to high-quality and compact MPEG-4 output formats. With the many graphical features and rendering options like texturing, animations from ANSYS CFD-Post software are sure to make a strong impression on your audience.

5.4 Calculators and Expressions

To quickly probe the solution and perform integral operations on the results data, ANSYS CFD-Post software provides a range of functions with its integrated calculators. The same functions can also be applied in a powerful expression language that is part of ANSYS CFD-Post to extract values from the original data or from any desired derived variables and quantities not in the original dataset. Insightful visualization, flexible calculation and comprehensive automation are fundamental to the advanced post-processing provided by ANSYS CFD-Post software, all in an advanced, easy-to-use user interface.

6. CFD commercial software's

PORFLOW

AR Software (TEP: a combustion analysis tool for windows)

COSMIC NASA

Fluent Inc. (Fluent/V4, Fluent/UNS, Rampant, Nekton)

Flowtech Int. AB (SHIPFLOW: analysis of flow around ships)

Fluid Dynamics International, Inc. (FIDAP)

AEA Technology (CFX: 3D fluid flow/heat transfer code)

ICEM CFD (ICEM CFD, Icepak)

KIVA (reactive flows)

CFD Research Corporation (ACE: reactive flows)

Computational Dynamics Ltd. (STAR-CD)

Analytical Methods, Inc. (VSAERO, USAERO, OMNI3D)

AeroSoft, Inc. (GASP and GUST)

Ithaca Combustion Enterprises (PDF2DS)

Flow Science, Inc. (FLOW3D)

ALGOR, Inc. (ALGOR)

Engineering Mechanics Research Corp. (NISA)

Reaction Engineering International (BANFF/GLACIER)

Combustion Dynamics Ltd. (SuperSTATE)

AVL List Gmbh. (FIRE)

IBM Corp. catalogue (30 positions)

Sun Microsystems catalogue (70 positions)

Cray Research catalogue (100 positions)

Silicon Graphics, Inc. catalogue (75 positions)

Pointwise, Inc. (Gridgen - structured grids)

Simulog (N3S Finite Element code, MUSCL)

Directory of CFD codes on IBM supercomputer environment

ANSYS, Inc. (FLOTRAN)

Flomercis Inc. (FLOTHERM)

Computational Mechanics Corporation

Computational Mechanics Company, Inc. (COMCO)

KASIMIR (shock tube simulation program)

Livermore Software Technology Corporation (LS-DYNA3D)

Advanced Combustion Eng. Research Center (PCGC, FBED)

NUMECA International s.a. (FINE, FINE/Turbo, FINE/Aero, IGG, IGG/Autogrid)

Computational Engineering International., Inc. (EnSight, ...)

Blocon Software Agency (HEAT2, HEAT3)

Adaptive Research Corp. (CFD2000)

Unicom Technology Systems (VORSTAB-PC)

Incinerator Consultants Incorporated (ICI)

PHOENICS/CHAM (multi-phase flow, N-S, combustion)

Synergium's CFD Tools (AVIA)

Innovative Aerodynamic Technologies (LAMDA)

XYZ Scientific Applications, Inc. (TrueGrid)

South Bay Simulations, Inc. (SPLASH)

PHASES Engineering Solutions

Amtec Engineering, Inc. (INCA, Tecplot)

Engineering Sciences, Inc. (UNIC)

Catalpa Research, Inc. (TIGER)

Swansea NS codes (LAM2D, TURB)

Engineering Systems International S.A. (PAM-FLOW, PAM-FLUID)

Daat Research Corp. (COOLIT)

Flomerics Inc. (FLOVENT)

Innovative Research, Inc.

Centric Engineering Systems, Inc. (SPECTRUM)

Blue Ridge Numerics, Inc.

WinPipeD

Exa Corporation (PowerFLOW)

Polyflow s.a.

Flow Pro

Computational Aerodynamics Systems Co.

Tahoe Design Software

ADINA-F

YFLOW

PSW

Advanced Visual Systems

Flo++

KSNIS

Flowcode

Concert

SMARTFIRE

VISCOUS

Polydynamics

Cullimore and Ring Technologies, Inc. (SINDA/FLUINT, SINAPS)

Linflow (ANKER - ZEMER ENGINEERING)

PFDReaction

Airfoil Analysis

Institute of Computational Continuum Mechanics GmbH

CFD++

RADIOSS-CFD

6.1 CFD open source codes list

Foil 1.0.2

PPM -- Piecewise Parabolic Method (DE and LR versions)

General Relativity NCSA group (black hole evolution)

NACA airfoils

College of Marine Studies (sci.geo.fluids models)

Ocean models (MOM, POM, POP, ...)

CLAWPACK (library of codes by Randall LeVeque)

HEATING (a multidimensional heating conduction code)

PHI3D (3D FEM Navier-Stokes code)

HYDRO (2D Lagrangian hydrodynamics code)

Smoothed Particle Hydrodynamics

Cullimore and Ring Technologies, Inc. (SINDA/FLUINT, SINAPS)

STSWM (Spectral Transform Shallow Water Model)

FCT (Flux-Corrected Transport)

NSC2KE (N-S finite-volume Galerkin 2D code)

HENSA archive (FLUX, NSUVP, TEAM, VORTEX)

FEMLAB (2D FEM with automatic error control)

Riemann Problem Package

CHAMMP repository (s-w equations in spherical geometry)

NACHOS (NACHOS - FEM for incompressible flows)

FMS-1D (Fluid Modeling System)

NPARC Flow Solver

QUICK 'n SIMPLE (2-D, Macintosh)

ViewProf (inviscid 2-d multiblock Oellers method panel solver)

Numerical Models at IMCS (ocean circulation/bottom boundary layer models)

Synergium's CFD Tools (free version of AVIA)

ALLSPD-3D Combustor Code

Unicom Technology Systems (VORSTAB-PC)

FORTRAN codes for computing the discrete Helmholtz integral operators

NaSt2D (2D Navier-Stokes solver)

WinPipeD (pipe hydraulics)

DROP

Test Problem Code Archive

USGS water resources applications software

The Center for Computational Sciences and Engineering, LBNL

Pab3d

CFX User Subroutine Archive

MicroTunnel

Mouse

Mathtools

7. ANSYS FACILITIES FOR CFD

ANSYS FLUENT:	General Purpose Computational Fluid Dynamic (CFD) Software using the finite volume method
ANSYS POLYFLOW:	A finite-element based CFD package for the analysis of polymer processing
ANSYS Icepak:	Design Tool for Electronics Cooling, using the finite volume method
ANSYS Airpak:	Specialized Software for HVAC&R (Humidity, Ventilation, Air Conditioning and Refrigeration)
ANSYS Design Modeler:	Geometry Creation

ANSYS TGrid:	Specialized preprocessor used to create unstructured tetrahedral and Hex Core meshes for complex and very large surface meshes
ANSYS CFX:	General purpose Computational Fluid Dynamic (CFD) Software using the finite volume method
ANSYS ICEM CFD:	Geometry acquisition, mesh generation and a wide variety of solver outputs and post-processing
ANSYS CFD:	Provides access to both ANSYS FLUENT and ANSYS CFX products
ANSYS CFD-Post:	ANSYS CFD-Post is? the common post-processor for all ANSYS fluid dynamics products
ANSYS Meshing:	Advanced meshing tool

8. Automobile Radiators

Almost all automobiles in the market today have a type of heat exchanger called a radiator. The radiator is part of the cooling system of the engine as shown in Figure 3 below. As you can see in the figure, the radiator is just one of the many components of the complex cooling system.

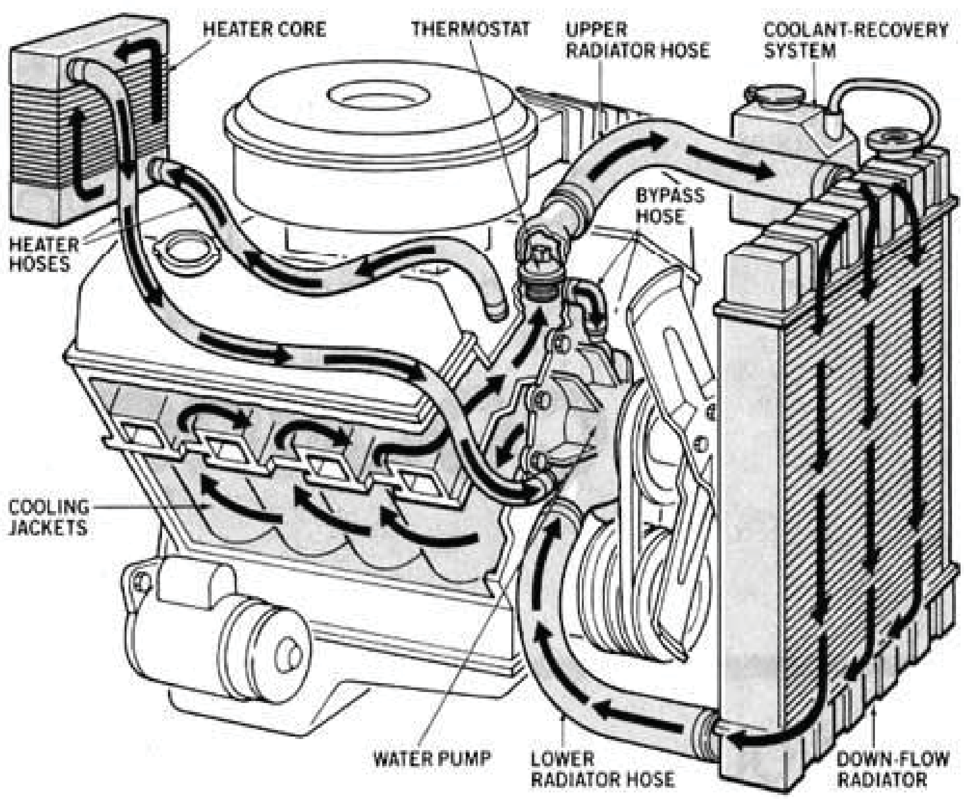
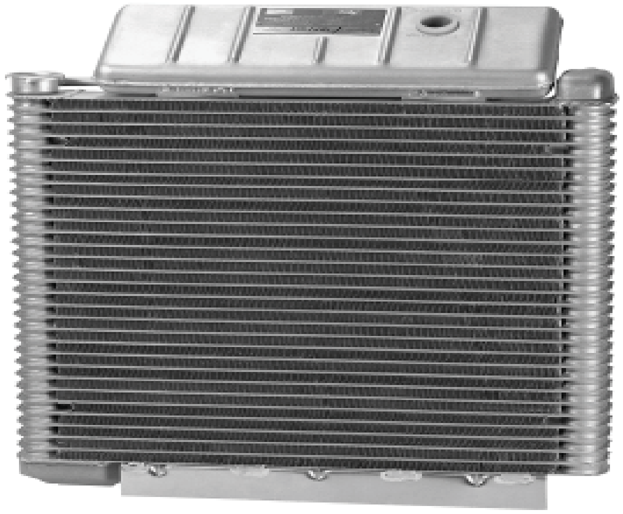
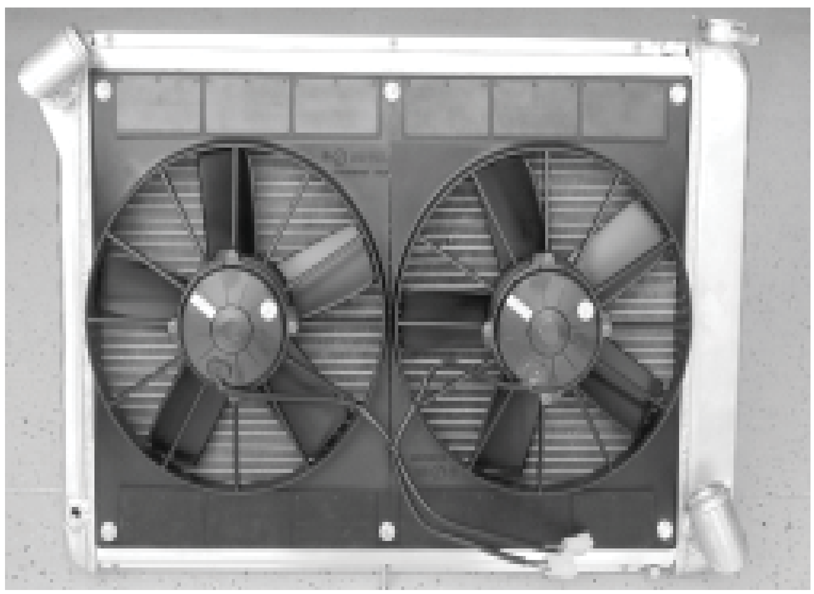


Figure (3) Coolant path and Components of an Automobile Engine Cooling System

Different types of Automobile Radiators:





Most commonly made out of aluminum, automobile radiators utilize a cross-flow heat exchanger design. The two working fluids are generally air and coolant (50-50 mix of water and ethylene glycol). As the air flows through the radiator, the heat is transferred from the coolant to the air. The purpose of the air is to remove heat from the coolant, which causes the coolant to exit the radiator at a lower temperature than it entered at. The benchmark for heat transfer of current radiators is 140 kW of heat at an inlet temperature of 95 °C. The basic radiator has a width of 0.5-0.6 m (20-23"), a height of 0.4-0.7 m (16-27"), and a depth of 0.025-0.038 m (1-1.5"). These dimensions vary depending on the make and model of the automobile. For current radiator designs, a common configuration is to use parallel tubes which have aluminum fins attached to them. In these designs, there are basically three modes of heat transfer: conduction between tube walls and fins, and two modes of convection. One mode of convection is due to the coolant flowing in the tubes and the second is caused by the air flowing through the radiator. Associated with each type of heat transfer is a thermal resistance which obstructs the heat transfer rate.

In current radiator designs, the largest thermal resistance is caused by the convective heat transfer (Rconv) that is associated with the air. This comprises of over 75% of the total thermal resistance. The second largest thermal resistance is caused by the convection that is associated with the fluid. Together, these resistances comprise of over 97% of the total thermal resistance. Since there is a large thermal resistance associated with the air, the increased heat transfer cannot be observed. Therefore, there is a need to design a radiator that reduces the percentage of thermal resistance associated with the air.

Limitations

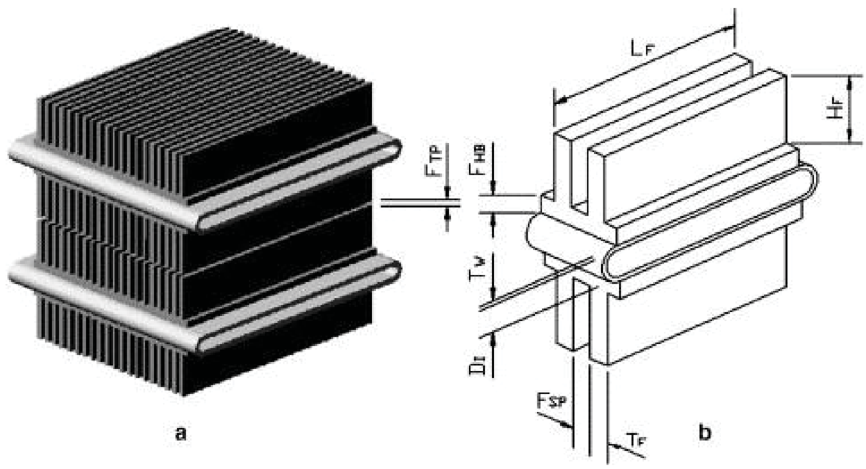
Current radiator designs are extremely limited and have not experienced any major advancements in recent years. As described above, the main problem is that current radiators experience a large resistance to heat transfer caused by air flowing over the radiator. Current radiators also experiences head resistance, are very bulky, and impose limitations on the design of the vehicle.

Case Studies

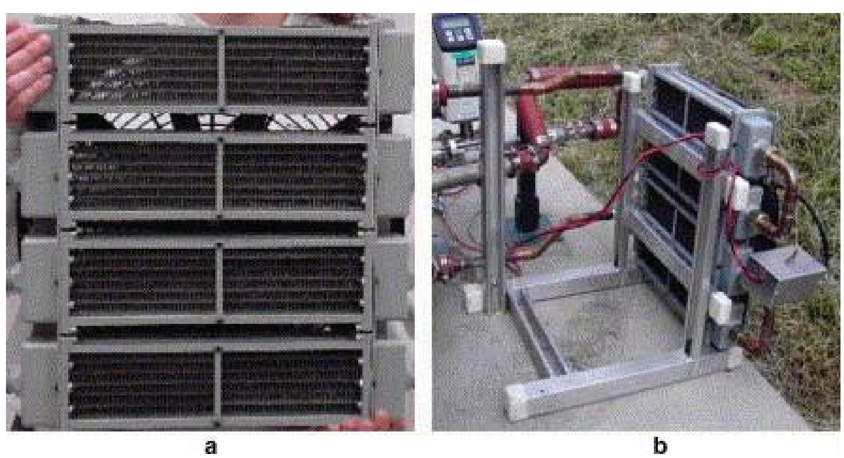
After searching technical papers, we found several related articles on different materials and designs for radiators. As shown in the case studies below, there are several ways to improve the current radiator design. This information will be used to develop a new design.

Case Study #1

Case study #1 showed that one way to decrease the thermal resistance associated with the air is to change the type of fin material used. Instead of using aluminum fins, fins constructed of carbon-foam were used. The fins were constructed out carbon-foam that had a porosity of 70%, a thickness of 0.762 mm, and a height of 8.725 mm. The fin density was set to 748 fins/m. The carbon-foam fins can be seen in Figure below.



Carbon-foam Fin



Test setup for Carbon-foam Finned Radiator

The setup for this case study is shown in above Figure. It showed that the percentage of thermal resistance associated with air-side convection was reduced to about 60%, therefore the percentage of the thermal resistance associated with the fluid was increased. With the shift in these percentages, the convective benefits of nanofluids would have a more significant role.

Case Study #2

In case study #2, a possible improvement to the automobile radiator was seen through the analysis of micro heat exchangers. These heat exchangers incorporated the use of micro-channels and were fabricated from plastic, ceramic, or aluminum. The micro heat exchanger can be seen in Figures (a) and (b) below.

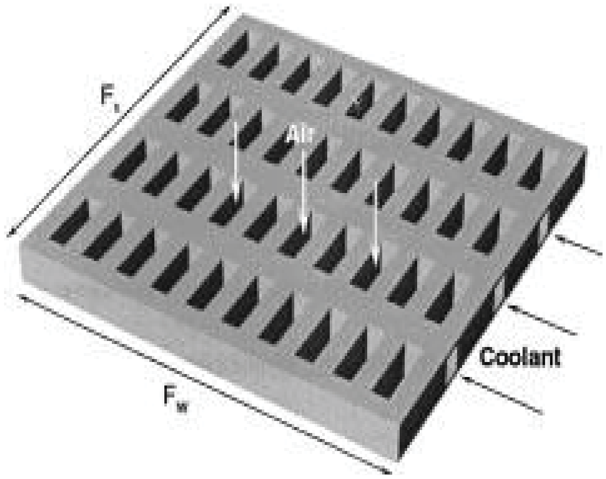


Figure (a) : **Micro-channel Heat Exchanger**

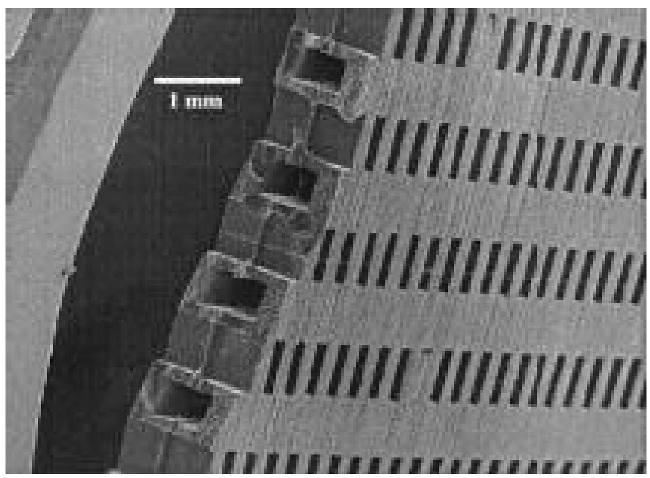


Figure (b): Micro-channel Heat Exchanger

When compared to several automobile radiators, the micro heat exchanger outperformed them in a couple of areas. One area was on a heat transfer rate to volume basis in which the micro heat exchanger was better by more than 300%. Another area was a heat transfer rate per mass basis. In this area, the micro heat exchanger showed improvement of about 200%. These improvements were achieved by limiting the flow to smaller channels which increased the surface area/volume ratio and reduced the convective thermal resistance associated with the solid/fluid interface. However, in this study, the automobile radiators did outperform the micro heat exchanger on a heat transfer rate per frontal area basis. Here, the micro heat exchanger showed a reduction of over 45%. However, it is possible to construct a micro heat exchanger that has the same heat transfer rate/frontal area as current automobile radiators by using a more conductive material and reducing the spacing between the fins. Therefore, when compared to automobile radiators, the use of micro heat exchangers allows the same amount of heat to be dissipated with a reduced volume and weight.

9. APPLICATIONS OF CFD

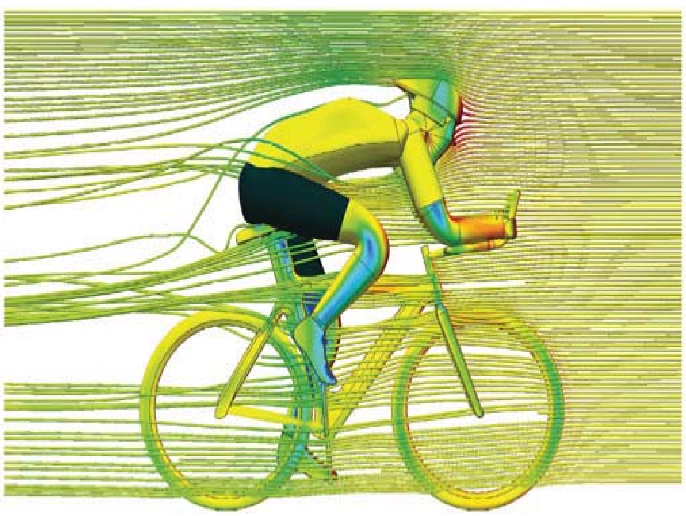
- **Biomedical**
- Electronics
- Defense
- Industrial
- Appliances
- Chemical & Petrochemical Processing
- Consumer Packaged Goods
- Food & Beverage

- Fuel Cells
- Glass Processing
- Metals, Minerals & Mining
- Mixing
- Nonwoven Materials
- Nuclear Power
- Oil & Gas
- Polymer Processing
- Power Generation
- Pump
- Turbomachinery
- Environmental

10.

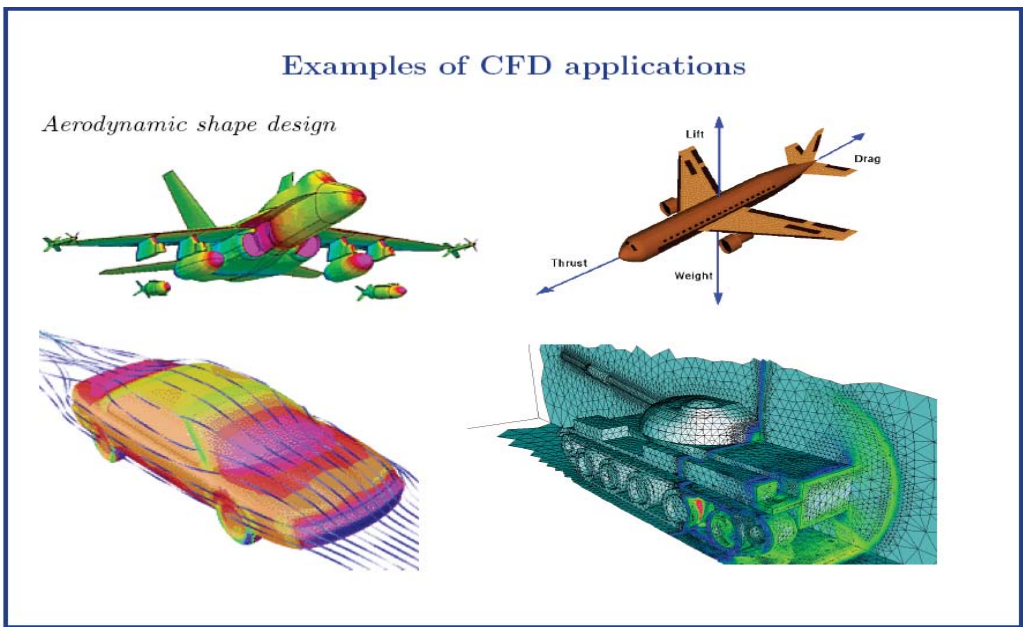
CFD is useful in a wide variety of applications and here we note a few to give you an idea of its use in industry. The simulations shown below have been performed using the FLUENT software.

CFD can be used to simulate the flow over a vehicle. For instance, it can be used to study the interaction of propellers or rotors with the aircraft fuselage the following figure shows the prediction of the pressure field induced by the interaction of the rotor with a helicopter fuselage in forward flight. Rotors and propellers can be represented with mode of varying complexity.



Prediction of the pressure field induced by the interaction of the rotor with a helicopter fuselage in forward flight

The temperature distribution obtained from a CFD analysis of a mixing manifold is shown below. This mixing manifold is part of the passenger cabin ventilation system on the Boeing767. The CFD analysis showed the effectiveness of a simpler manifold design without the need for field testing.



Examples of CFD applications

11. CONCLUSION

The radiator size can be reduced without affecting the heat transfer characteristics with improved rate of heat transfer. Depending on the requirements suitable modifications need to be incorporated thereby making the design more indigenous both from design point of view and economical point of view.

12. REFERENCES

1. W.M. Rohsenhow, J.P. Hartnett, Y.I. Cho, Handbook of Heat Transfer,

[10] 3rd ed., Mc-Graw Hill, New York, 1998.

[11] R.M. Maglik, A.E. Bregles, Heat transfer and pressure drop correlations
[12] for the rectangular offset-strip-fin compact heat exchanger, Exp. Therm.
[13] Fluid Sci. 10 (1995) 171–180.
[14] W.M. Kays, A.L. London, Compact Heat Exchangers, 3rd ed., Mc-Graw
[15] Hill, New York, 1984.
[16] M. Yaghoubi, M. Rahnema, Numerical study of turbulent flow and heat
[17] transfer from an array of thick plates, Int.J. Therm. Sci. 39 (2000) 213–224
[18] http://en.wikipedia.org/wiki/Computational_fluid_dynamics