



# COMPUTATIONAL FLUID DYNAMICS (CFD)

## Introduction

Historically there were two methods to explain fluid flow phenomena. The first one was Analytical Fluid Dynamics (AFD) which using mathematical formula and makes differential analysis on control volume for the studying case and gives approximate solutions for practical applications. The second method was Experimental Fluid Dynamics (EFD) which using experimental methodology and procedures for solving fluids engineering systems, including full and model scales, large and table top facilities, measurement systems (instrumentation, data acquisition and data reduction), uncertainty analysis, and dimensional analysis and similarity. Computational Fluid Dynamics (CFD) has become feasible due to the advent of high speed digital computers.

## Computational fluid dynamics (CFD)

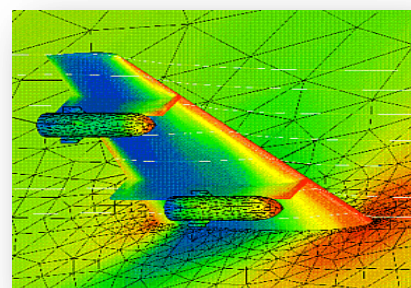
is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. Even with high-speed supercomputers only approximate solutions can be achieved in many cases. Ongoing research, however, may yield software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is often performed using a wind tunnel with the final validation coming in flight test.

## 5.2 Importance of CFD in engineering studies

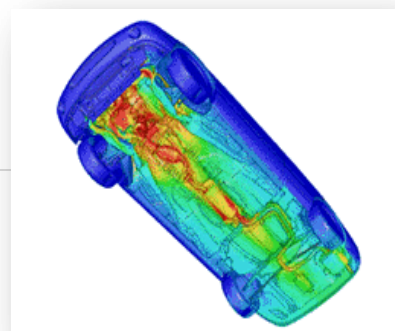
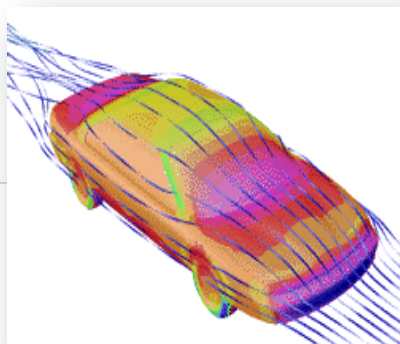
As a matter of fact the importance of CFD is in two essential directions. First in Simulation-based design CFD is more cost effectively and more rapidly than build and test experiments. It also provides high-fidelity database for interrogation of flow field. On the other hand there are some applications which are difficult to be measured by experiments. CFD is making Simulation of physical fluid phenomena for these applications such as scale simulations (e.g., full-scale ships and airplanes), hazards (e.g., explosions, radiation, and pollution) and physics (e.g., weather prediction and planetary boundary layer).

## 5.3 CFD Applications

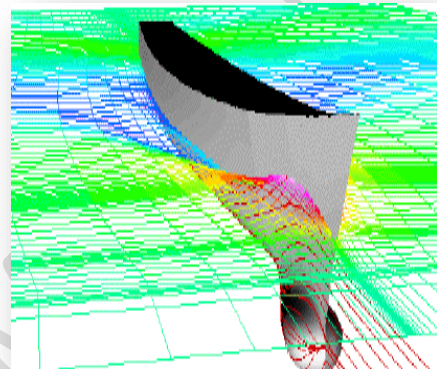
- *Aerospace*



- *Automotive*



- *Marine*

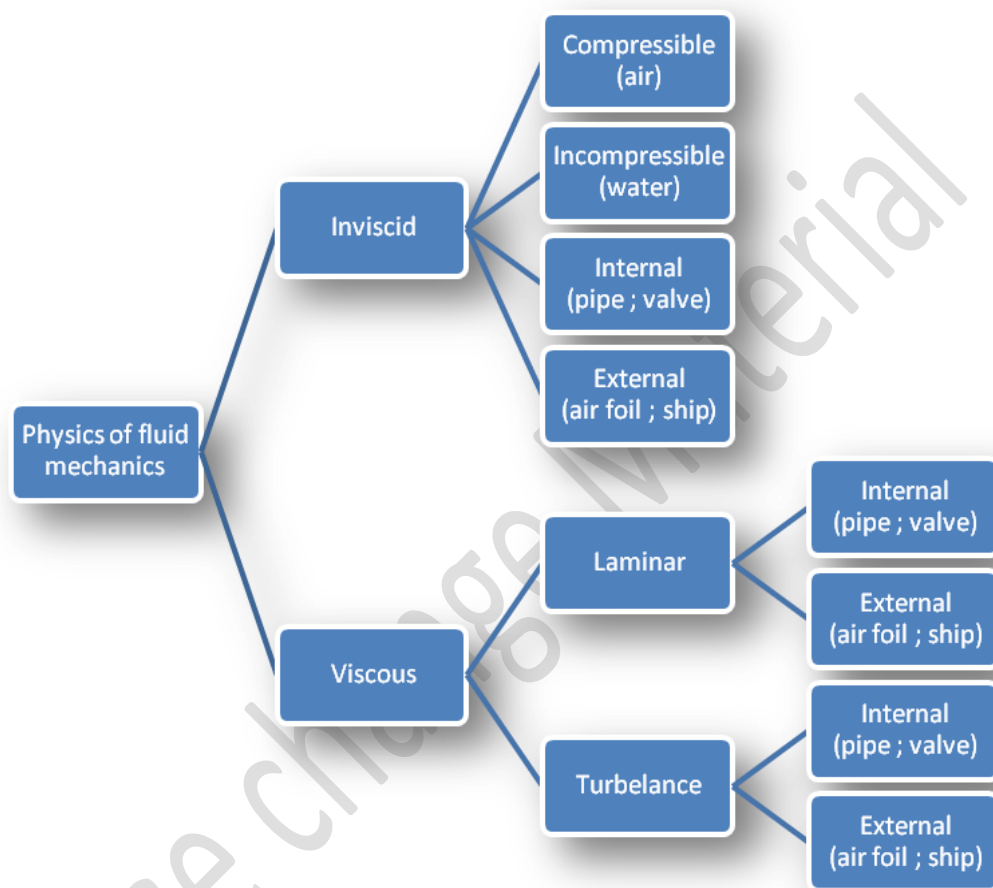


## 5.4 Physics **and** Modeling of CFD

### **PHYSICS**

CFD codes typically designed for representation of specific flow phenomenon

1. Viscous or inviscid (no viscous forces).
2. Turbulent or laminar (Re).
3. Incompressible or compressible (Ma).
4. Single or multi-phase.
5. Thermal/density effects and energy equation (Pr, g, Gr)
6. Free-surface flow and surface tension (Fr, We)
7. Chemical reactions, mass transfer.



### **MODELING**

Mathematical physics problem formulation of fluid engineering system:

- Governing equations: Navier-Stokes equations (momentum), continuity equation, energy equation, ideal gas law, combustions (chemical reaction equation) , multi-phase flows (e.g. Rayleigh equation) and turbulent models (RANS, LES, DES).
- Coordinates: Cartesian, cylindrical and spherical coordinates result in different form of governing equations

- Initial conditions(initial guess of the solution) and Boundary Conditions (no-slip wall, free-surface, zero-gradient, symmetry, velocity/pressure inlet/outlet)
- Flow conditions: Geometry approximation, domain, Reynolds Number, and Mach number, etc.

## 5.5 CFD processes

### 1. Geometry description:

- a. Make assumptions and simplifications.
- b. CAD/CAE integration.
- c. Engineering drawings.
- d. Coordinates include Cartesian system (x, y, z), cylindrical system (r,  $\theta$ , z), and spherical system(r,  $\theta$ ,  $\Phi$ ).

### 2. Specification of flow conditions and properties:

Flow conditions and properties required are unique for each flow code and application.

### 3. Selection of models:

- a. Direct Numerical Simulations (DNS) is to solve the N-S equations directly without any modeling. Grid must be fine enough to resolve all flow scales. Applied for laminar flow and rare be used in turbulent flow.
- b. Reynolds Averaged Navier-Stokes (NS) equations (RANS) are to perform averaging of NS equations and establishing turbulent models for the eddy viscosity. Too many averaging might damping vertical structures in turbulent flows
- c. Large Eddy Simulation (LES), Smagorinsky' constant model and dynamic model. Provide more instantaneous information than RANS did. Instability in complex geometries
- d. Detached Eddy Simulation (DES) is to use one single formulation to combine the advantages of RANS and LES.

#### 4. Specification of initial and boundary conditions:

##### a. For steady/unsteady flow

- IC should not affect final solution, only convergence path, i.e. iteration numbers needed to get the converged solution.
- Robust codes should start most problems from very crude IC. But more reasonable guess can speed up the convergence.

##### b. Boundary conditions

No-slip or slip-free on the wall, periodic, inlet (velocity inlet, mass flow rate, constant pressure, etc.), outlet (constant pressure, velocity convective, buffer zone, zero-gradient), and non-reflecting (compressible flows, such as acoustics), etc.

#### 5. Grid generation and transformation:

Grids can either be structured (hexahedral) or unstructured (tetrahedral). Depends upon type of discretization scheme and application.

#### 6. Specification of numerical parameters:

#### 7. Numerical parameters are used to control flow solution:

##### a. CFD Labs using Flow Lab

- Monitor residuals (change of results between iterations)
- Number of iterations for steady flow or number of time steps for unsteady flow

##### b. Flow solution

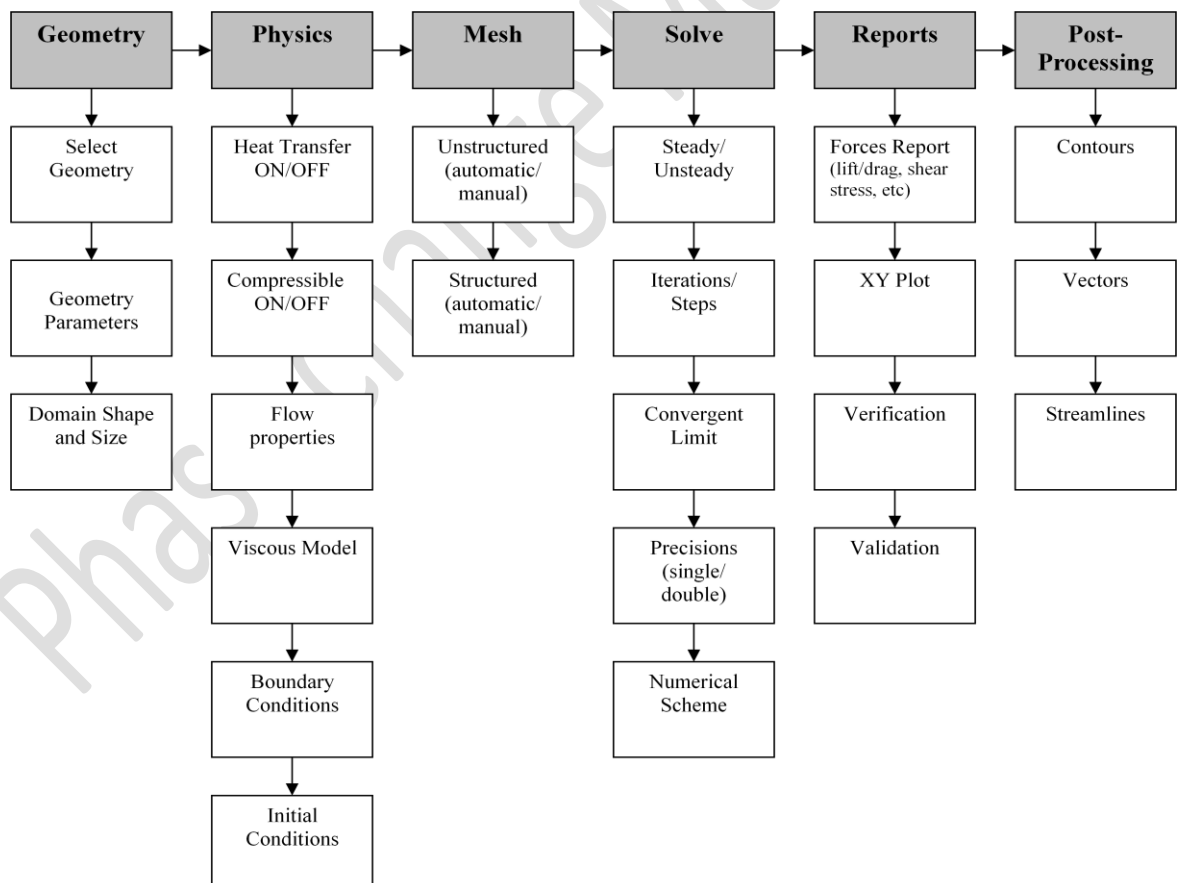
Solve the momentum and get flow field quantities, such as velocity, turbulence intensity, pressure and integral quantities (drag forces).

#### 8. Flow solution

#### 9. Post processing (Analysis, and visualization):

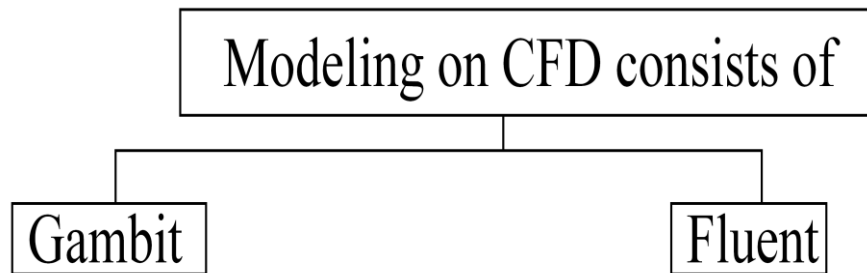
##### a. Calculation of derived variables

- Vorticity
  - Wall shear stress
- b. Calculation of integral parameters: forces, moments
- c. Visualization (usually with commercial software)
- Simple X-Y plots
  - Simple 2D contours
  - 3D contour carpet plots
  - Vector plots and streamlines (streamlines are the lines whose tangent direction is the same as the velocity vectors)
  - Animations (dozens of sample pictures in a series of time were shown continuously)





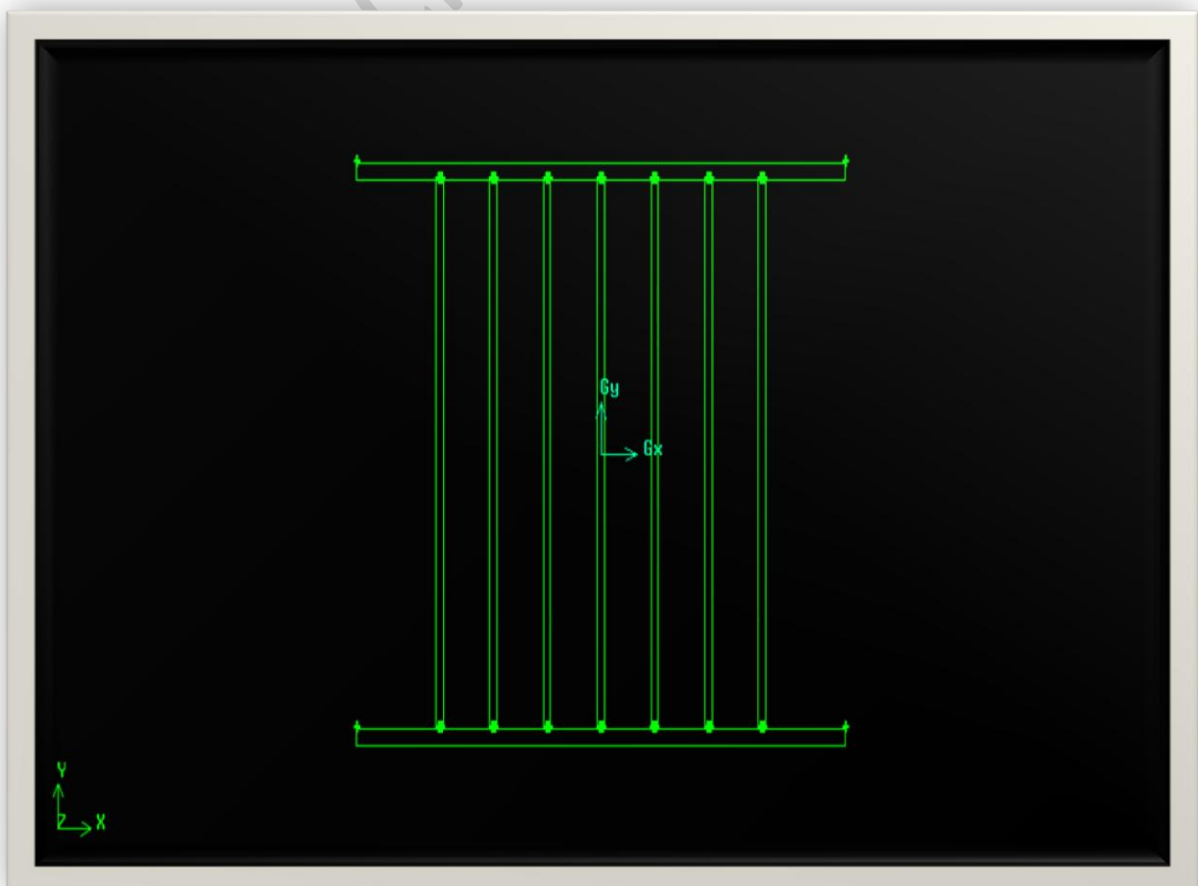
## 5.6 Thermal solar collector modeling on CFD



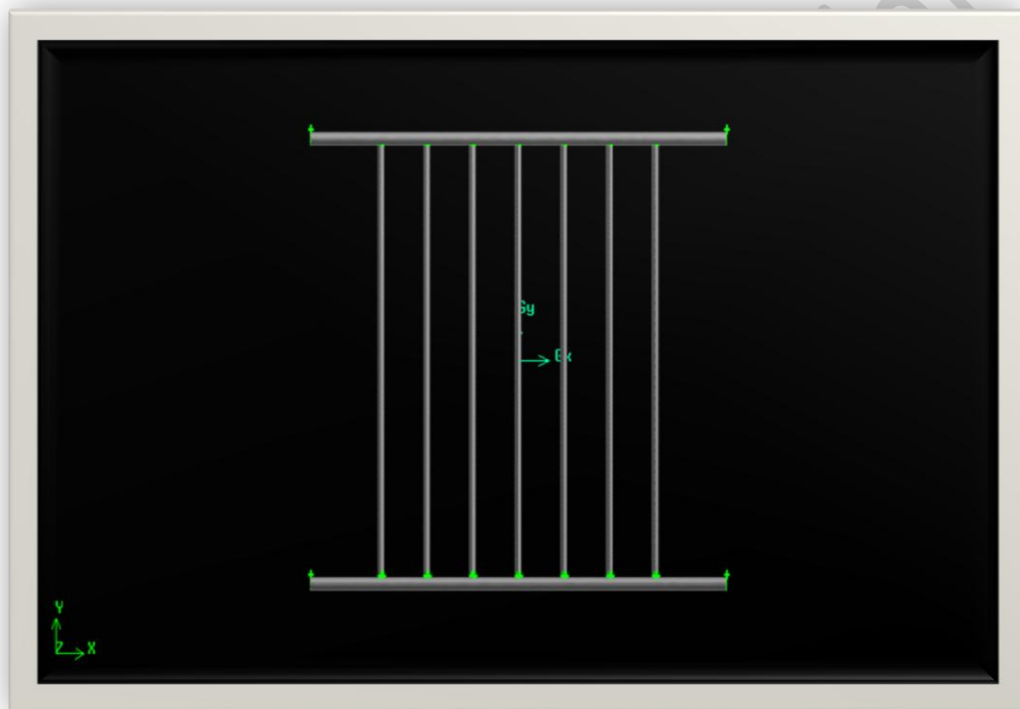
### 5.6.1 Gambit

#### 1- The solar collector drawing:

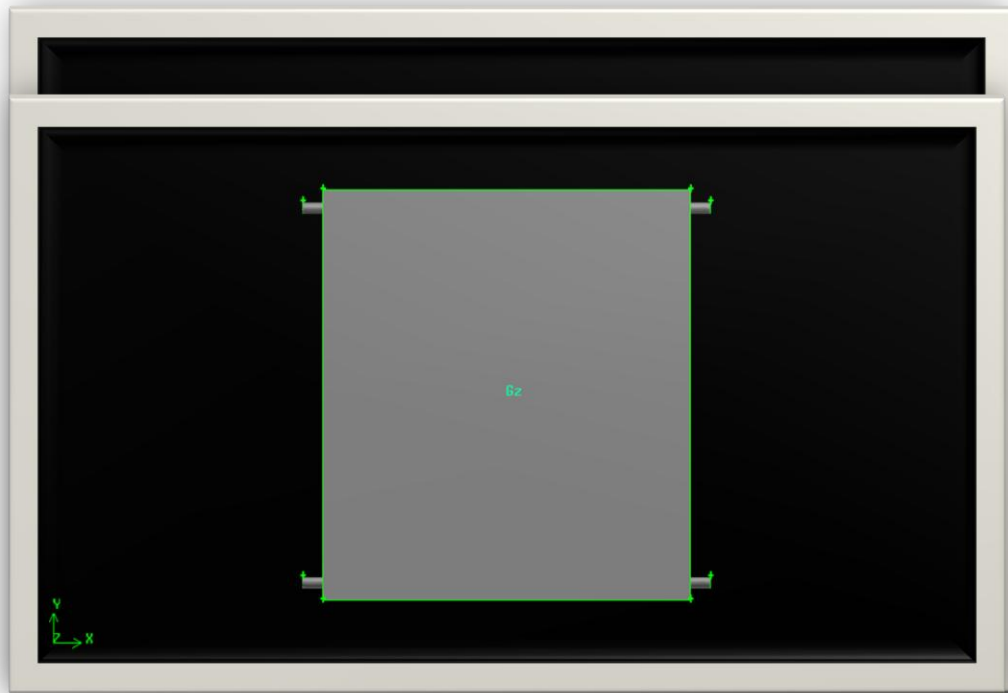
- Draw a flat plate (absorber) made of red copper with 90 cm width, 90 cm depth .
- Draw one header pipe which is made of red copper with 1.5 cm in diameter and 110 cm length.
- To draw the seven risers which are made of copper with 1.1 cm in diameter 88.5cm length and the distance between them is 11 cm.
- Draw one pipe with the previous dimensions.
- Make 6 copies at distance 11cm from the first pipe in x- direction.



- f. Unite real volumes for absorber, headers and risers.



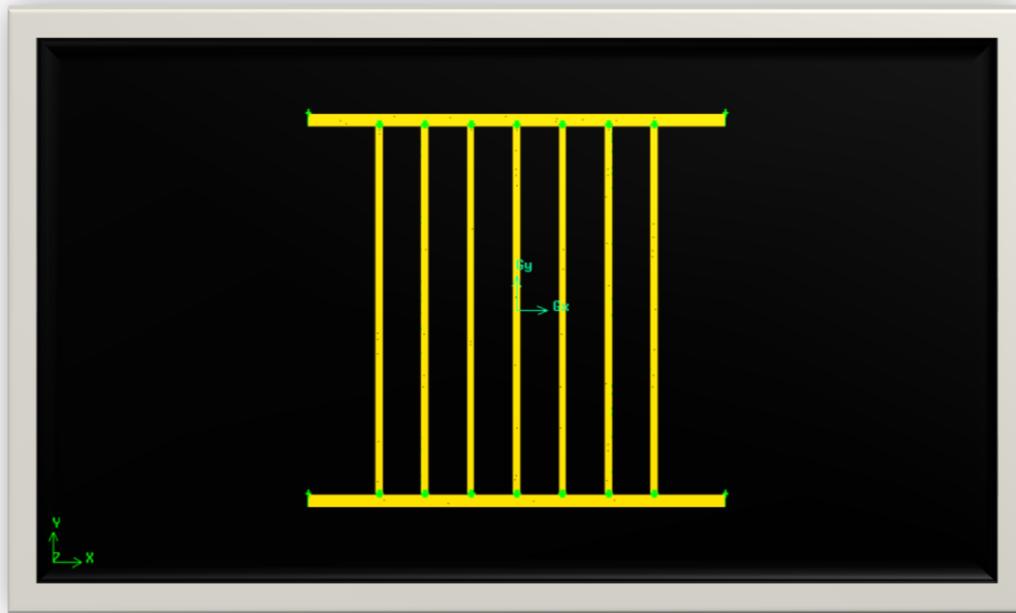
- g. Draw two layers of thermal insulation (fiber glass) and with 95cm length, 95 cm width and 4 cm height.



- h. Draw the glass front cover with 95 cm length, 95 cm width and 0.5 cm thickness.

## 2-Grid generation steps :

1. Draw the two header pipes and the seven riser pipes as explained previously in Gambit steps.
2. Specify boundary layer



3. 3.From file menu choose export mesh which will use in fluent iteration.

### 5.6.2 Fluent

It makes the numerical solution for the model that drawn on Gambit.

#### *The equations that being used in Fluent*

1. Navier-Stokes Equation for the boundary layer

$$\rho \left( \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) = -\nabla p + \mu \nabla^2 \mathbf{v} + \mathbf{f}.$$

Where

$\mathbf{v}$  Is the flow velocity,

$\rho$  is the fluid density,

$p$  is the pressure,

$\mathbf{f}$  Represents body forces (per unit volume) acting on the fluid

$\nabla$  is the del operator.

2. The continuity equation reads:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0.$$

When the flow is at steady-state,  $\rho$  does not change with respect to time. The continuity equation is reduced to:

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0.$$

When the flow is incompressible,  $\rho$  is constant and does not change with respect to space. The continuity equation is reduced to:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0.$$

The velocity components (the dependent variables to be solved for) are typically named  $u$ ,  $v$ , and  $w$ .

3. Energy Equation

$$Q_{In} + Q_{gen} = Q_{Out} + Q_{St}$$

$$Q_{In} = Q_{St}(store) + +$$

$$A \times S \times t = M$$

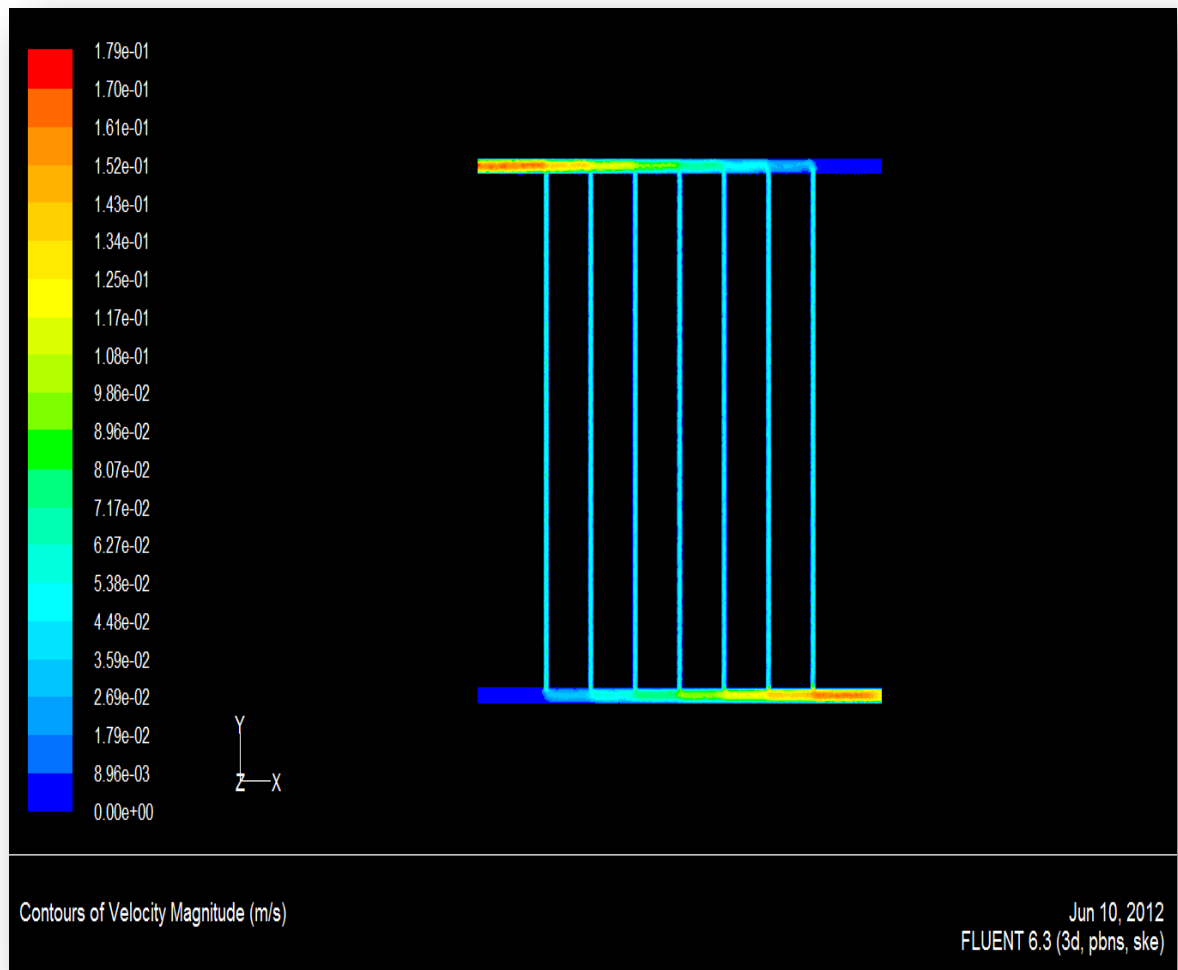
Then do iterations for the case after define all solver models, type of flow (laminar or turbulent), insert energy equation, boundary conditions, materials and operating conditions.

Steps of Fluent:

1. Read the mesh file Press File then Read Case
2. From Grid menu select Check
3. From Define menu press Models then Solver and select (segregated, implicit, 3D, steady and absolute).
4. From Define menu press Energy then from the same menu select viscous.
5. From Define menu select material then liquid water
6. From the same menu press operating conditions select gravity (in y-direction write 9.81 m/s<sup>2</sup>)
7. From Define press Boundary condition
  - a. For Inlet  $x = 0.094$  m/s
  - b. Header 1, Header 2 and Risers select Heat flux
8. From solver menu press control then select
  - a. solution(second order)
  - b. Initialize then init ( $x = 0.094$  m/s)
  - c. Monitor then Residual
9. Plot and choose 1000 iterations
10. From file menu choose Write then case
11. From Solve menu select Iterate
12. From File menu choose Write then Data

## 5.6 CFD Results

Angle 30°

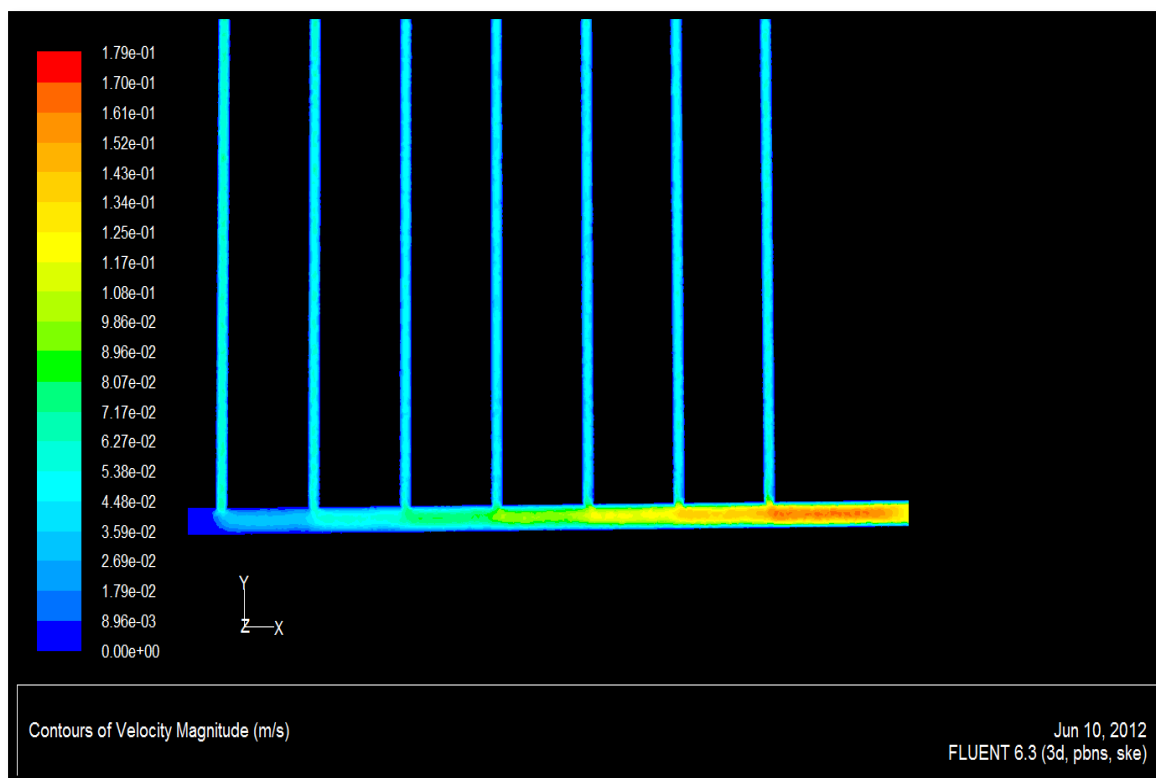


These figures indicate that the velocity enters the tubes at high value then it decreases due to friction between water and pipes.

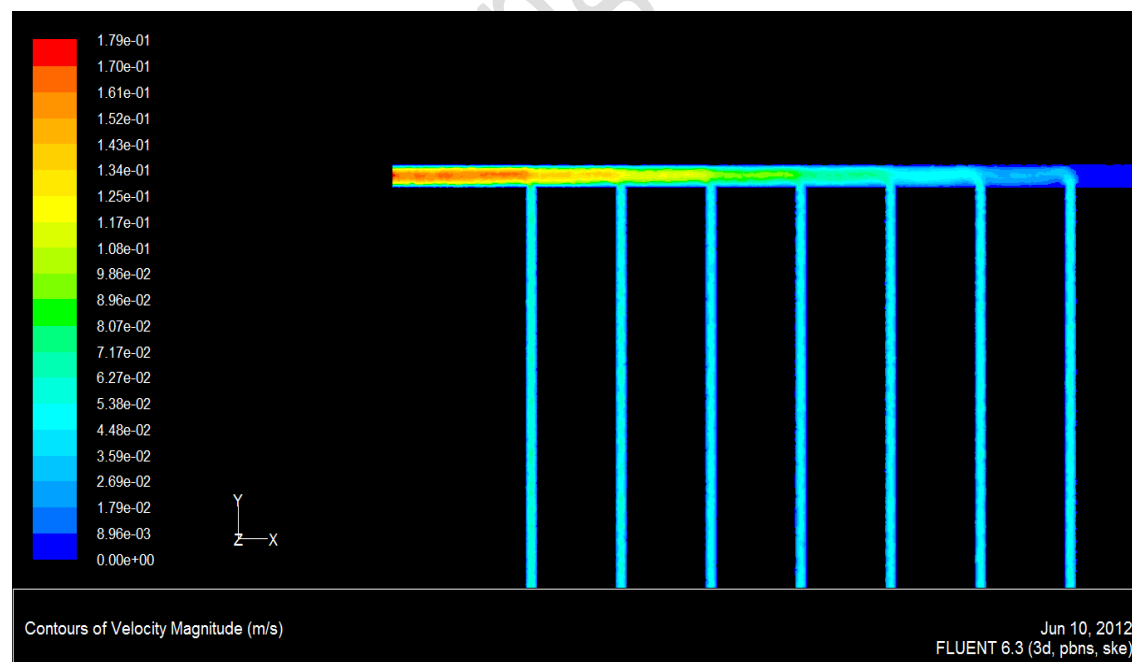
The dark blue indicates that there is no velocity as it is a closed pipes.

The magnitude of velocity at inlet higher than at outlet due to Thermosyphon system.

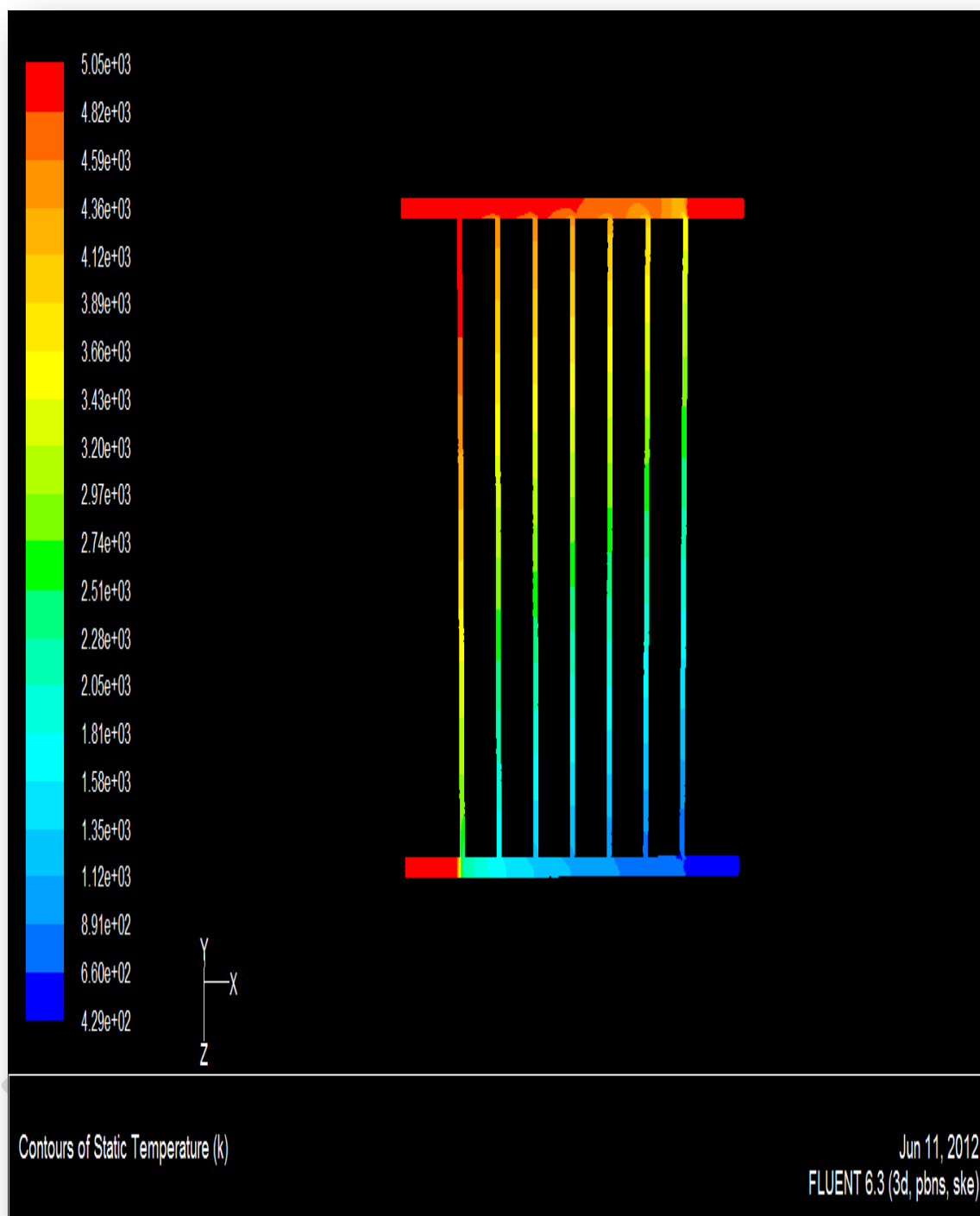
### Velocity at inlet

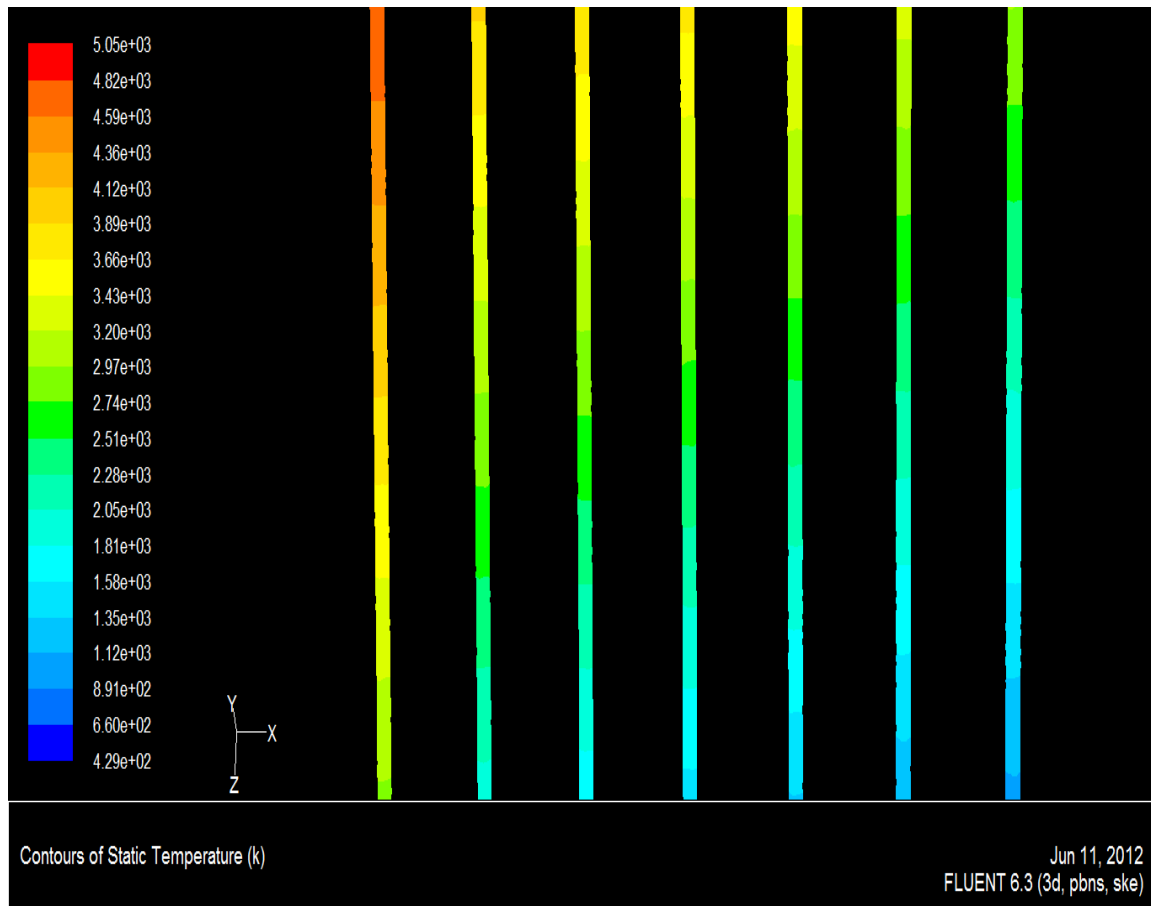


### Velocity at outlet







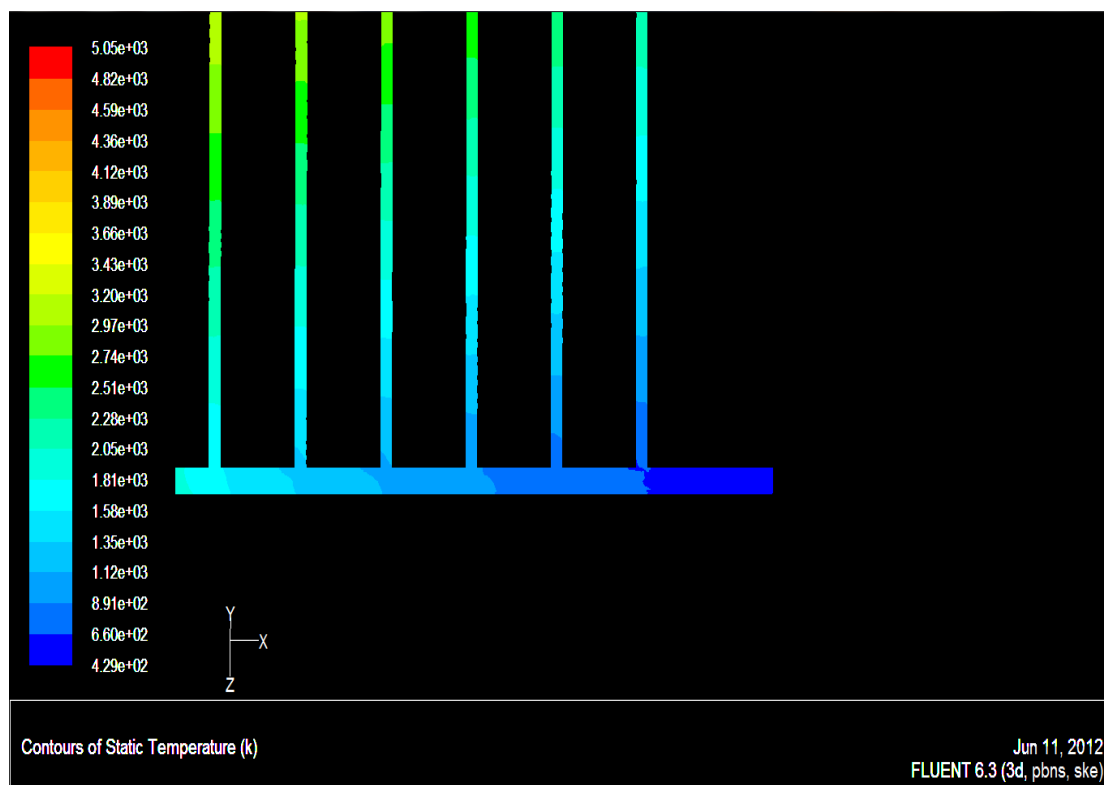


These figures indicate that the temperature at inlet then it increases due to heat flux incident on flat plat collector.

The temperatures at closed ends have high value.

From inlet to outlet pass the temperature increases to give high value at outlet.

**Temp.at inlet**



Temp.at out let

