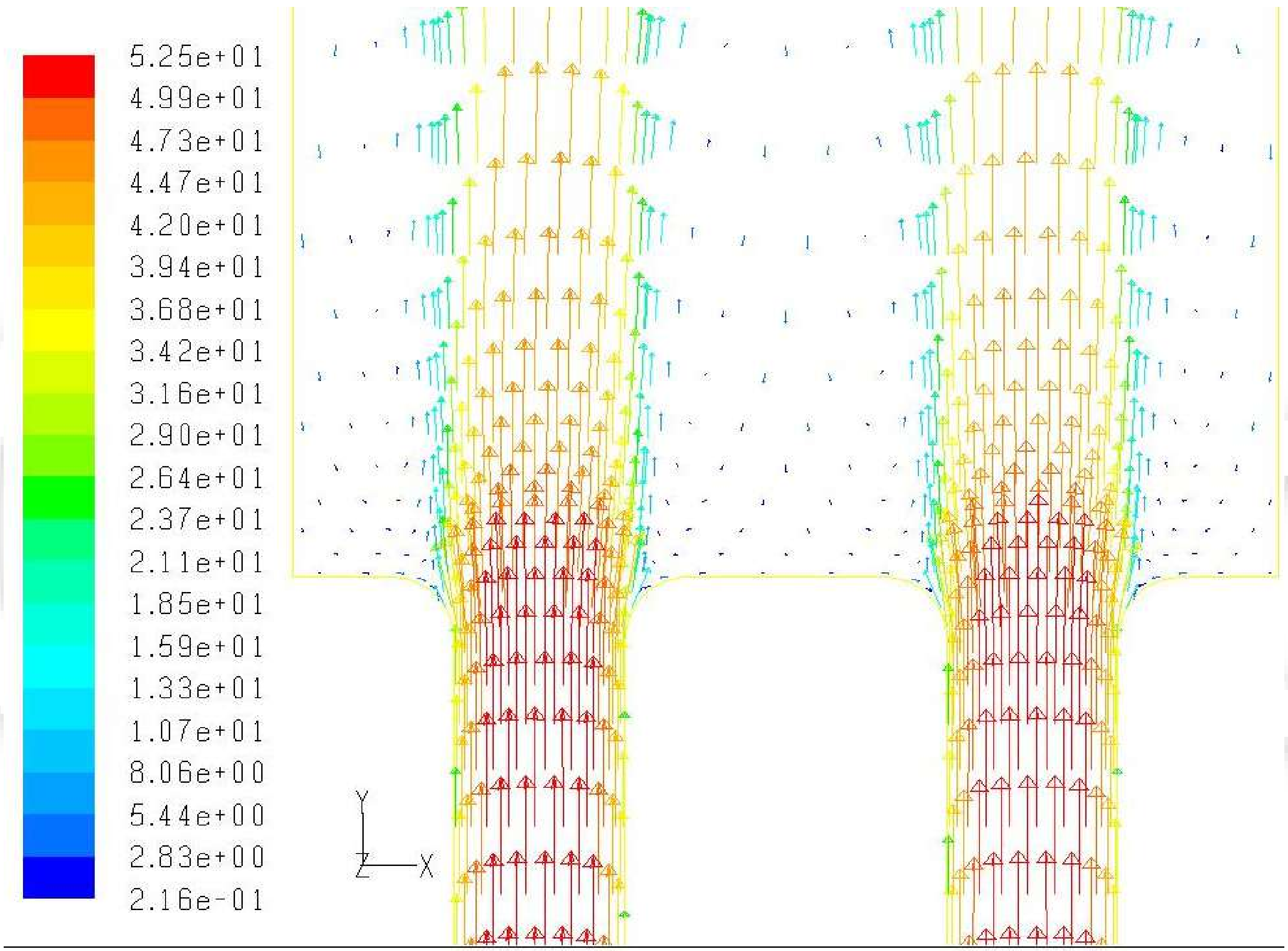
# Post-processing:

# Vector Plot & Streamlines

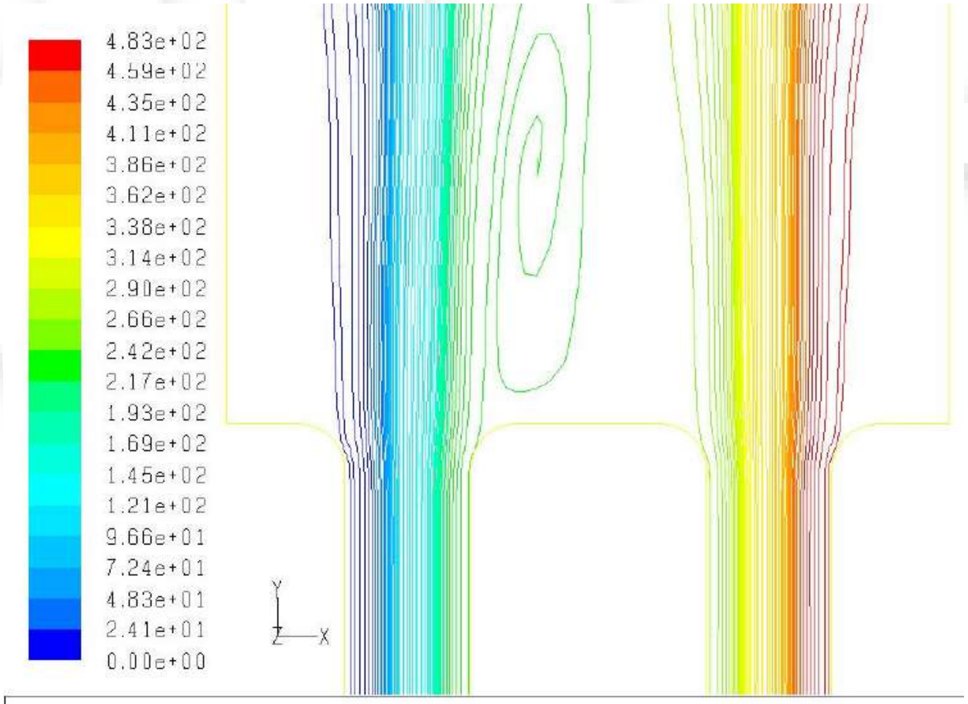
A vector plot is a qualitative representation of spatial velocity vector "colored" by magnitude. The only limitation is that it can be drawn only on surfaces, be it a plane or a 3D twisted surface.




Velocity Vectors Colored By Velocity Magnitude (m/s)

Streamlines are a very good representation of velocity field, at least to beginners in CFD. It is very closely related to velocity vector and any inconsistency may arise only because of post processing interpolation on coarse meshes.

As theoretically explained, tangent to streamlines gives direction of the local velocity field at that point. This statement holds true for velocity vector plot and streamline plots received from post-processing tools.




Pathlines Colored by Particle ID

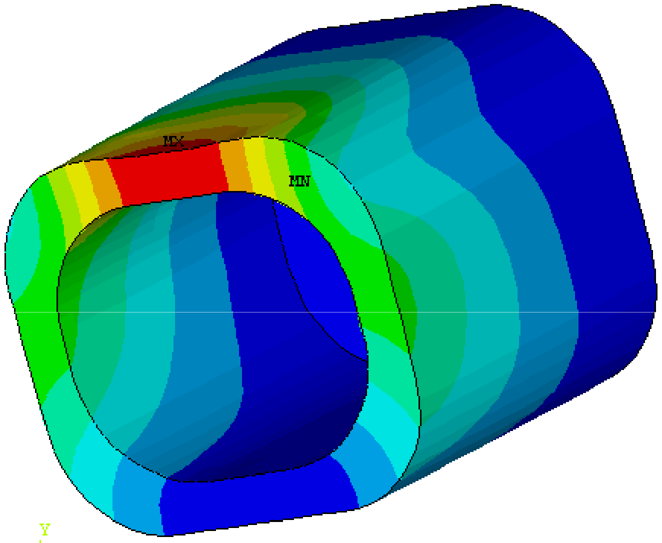
# Post-processing:

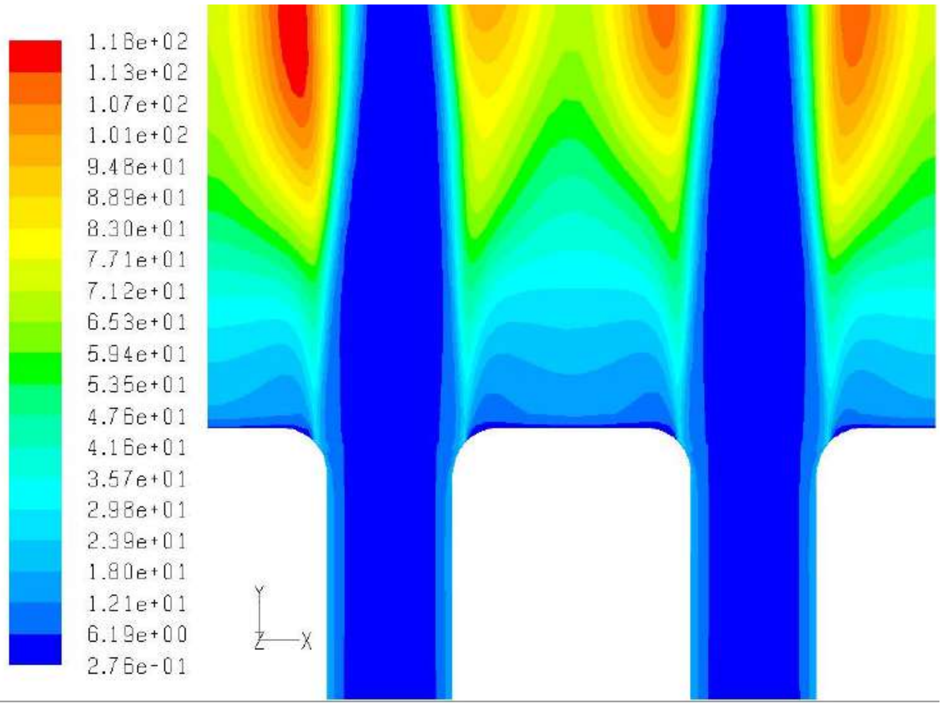
# Contour Plot & Quantitative Data

# Contour Plot – What does it convey?

A contour plot is a "color-band" plot of any variable where range of value is visualized by different color. This is good presentation of information in qualitative-quantitative format.

| |
|---|
| 99-80 |
| 79-60 |
| 59-40 |
| 39-20 |
| 19-0 |





Contours of Turbulent Kinetic Energy (k) (m2/s2)

CFD post-processor allows you to select portions of the domain to be used for visualizing the flow field. The domain portions are called surfaces in Fluent and Planes in CFX, and there are a variety of ways to create them.

Surfaces are required for graphical analysis of 3D problems, since you cannot display vectors, contours, etc. or create an XY plot for the entire domain at once. In 2D you can usually visualize the flow field on the entire domain, but to create an XY plot of a variable in a portion of the interior of the domain, you must generate a surface.

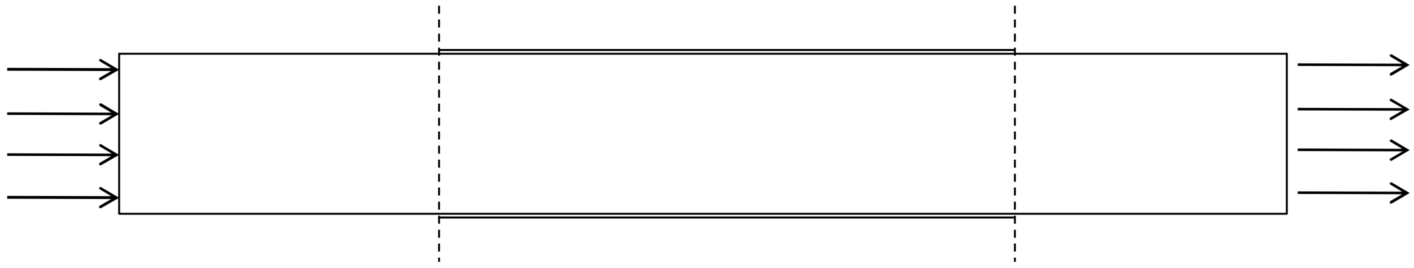
In CFX, one can very easily create planes if they are parallel to either of the co-ordinate plane. From co-ordinate geometry, we know that 3 points always form a plane and a plane is uniquely defined by a point and normal vector. Same techniques are also used in these software to help users create planes at their point of interest.

Iso-surfaces are surface or planes with constant value of a particular variable. Hence, whereas CFX-post has feature to create interactively , same feature is available in Fluent though Iso-surface. Hence, to create a plane parallel to X-Y plane, Z value will remain constant.

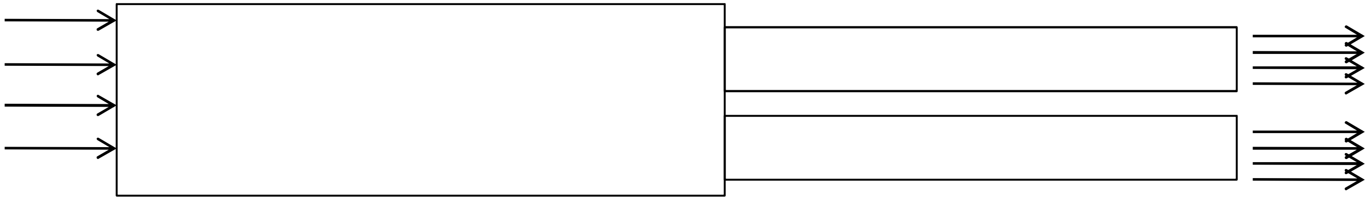
Iso-surface are also useful to visualize the effect of one variable for a given variable over the entire domain. For example, generating an iso-surface based on pressure will allow you to display data for, say velocity, on a surface of constant pressure. You can create an iso-surface from an existing surface or from the entire domain.

The features explained on previous slides are qualitative in nature and gives no data useful for scientific analysis of the result. The discrete values of field variables are obtained by "Area-weighted Averaging" and/or "Mass-weighted Averaging".

For example, if you want to estimate average temperature at a given section of internal flow, mass-weighted average temperature is the correct method.



In the pipe flow example above, the walls at the centre are isothermal. Hence, heat transfer to the fluid should equal rise in internal energy (m. Cp. ΔT) of the fluid. While calculating temperature at the planes shown by dashed vertical lines, one should take mass-weighted average only. The area-weighted average is function of mesh resolution near the wall and may not capture large gradients correctly.



The area-weighted average velocity is not in the ratio of flow areas even though the flow is modeled to be incompressible.

This is because of the error in averaging due to sharp gradient of velocity in boundary layer and mesh not fine enough to capture it. Also note that the narrower sections have 4 boundary layers as compared to 2 boundary layers in the larger section.

# Estimation of Errors

Definition, Methods & Best Practice Guidelines (BPGs)

# Source of Errors:

When you do a numerical solution, you create a “model” and not a “replica”! That is there is an inevitable deviation “between the real flow and the model” and “between numerical solution of governing PDEs and the exact solution of the model”.

This includes errors due to the fact that

1. The exact geometry and operating conditions “modelled” are not same as actual “tangible” geometrical entities.
2. The exact governing flow equations are not solved but are replaced with a physical model of the flow (read RANS) that may not be a good model of reality.

AIAA has attempted to clarify some definitions:

**Error:** A recognisable deficiency that is not due to lack of knowledge.

For example, common known errors are the round-off errors in a computers and the convergence error in an iterative numerical scheme. CFD analyst should be capable of estimating the likely magnitude of the error. It may also arise due to mistakes in input (such as material property variation with temperature).

# Source of Errors:

**Uncertainty:** A potential deficiency that is due to lack of knowledge.

Uncertainties arise because of incomplete knowledge of a physical characteristic, such as the turbulence structure at inlet to a flow domain or because there is uncertainty in the validity of a particular flow model being used. Uncertainty cannot be removed as it is rooted in lack of knowledge (wither physics of the flow or the behaviour of numerical codes).

**Verification:** It is the procedure intended to ensure that the program solves the equations correctly.

**Validation:** This procedure is intended to test the extent to which the model accurately represents reality.

**Calibration:** This procedure to assess the ability of a CFD code to predict global quantities of interest for specific geometries of engineering design interest.

# Error Detection and Prevention in Numerical Solutions

(a) Numerical Errors / **Discretization** Error: Difference between exact equation of the conservation equations & the exact solution of the algebraic set of discretized equations

*Remedy: Grid Convergence Study*

(b) Modeling Errors: Defined as the difference between the actual flow & the exact solution of the mathematical model. This error arises due to following:

- Assumptions made in deriving the transport equation
- Simplifying the geometry of the solution domain

*Remedy: These errors are not known a priori, they can only be evaluated by comparing solutions in which the discretization & convergence errors are negligible with accurate experimental data*

(c) Iteration Errors: Difference between iterative & exact solution of discretized algebraic equation

*Remedy: Exploiting the solver control parameters to get a convergence for different levels of Residual such as 1e-04, 1E-05, 1E-06*

(d) User Error & Application Uncertainties:

Wrong selection of turbulence model / Insufficient Information about BC setting

Poor quality grid generation and **Boundary Layer Resolution**

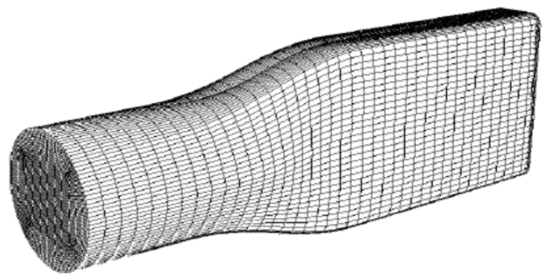
*Remedy: These can be minimized through experience, Best Practice Guidelines & optimization of resources*

# Approximations in Numerical Solutions: Discretization

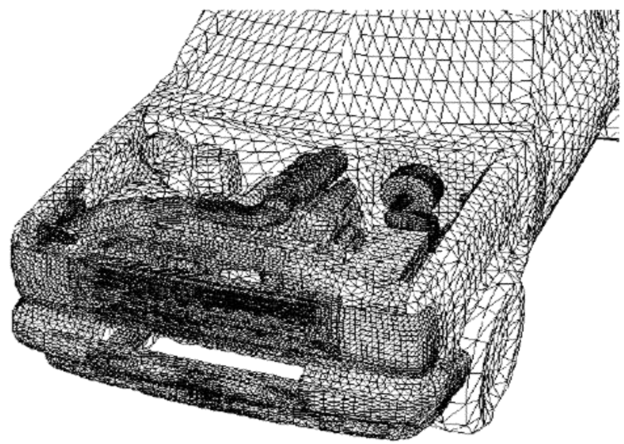
**Discretization:** Close form mathematical equations like Navier Stokes equations has infinite degree of freedom that is infinite values of continuum in a given domain. For numerical solutions of these equations, we need to reduce these equations which prescribes values at only a finite number of DISCRETE points in the domain. This process if called DISCRETIZATION of the governing differential equation.

# Tri/Tet vs. Quad/Hex Meshes

- For **simple** geometries, quad/hex meshes can provide high-quality solutions with fewer cells than a comparable tri/tet mesh.
  - Align the gridlines with the flow.



- For **complex** geometries, quad/hex meshes show no numerical advantage, and you can save meshing effort by using a tri/tet mesh.



# Quality Assurance Procedure-1

**MESH CONVERGENCE STUDY:** The formal method of establishing mesh convergence requires a curve of a critical result parameter (typically some kind of coefficient such as skin friction coefficient) in a specific location, to be plotted against some measure of mesh density. At least three convergence runs will be required to plot a curve which can then be used to indicate when convergence is achieved or, how far away the most refined mesh is from full convergence. However, if two runs of different mesh density give the same result, convergence must already be achieved and no convergence curve is necessary.

Define target variables (usually scalars like Force, Drag Coefficient, Heat Flux, HTC, MAX Temperature, etc). Check the variation in target variable (e.g. mass flow rate at any plane) for various refinements of mesh, keep ITERATION ERROR limit constant say 0.00001

ITERATION ERROR - Check variation in target variable for different convergence limit say 0.001, 0.0001, 0.00001 for a given mesh refinement.

# Quality Assurance Procedure-2

**Study Effect of Inlet B.C.:** Inlet velocity profile is very important to study the accuracy of CFD result for a particular application. In any commercial software, the inputs required at inlet are

- Mean Flow Parameters which can be
  - Constant Average Velocity
  - Velocity Profile from a Measurement
  - Velocity Profile as per Power Law such as $u = U\,(y/d)^{1/n}$, where n = 6, 7, 8, … depending upon Re value
- Turbulence Parameters such as
  - Value and Method to Specify TKE, Turbulence Kinetic Energy, k
  - Value and Method to Specify Turbulent Eddy Dissipation, $\varepsilon$

These sensitivity study are "particularly" important in cases where Separation and Reattachment are likely to occur. **For Example, in case of a flow over Backward Facing Step, there is decreases in location of reattachment length as the turbulence intensity increases and is very sensitive to TI value specified at inlet.**

# Quality Assurance Procedure-3

**Study Effect of Wall Treatment:** The "wall treatment" is a semi-empirical treatment of for steep gradient of velocity and turbulence parameters near wall. Unfortunately, there is no "universal wall-function" that is the formulation which can be applied to all sort of flow conditions without loss of accuracy.

These wall functions affects the result in two ways:

1. They are very sensitive to near wall mesh resolution and in most of the cases, the recommended value of Y+ >= 30.

2. Their accuracy increases with Reynolds number, that is, they tend to produce correct result only at very high Re numbers.

Though "Standard Wall Treatment" available in commercial software are the most prevalent and stable option. However, this does not always yield correct result. Some more refined versions of Wall Treatment such as "Scalable in CFX" and "Enhanced in Fluent" must be explored for its suitability for the particular flow configuration being investigated.