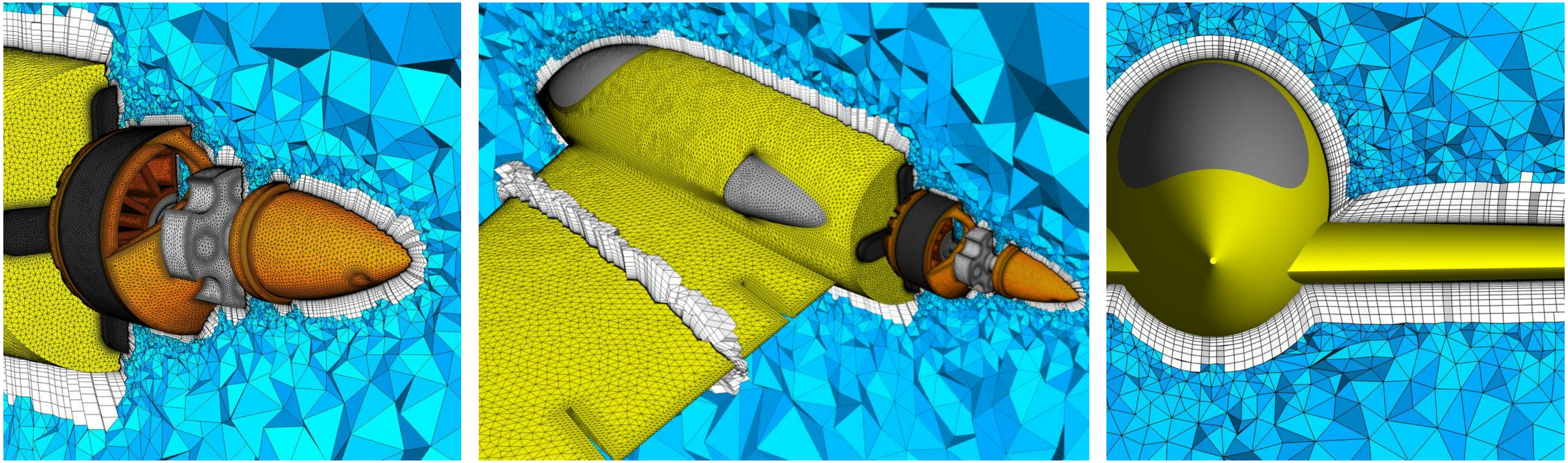
What is a good mesh?

- There is no written theory when it comes to mesh generation. Basically, the whole process depends on user experience and good standard practices.
- A standard rule of thumb is that the elements shape and distribution should be pleasing to the eye.

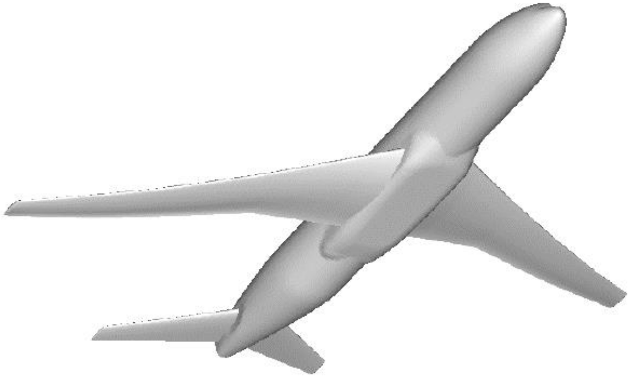


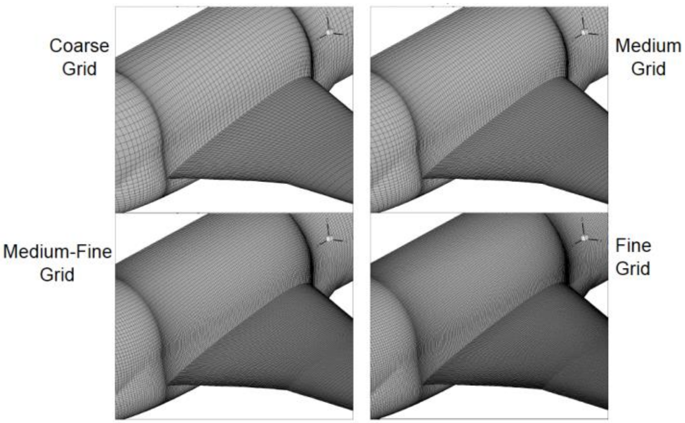
22rd IMR Meshing Maestro Contest Winner
Travis Carrigan, John Chawner and Carolyn Woeber. Pointwise.
http://imr.sandia.gov/22imr/MeshingContest.html

What is a good mesh?

- The user can rely on grid dependency studies, but they are time consuming and expensive.

	Coarse mesh	Medium mesh	Fine mesh	Extra fine mesh
Cells	≈ 3 500 000	≈ 11 000 000	≈ 36 000 000	≈ 105 000 000
C_D	0.0282	0.0270	0.0268	0.0269
C_M	-0.0488	-0.0451	-0.0391	-0.0391





Results from AIAA CFD Drag Prediction Workshop

What is a good mesh?

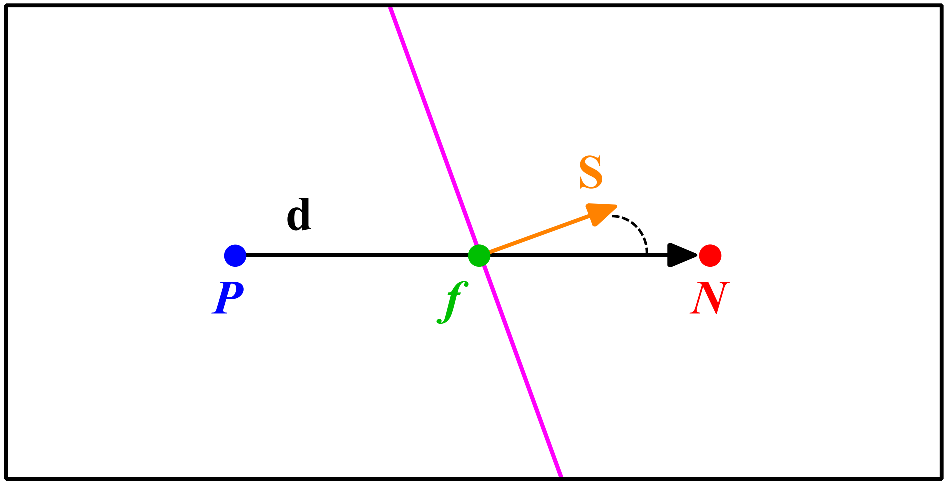
- No single standard benchmark or metric exists that can effectively assess the quality of a mesh, but the user can rely on suggested best practices.
- Hereafter, we will present the most common mesh quality metrics:
  - Orthogonality.
  - Skewness.
  - Aspect Ratio.
  - Smoothness.

- After generating the mesh, we measure these quality metrics and we use them to assess mesh quality.
- Have in mind that there are many more mesh quality metrics out there, some of them are not very easy to interpret (e.g., jacobian matrix, determinant, flatness, equivalence, condition number, and so on).
- It seems that it is much easier diagnosing bad meshes than good meshes.

What is a good mesh?

Mesh quality metrics. Mesh orthogonality

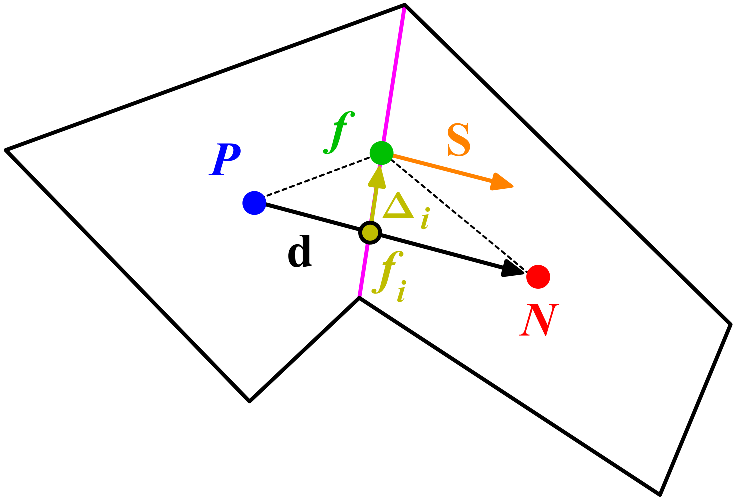
- Mesh orthogonality is the angular deviation of the vector **S** (located at the face center *f*) from the vector **d** connecting the two cell centers **P** and **N**. In this case is 20°.
- Affects the gradient of the face center *f*.
- It adds numerical diffusion to the solution.
- It mainly affects the diffusive terms.



What is a good mesh?

Mesh quality metrics. Mesh skewness

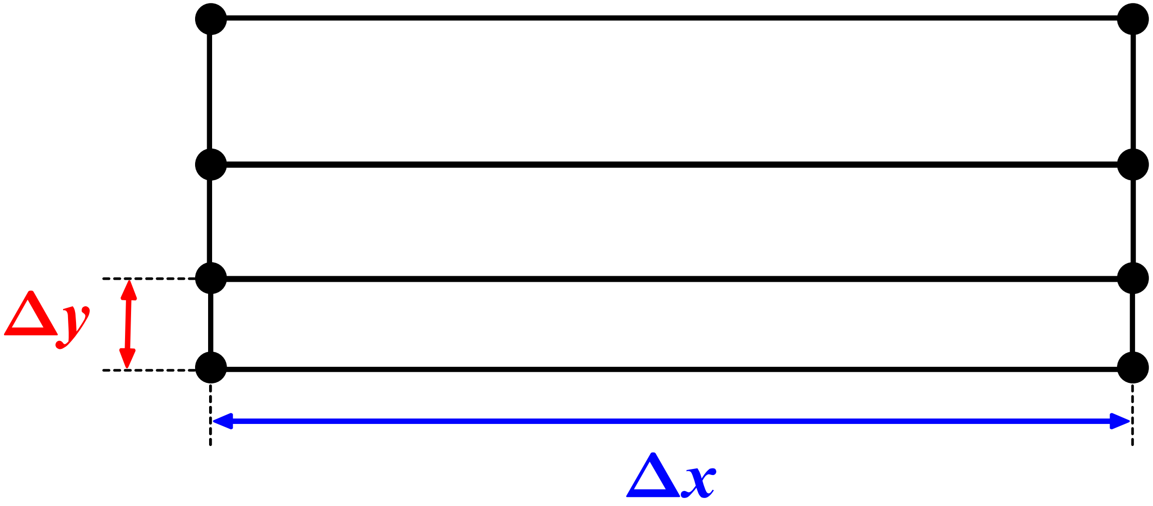
- Skewness is the deviation of the vector **d** that connects the two cells **P** and **N**, from the face center ***f***.
- The deviation vector is represented with Δ and f_i is the point where the vector **d** intersects the face ***f*** .
- Affects the interpolation of the cell centered quantities to the face center ***f***.
- It adds numerical diffusion and wiggles to the solution.
- It affects the convective and diffusive terms.



What is a good mesh?

Mesh quality metrics. Mesh aspect ratio AR

- Mesh aspect ratio AR is the ratio between the longest side Δx and the shortest side Δy .
- Large AR are ok if gradients in the largest direction are small.
- High AR smear gradients.

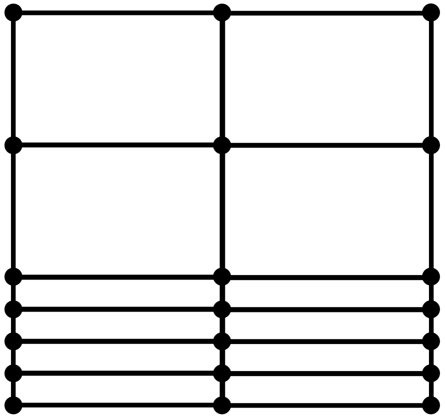


What is a good mesh?

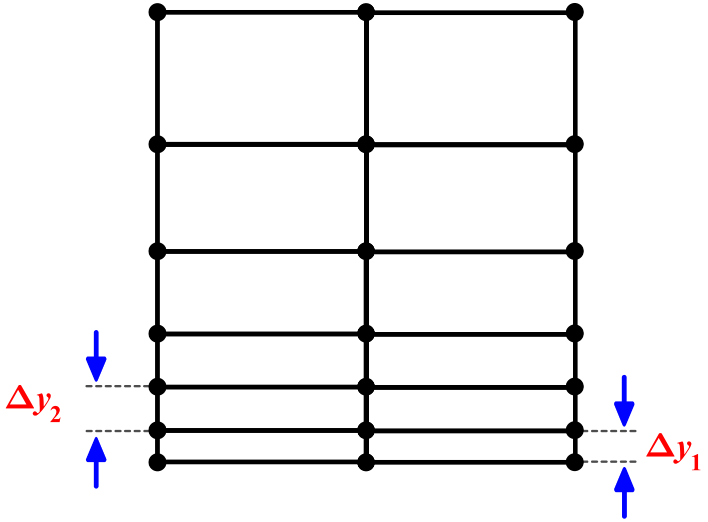
Mesh quality metrics. Smoothness

- Smoothness, also known as expansion rate, growth factor or uniformity, defines the transition in size between contiguous cells.
- Large transition ratios between cells add diffusion to the solution.
- Ideally, the maximum change in mesh spacing should be less than 20%:

$$\frac{\Delta y_2}{\Delta y_1} \leq 1.2$$



Steep transition

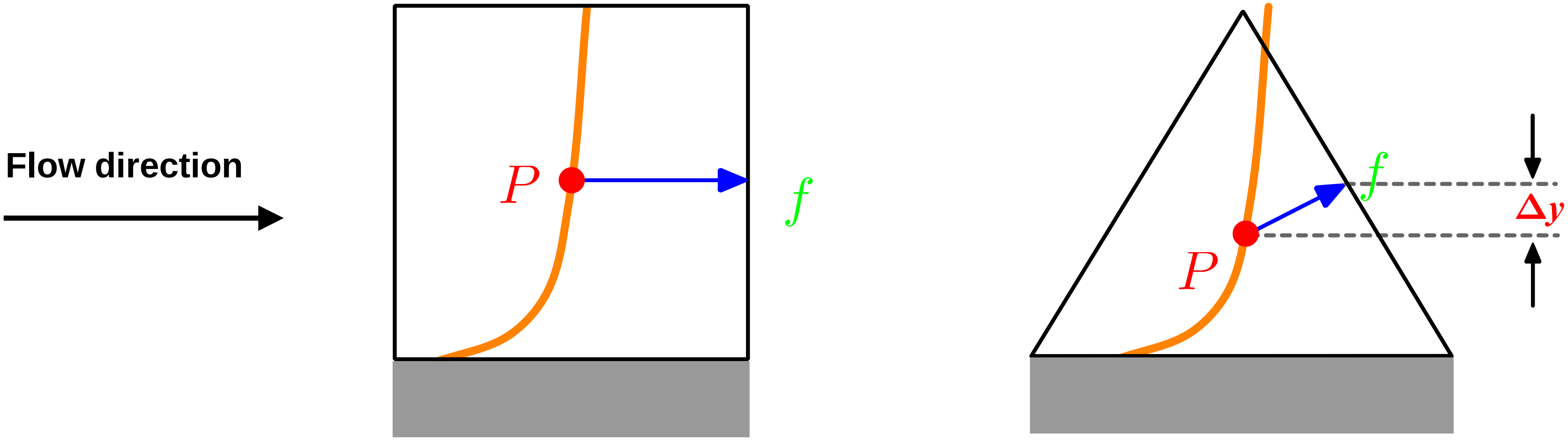


Smooth transition

What is a good mesh?

Mesh quality metrics. <u>Element type close to the walls - Cell/Flow alignment</u>

- Hexes, prisms, and quadrilaterals can be stretched easily to resolve boundary layers without losing quality.
- Triangular and tetrahedral meshes have inherently larger truncation error.
- Less truncation error when faces aligned with flow direction and gradients.



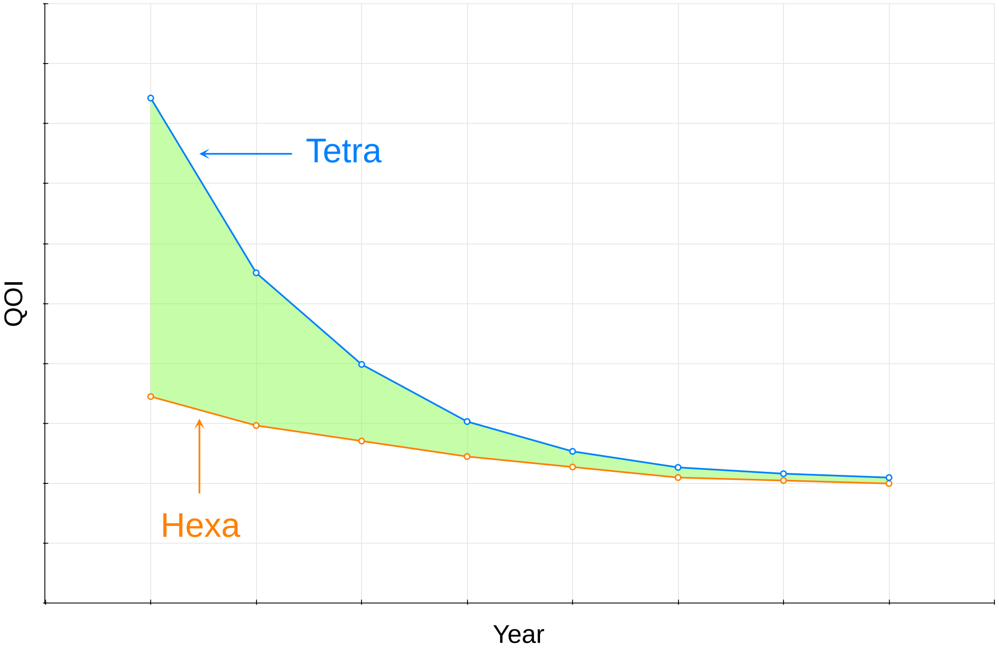
$$\phi_f = \phi_P + \cancel{\frac{\partial \phi}{\partial y}\Delta y} + \cancel{\mathcal{O}(\Delta y^2)}$$

$$\phi_f = \phi_P + \frac{\partial \phi}{\partial y}\Delta y + \mathcal{O}(\Delta y^2)$$

What is a good mesh?

Striving for quality

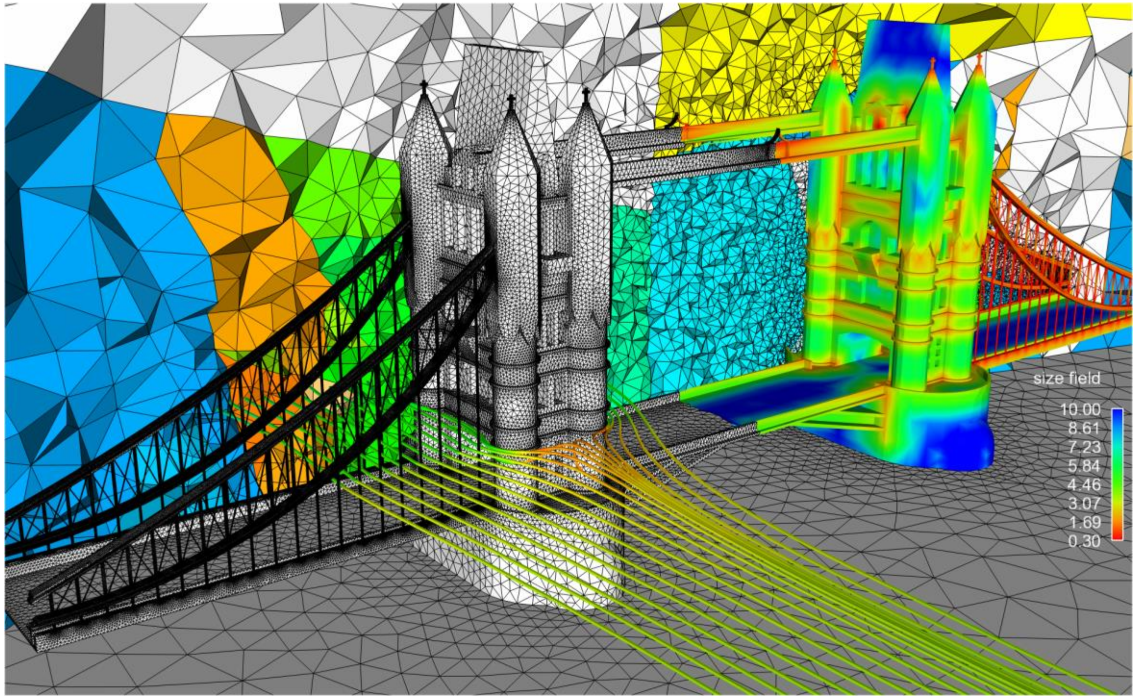
- For the same cell count, hexahedral meshes will give more accurate solutions, especially if the grid lines are aligned with the flow.
- But this does not mean that tetrahedral meshes are not good, by carefully choosing the numerical scheme you can get the same level of accuracy as in hexahedral meshes. The problem with tetrahedral meshes is mainly related to the way gradients are computed.



What is a good mesh?

Striving for quality

- The mesh density should be high enough to capture all relevant flow features. In areas where the solution change slowly, you can use larger elements.
- A good mesh does not rely in the fact that the more cells we use the better the solution.

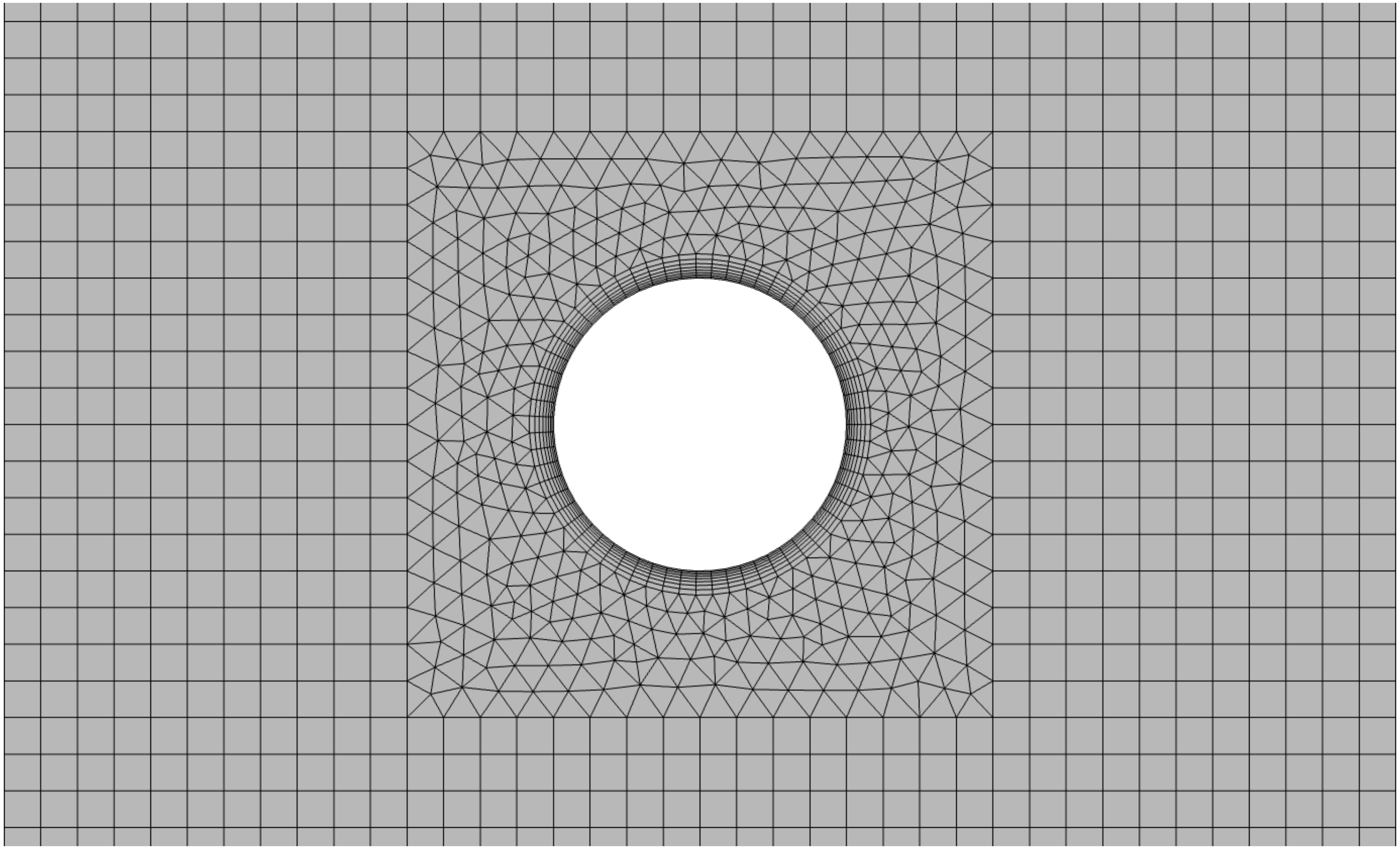


23rd IMR Meshing Maestro Contest Winner
Zhoufang Xiao , Jianjing Zheng, Dawei Zhao, Lijuan Zeng, Jianjun Chen, Yao Zheng
Center for Engineering & Scientific Computation, Zhejiang University, China.
http://www.sandia.gov/imr/MeshingContest.html

What is a good mesh?

Striving for quality

- And by the way, you can combine all cell types to get a hybrid mesh.



What is a good mesh?

Striving for quality

- Hexes, prisms, and quadrilaterals can be easily aligned with the flow.
- They can also be stretched to resolve boundary layers without losing much quality.
- Triangular and tetrahedral meshes can easily be adapted to any kind of geometry. The mesh generation process is almost automatic.
- Triangular and tetrahedral meshes have inherently larger truncation error.
- Tetrahedral meshes normally need more computing resources during the solution stage. But this can be easily offset by the time saved during the mesh generation stage.
- Increasing the cells count will likely improve the solution accuracy, but at the cost of a higher computational cost. But attention, a finer mesh does not mean a better mesh.
- To keep cell count low, use non-uniform meshes to cluster cells only where they are needed. Use local refinements and solution adaption to further refine only on selected areas.
- In boundary layers, quads, hexes, and prisms/wedges cells are preferred over triangles, tetrahedrons, or pyramids.
- If you are not using wall functions (turbulence modeling), the mesh next to the walls should be fine enough to resolve the boundary layer flow. Have in mind that this will rocket the cell count and increase the computing time.

What is a good mesh?

Striving for quality

- Use hexahedral meshes whenever is possible, specially if high accuracy in predicting forces is your goal (drag prediction) or for turbo machinery applications.
- For complex flows without dominant flow direction, quad and hex meshes loose their advantages.
- Keep orthogonality, skewness, and aspect ratio to a minimum.
- Change in cell size should be smooth.
- Always check the mesh quality. Remember, one single cell can cause divergence or give you inaccurate results.
- Plan your meshing approach.
- When you strive for quality, you avoid the GIGO syndrome (garbage in, garbage out).
- Just to end for good the mesh quality talk:
  - A good mesh is a mesh that serves your project objectives.
  - So, as long as your results are physically realistic, reliable and accurate; your mesh is good.
  - Know your physics and generate a mesh able to resolve the physics involve, without over-doing.

What is a good mesh?

A good mesh might not lead to the ideal solution, but a bad mesh will always lead to a bad solution.

P. Baker – Pointwise

Who owns the mesh, owns the solution.

H. Jasak – Wikki Ltd.

Avoid the GIGO syndrome (Garbage In – Garbage Out).
As I am a really positive guy I prefer to say,
good mesh – good results.

J. G.

Mesh quality assessment in OpenFOAM®

Mesh quality metrics in OpenFOAM®

- In the file *primitiveMeshCheck.C* located in the directory **$WM_PROJECT_DIR/src/OpenFOAM/meshes/primitiveMesh/primitiveMeshCheck/** you will find the quality metrics used in OpenFOAM®. Their maximum (or minimum) values are defined as follows:

```
Foam::scalar Foam::primitiveMesh::closedThreshold_  = 1.0e-6;
Foam::scalar Foam::primitiveMesh::aspectThreshold_  = 1000;
Foam::scalar Foam::primitiveMesh::nonOrthThreshold_ = 70;    // deg
Foam::scalar Foam::primitiveMesh::skewThreshold_    = 4;
Foam::scalar Foam::primitiveMesh::planarCosAngle_   = 1.0e-6;
```

- You will be able to run with mesh quality errors such as high skewness, high aspect ratio, and high non-orthogonality. But remember, they will affect the solution accuracy, might give you strange results, and eventually can made the solver blow-up.

Checking the mesh quality in OpenFOAM®

- To check the mesh quality and validity, OpenFOAM® comes with the utility `checkMesh`.
- To use this utility, just type in the terminal `checkMesh`, and read the screen output.
- `checkMesh` will look for/check for:
    - Mesh stats and overall number of cells of each type.
    - Check topology (boundary conditions definitions).
    - Check geometry and mesh quality (bounding box, cell volumes, skewness, orthogonality, aspect ratio, and so on).
- If for any reason `checkMesh` finds errors, it will give you a message and it will tell you what check failed.
- It will also write a set with the faulty cells, faces, and/or points.
- These sets are saved in the directory **`constant/polyMesh/sets/`**

Mesh quality assessment in OpenFOAM®

Checking the mesh quality in OpenFOAM®

- Mesh topology and patch topology errors must be repaired.
- You will be able to run with mesh quality errors such as skewness, aspect ratio, minimum face area, and non-orthogonality.
- But remember, they will severely tamper the solution accuracy, might give you strange results, and eventually can made the solver blow-up.
- Unfortunately, `checkMesh` does not repair these errors.
- You will need to check the geometry for possible errors and generate a new mesh.
- You can visualize the failed sets directly in `paraFoam` or you can use the utility `foamToVTK`.
- The utility `foamToVTK` converts the failed sets to VTK format.

Mesh quality assessment in OpenFOAM®

Visualizing the failed sets in OpenFOAM®

- To convert the failed faces/cells/points to VTK format, you can proceed as follows:

  - `$> foamToVTK -set_type name_of_sets`

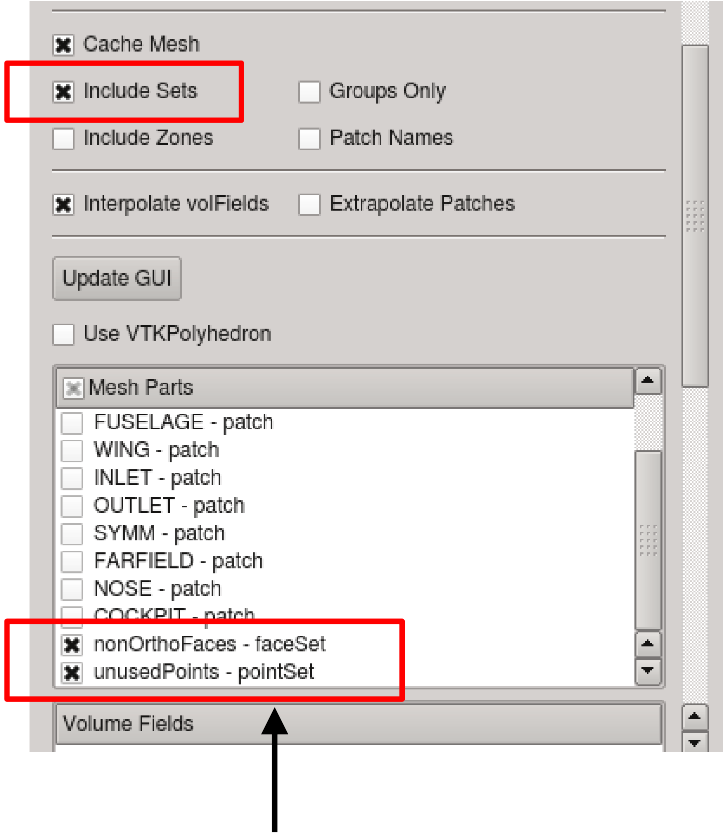
  where set_type is the type of sets (faceSet, cellSet, pointSet, surfaceFields) and name_of_sets is the name of the set in the directory **`constant/polyMesh/sets`** (highAspectRatioCells, nonOrthoFaces, wrongOrientedFaces, skewFaces, unusedPoints, and so on)

- At the end, `foamToVTK` will create a directory named **`VTK`**, where you will find the failed faces/cells/points in VTK format. At this point you can use `paraview/paraFoam` to open the VTK files and visualize the failed sets.

Mesh quality assessment in OpenFOAM®

Visualizing the failed sets in OpenFOAM®

- Alternatively, you can load the failed sets directly within `paraFoam`.
- Remember, you will need to create the sets. To do so just run the `checkMesh` utility.
- If there are problems in the mesh `checkMesh` will automatically save the sets in the directory **`constant/polyMesh/sets`**
- In `paraFoam`, simply select the option **Include Sets** and then select the sets you want to visualize.
- Just to be clear, this method only works with `paraFoam`.
- Most of the times or when visualizing large meshes we prefer to use the option `foamToVTK`.



☒ Cache Mesh

☒ Include Sets □ Groups Only

□ Include Zones □ Patch Names

☒ Interpolate volFields □ Extrapolate Patches

□ Use VTKPolyhedron

☒ Mesh Parts

□ FUSELAGE - patch

□ WING - patch

□ INLET - patch

□ OUTLET - patch

□ SYMM - patch

□ FARFIELD - patch

□ NOSE - patch

□ COCKPIT - patch

☒ nonOrthoFaces - faceSet

☒ unusedPoints - pointSet

Failed sets