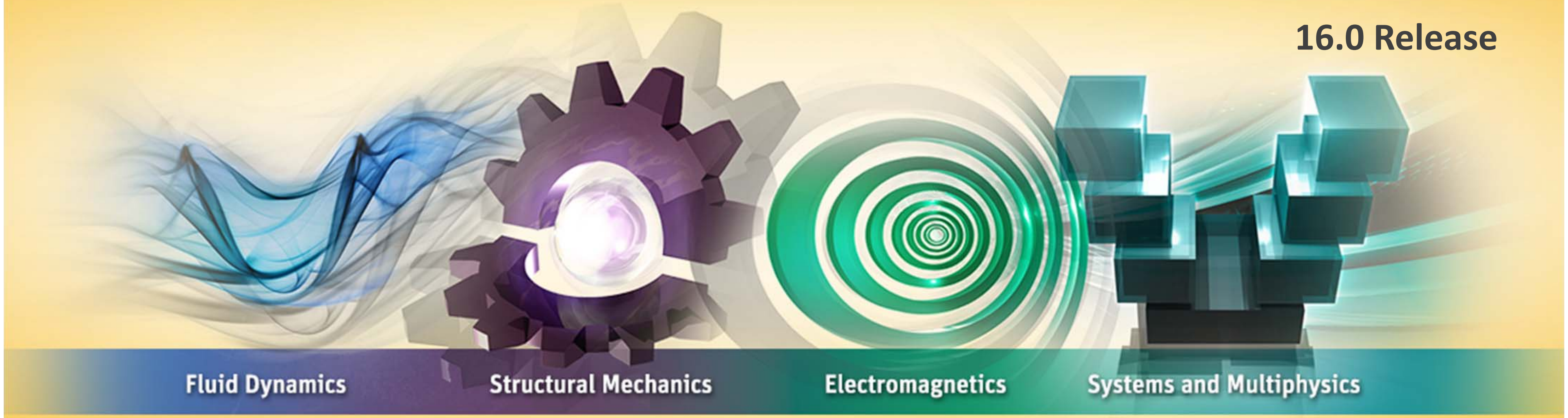
Lecture 9:
Best Practice Guidelines

16.0 Release



Introduction to ANSYS Fluent

Introduction

Lecture Theme:

The accuracy of CFD results can be affected by different types of errors. By understanding the cause of each different error type, best practices can be developed to minimize them. Meshing plays a significant role in the effort to minimize errors.

Learning Aims:

You will learn:

- Four different types of errors
- Strategies for minimizing error
- Issues to consider during mesh creation such as quality and cell type
- Best practices for mesh creation

Learning Objectives:

You will understand the causes of error in the solution and how to build the mesh and perform the simulation in a manner that will minimize errors

Motivation for Quality

CFD-Results are used for many different stages of the design process:

- Design & optimization of components and machines
- Safety analyses
- Virtual prototypes

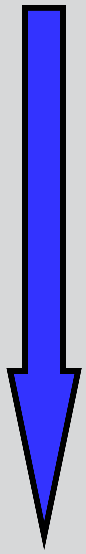
When undertaking a CFD model, consideration should be given to the purpose of the work:

- What will the results be used for?
- What level of accuracy will be needed?

Different Sources of Error

There are several different factors that combine to affect the overall solution accuracy. In order of magnitude:

- **Round-off errors**
  - Computer is working to a certain numerical precision
- **Iteration errors**
  - Difference between 'converged' solution and solution at iteration 'n'
- **Solution errors**
  - Difference between converged solution on current grid and 'exact' solution of model equations
  - 'Exact' solution → Solution on infinitely fine grid
- **Model errors**
  - Difference between 'exact' solution of model equations and reality (data or analytic solution)

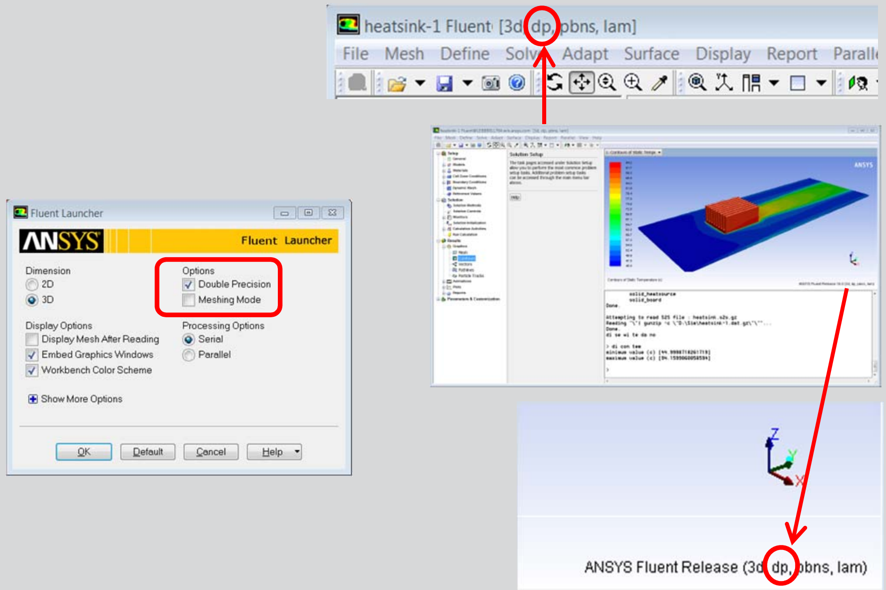


Round-Off Error

Inaccuracies caused by machine round-off:

- High grid aspect ratios
- Large differences in length scales
- Large variable range

Tip: Look for "dp" in the title bar of the Fluent window or the lower right corner of the graphics window to check if your current session is using double precision



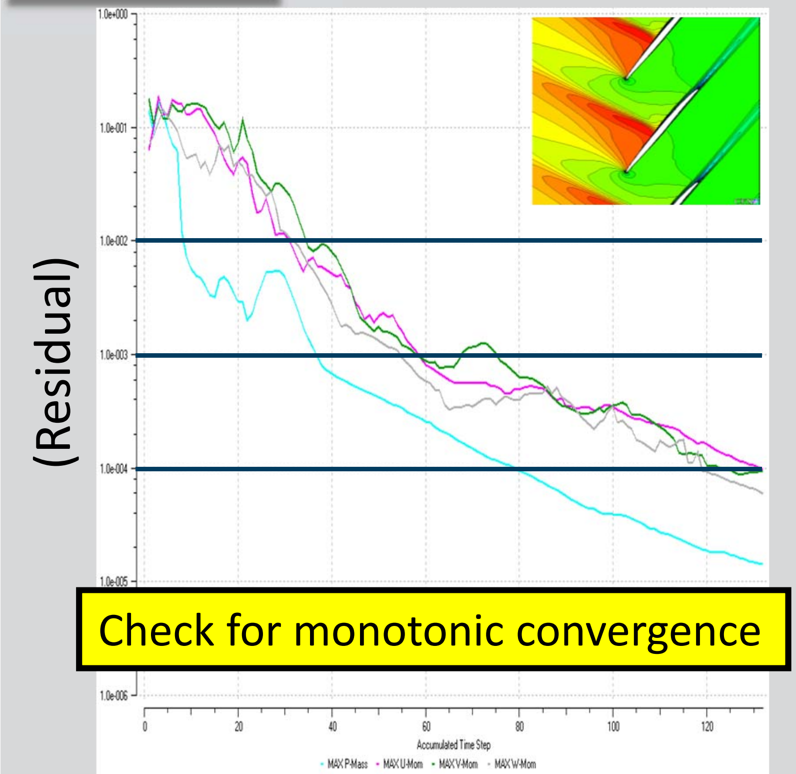
How to identify if round-off error is a problem:

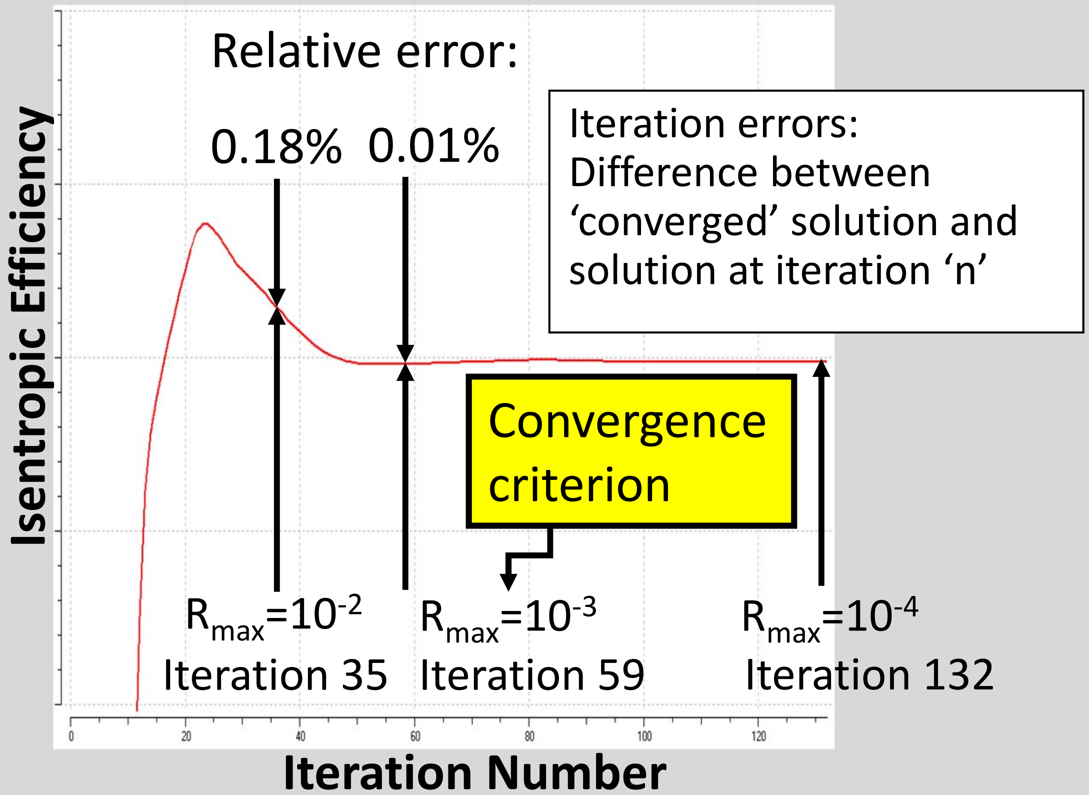
- Calculate with **double precision** if your case meets the above criteria
- Compare results with a solution that has been calculated with single precision
- If important quantities (target variables) are different, double precision should be used for subsequent calculations

Iteration Error - Best Practice

- **Define quantity or quantities of interest for your simulation (target variables):**
  - Head rise
  - Efficiency
  - Mass flow rate
  - …
- **Select convergence criterion for the residuals**
- **Plot the value of the quantities of interest as the solution iterates**
- **Select a tighter convergence criterion and continue iterating and plotting**
- **Repeat until the values of the quantities of interest no longer change**
  - This will identify what residual level it is necessary to achieve in order to ensure the solution is free from iteration error
- **Monotonic convergence of residuals (next slide) is desirable, although not always possible**
- **Report mass and energy fluxes to ensure these are being conserved**

Iteration Error Example: 2D Compressor Cascade





Discretization Error

All <u>discrete</u> methods have solution errors:

- Finite volume methods
- Finite element methods
- Finite difference methods
- ...

Difference between solution on a given grid and “exact‘ solution on an infinitely fine grid

Exact solution not available → <u>Discretization error estimation</u>

Introduction | **Error Types** | Best Practices for Meshing | Summary

Discretization Error Estimation

Impinging jet flow with heat transfer

2-D, axisymmetric

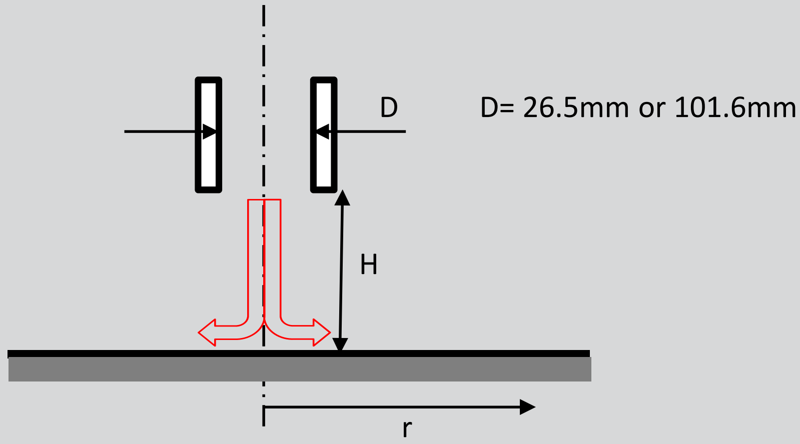
Compared Grids:

- 50 × 50 → 800 × 800

SST turbulence model

Discretization schemes:

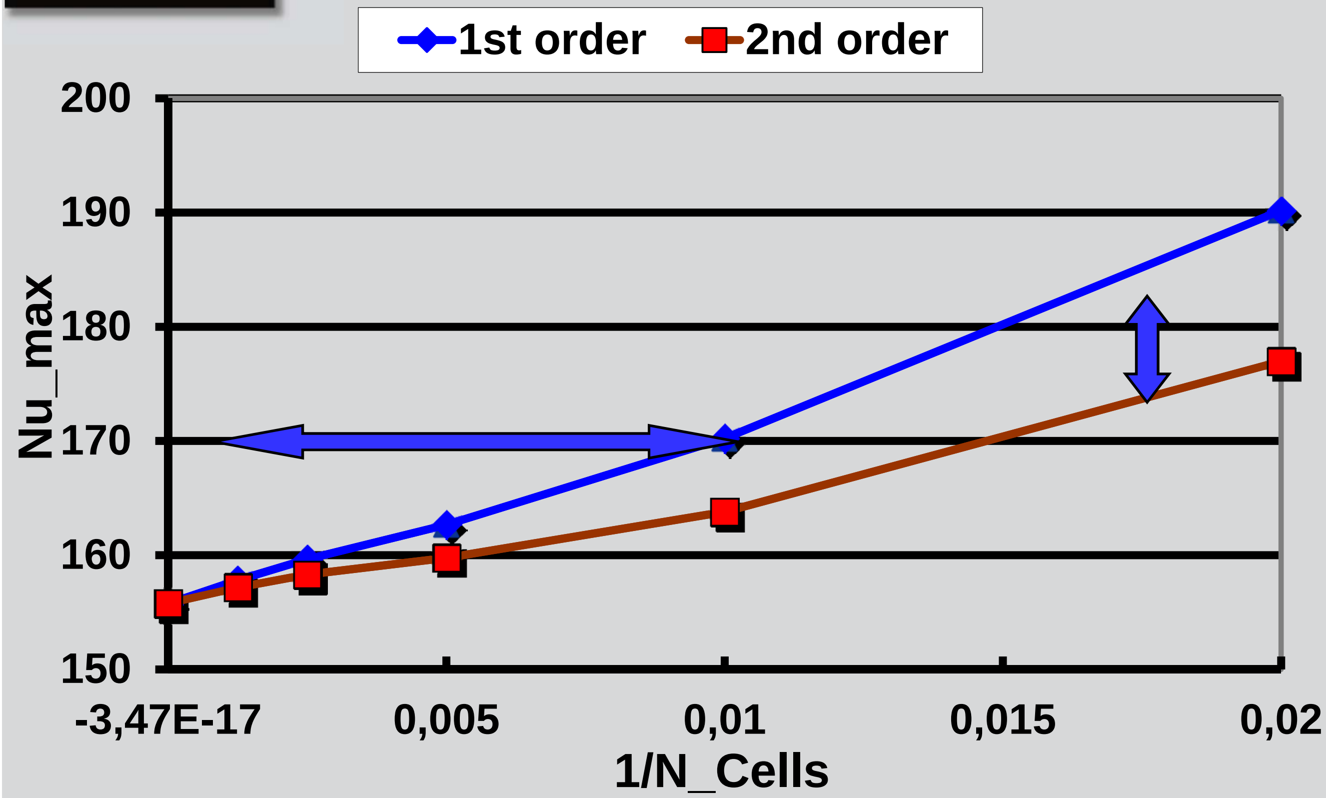
- 1st order Upwind
- 2nd order Upwind



- Target quantities:
  - Heat transfer
  - Maximum Nusselt number

Introduction | Error Types | Best Practices for Meshing | Summary

Discretization Error Estimation



The plot shows

- If the grid is fine enough, 1st and 2nd order solutions are the same
- On coarser meshes, the 2nd order solution is closer to the final solution

Practical alternatives for industrial cases are:

- Compare solutions from different order schemes
- Compare solutions on locally or regionally refined meshes

Model Errors

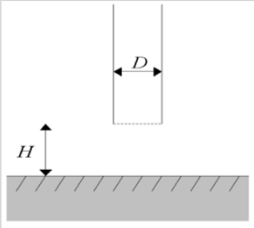
Inadequacies of (empirical) mathematical models:

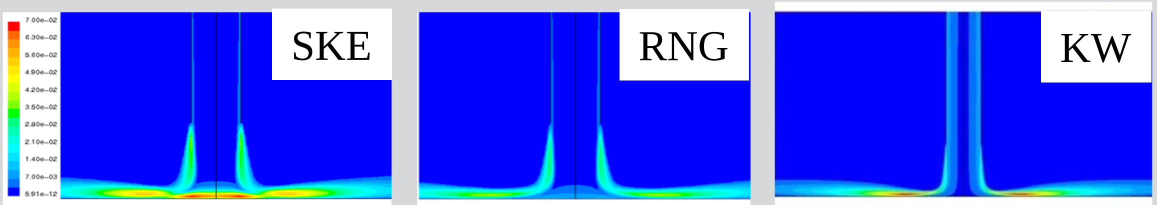
- Base equations (Euler vs. RANS, steady-state vs. unsteady-state, …)
- Turbulence models
- Combustion models
- Multiphase flow models
- …

Discrepancies between data and calculations remain, even after all numerical errors have become insignificant!

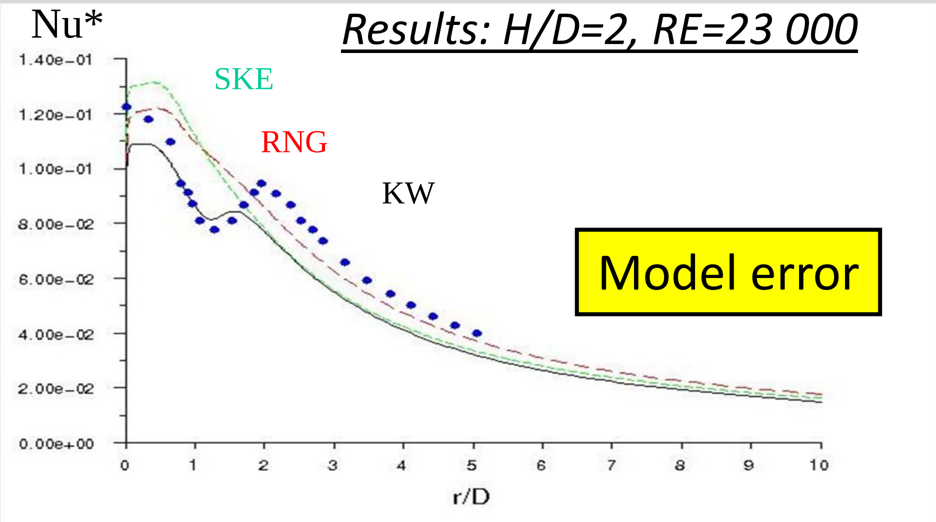
ANSYS

Model Error: Impinging Jet





← TKE Contours



- **Note how the predictions differ depending on which turbulence model is used**
- **The k-ω model (KW) performs better than the standard (SKE) or RNG k-ε models in this case**
  - **The k-ε based models overestimate the production of turbulence at the stagnation point, causing the predicted Nusselt number to be too high**

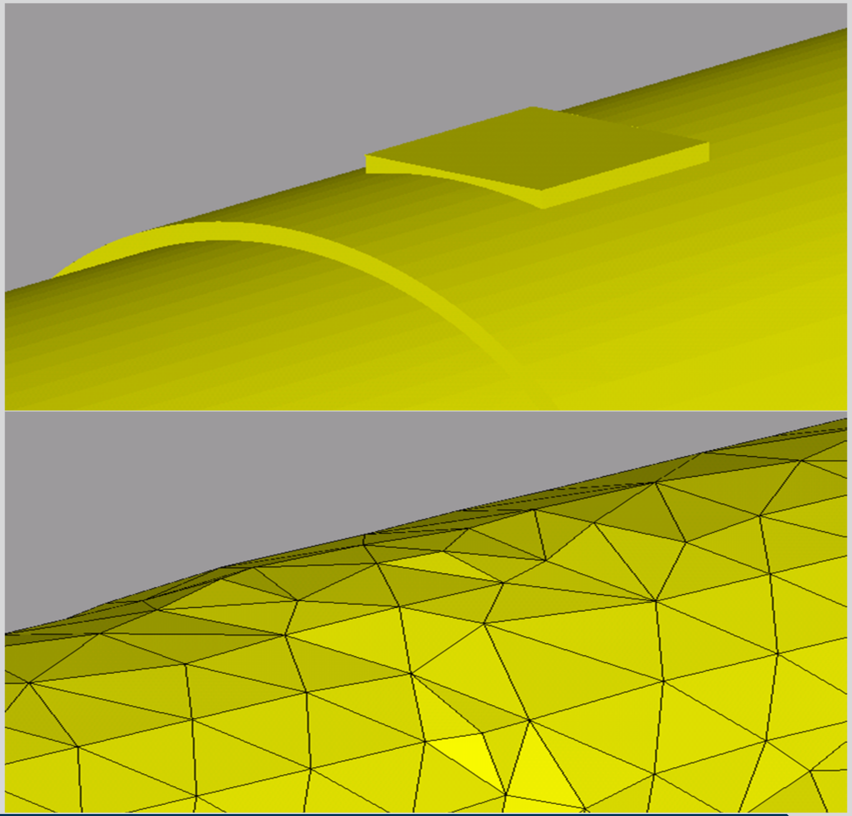
Systematic Errors

Discrepancies remain

- **Even if numerical and model errors are insignificant**

'Systematic errors':

- **Approximations of:**
  - Geometry
  - Component vs. machine
  - Boundary conditions
  - Fluid and material properties, …



Try to 'understand' application and physics

Document and defend assumptions !

Perform <u>uncertainty analysis</u>

Meshing Best Practice Guidelines

Choosing your mesh strategy depends on

1.ACCURACY

2. EFFICIENCY

3. EASINESS TO GENERATE

Desired mesh quality
What is the maximum skewness and aspect ratio you can tolerate?

Desired cell count
- Low cell count for resolving overall flow features vs High cell count for greater details

Time available
- Faster Tet-dominant mesh vs crafted Hex/hybrid mesh with lower cell count

Goal: Find the best compromise between accuracy, efficiency and easiness to generate

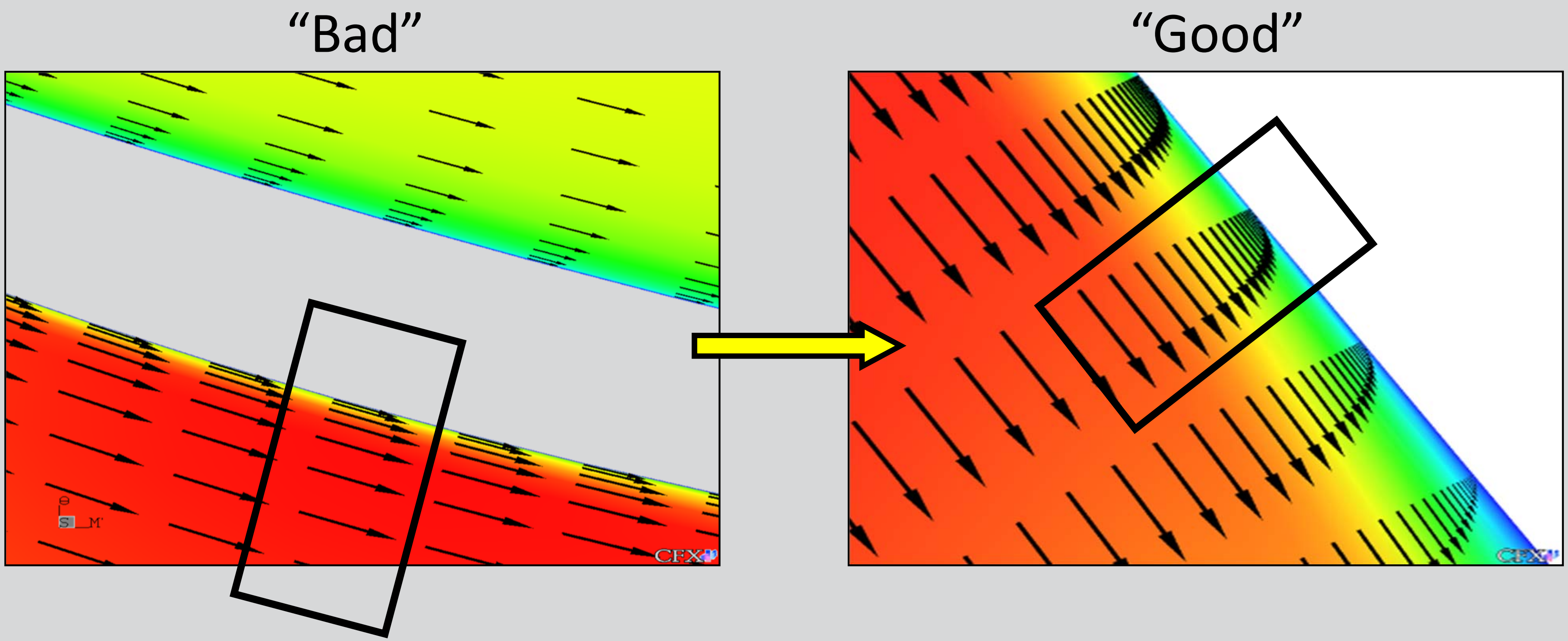
Meshing: Capture Flow Physics

- **Grid must be able to capture important physics:**
  - Boundary layers
  - Heat transfer
  - Wakes, shock
  - Flow gradients

- **Recommended meshing guidelines for boundary layers**
  - Both the velocity and thermal boundary layers must be resolved
  - There should be a minimum of 10-15 elements across the boundary layer thickness
  - The mesh expansion ratio in the wall normal direction should be moderate:
    - $\leq$ 1.2 … 1.3
  - y+ $\approx$ 1 for heat transfer and transition modeling

Meshing: Capture Flow Physics

- **Example: Velocity profiles at airfoil**

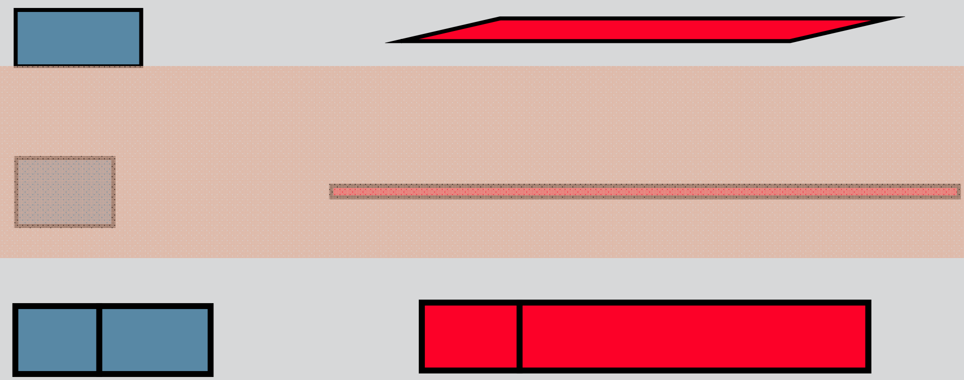


Mesh Quality

A good mesh depends on :

Good | Not Good

- **Cell not too distorted**
- Cell not too stretched
- **Smooth Cells transition**



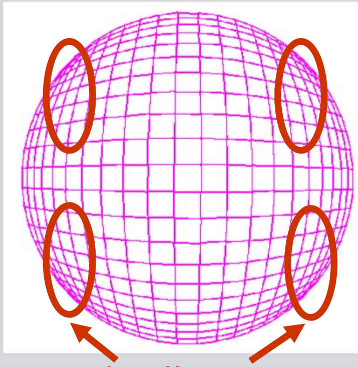
Mesh Quality

Grid generation:

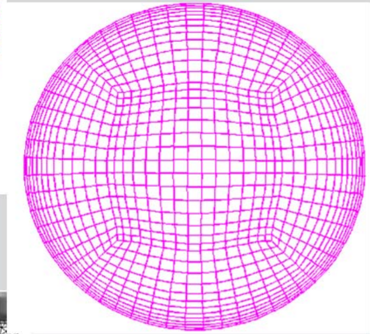
- Scalable grids
- Skewness < 0.95 (accuracy, convergence)
  - also worst Orthogonal Quality > .01 and average value much higher
- Aspect ratios < 100
- Expansion ratios < 1.5 …2
- Capture physics based on experience (shear layers, shocks)
- Angle between grid face & flow vector
- Concrete, quantitative recommendations for these factors presented in the Introduction to ANSYS Meshing course are included in the appendix of this presentation

Grid refinement:

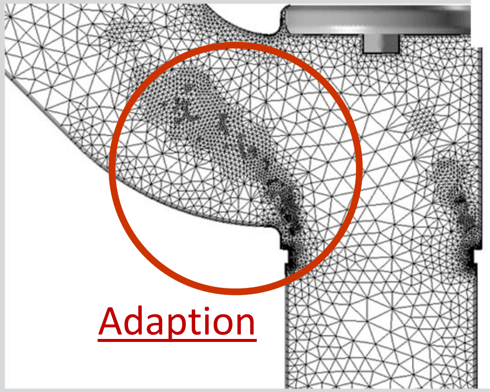
- Manual, based on error estimate
- Automatic adaptive based on 'error sensor'



Bad cells



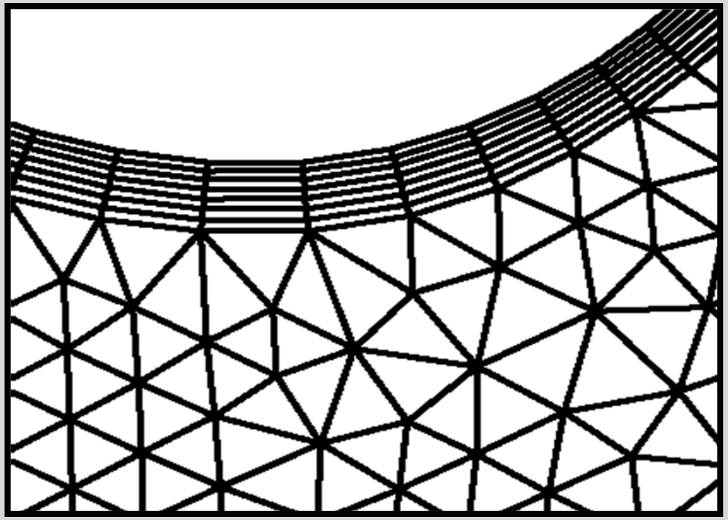
No Bad cells



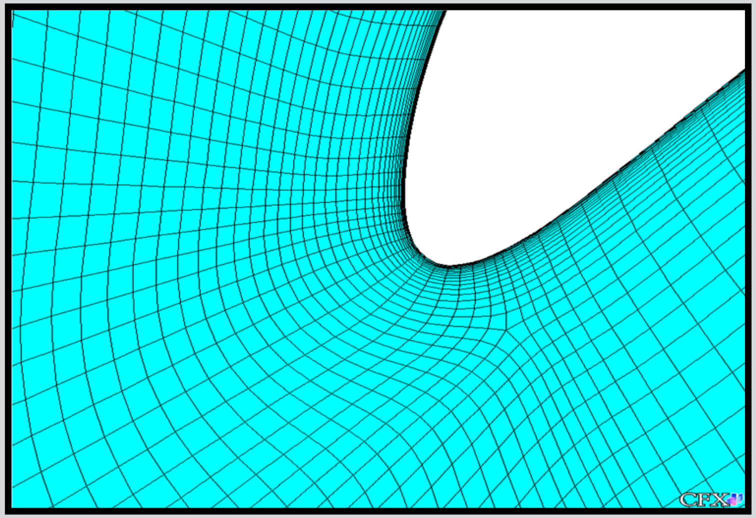
Adaption

Mesh Quality

Avoid sudden changes in mesh density



Not good

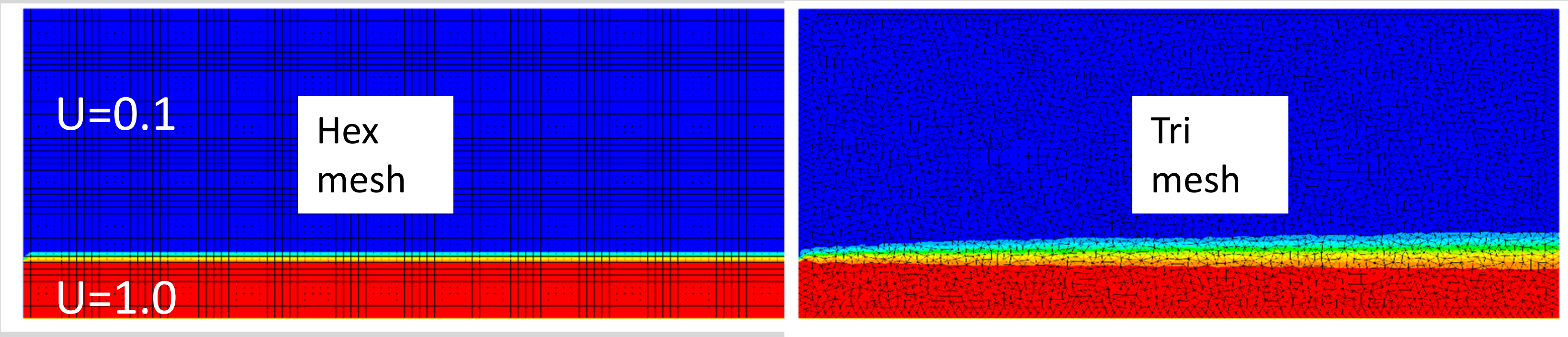


Good

Hex vs Tet Mesh : Accuracy Comparison

- Direction of the flow well known

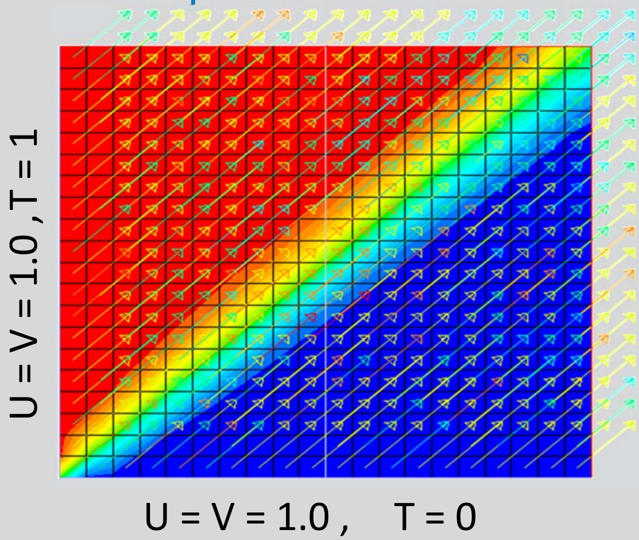
⇨ Quad/Hex aligned with the flow are more accurate than Tri with the same interval size

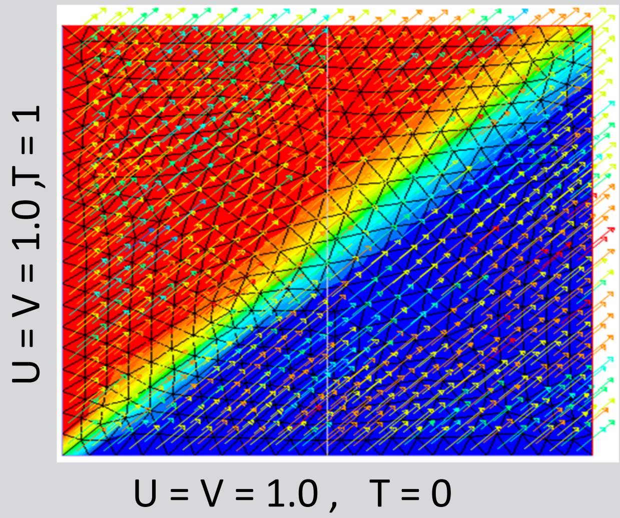


Contours of axial velocity magnitude for an inviscid co-flow jet

Hex vs Tet Mesh : Accuracy comparison

- For complex flows without dominant flow direction, Quad and Hex meshes lose their advantage
  ⇨ Quad & Tri equivalent





Contours of temperature for inviscid flow

Summary

- **Try to ‘understand’ application and physics of the application**
- **Distinguish between numerical, model and other errors**
- **Document and defend assumptions**
  - Geometry
  - Boundary conditions
  - Flow regime (laminar, turbulent, steady-state, unsteady-state, …)
  - Model selection (turbulence, …)
- **Sources of systematic error**
  - Approximations
  - Data
- **Accuracy expectations vs. assumptions?**

Resources

ERCOFTAC SIG: ‚Quantification of Uncertainty in CFD‘

Roache, P.J., *Verification and Validation in Computational Science and Engineering*, Hermosa Publishers, 1998

On the ANSYS customer portal, search for "best practice"

Introduction | Error Types | Best Practices for Meshing | Summary