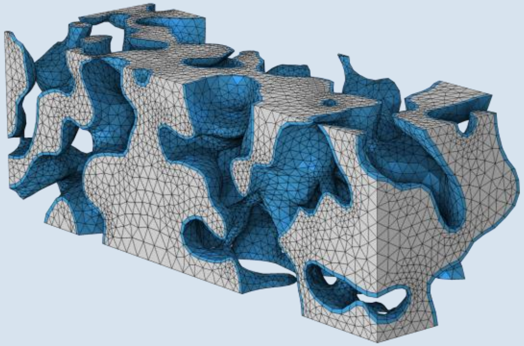
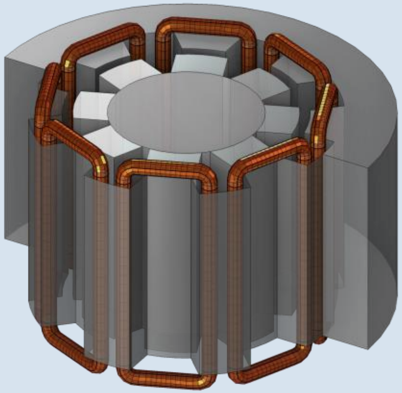
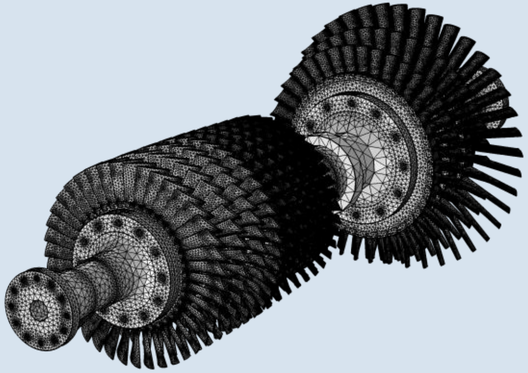
COMSOL Day
Fluids & Heat

Meshing CFD Problems

Nancy Bannach
Development Manager
COMSOL







Why Do We Need a Mesh?

FEM Foundation

FEM is based on discretization of the geometry into small units called *mesh elements*.

It Serves Two Purposes

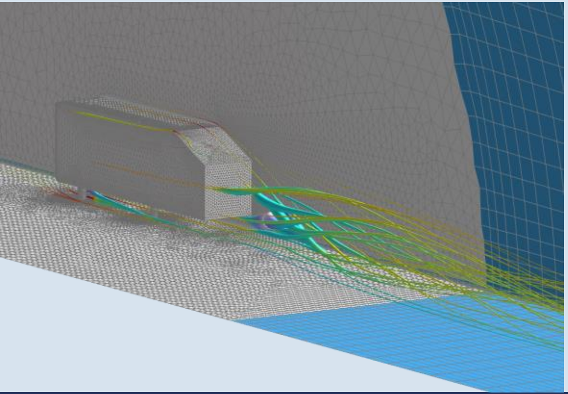
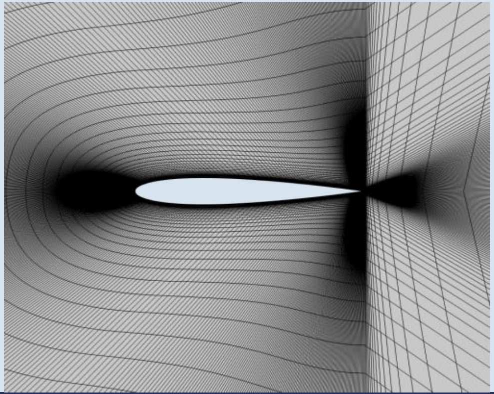
It represents the solution field and it represents the geometry.

Numerical Stability

The mesh should result in a well-conditioned stiffness matrix. It largely depends on size and shape of the mesh elements (mesh quality is a good indication).

Refined Mesh Leads To...

A more accurate approximation and solution are achieved with a refined mesh. It requires more memory and time to refine the mesh.

Why Meshing in CFD is so important?

Flow Characteristics

Important flow features like vortices, turbulences, and shocks require local mesh refinement.

Boundary layers

Fine mesh near (no slip) walls needed to resolve steep gradients.

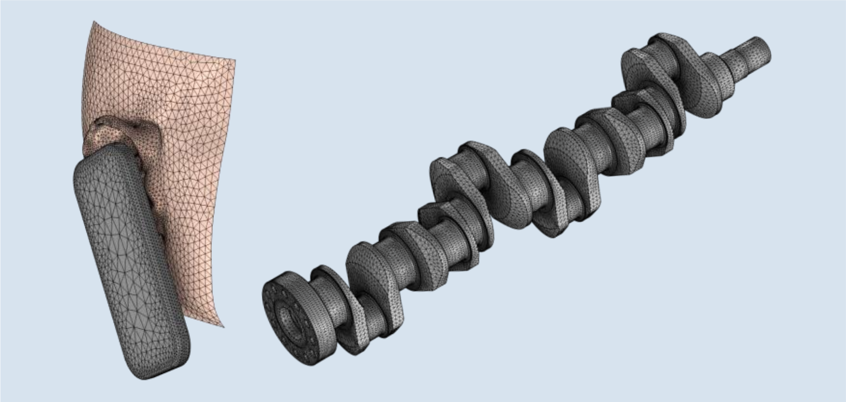
Accuracy

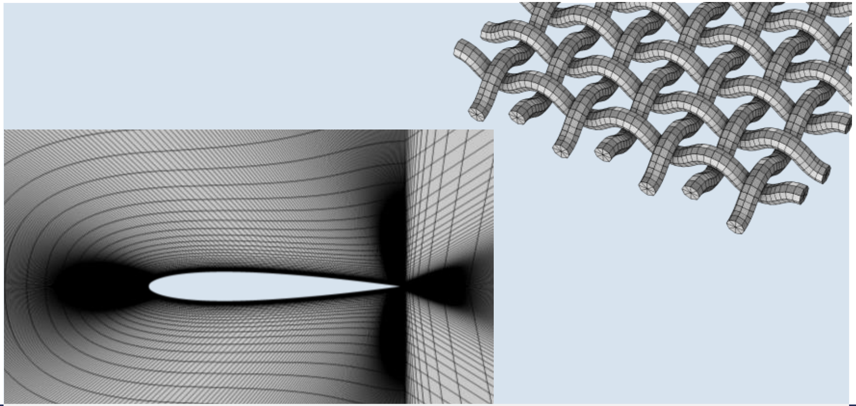
Coupled equations (e.g. transport equations) depend on accurate flow data. A poor mesh can distort how this information is passed between cells, reducing solution quality.

Stability

A proper mesh helps to improve stability and convergence of CFD problems where pressure and velocity are strongly interdependent.

Structured vs. Unstructured Meshes





Unstructured Mesh

- Triangular, quadrilateral, or tetrahedral

+ Most general mesh: No constraints on the geometry

+ Robust

- Might not be the most efficient mesh

Structured Mesh

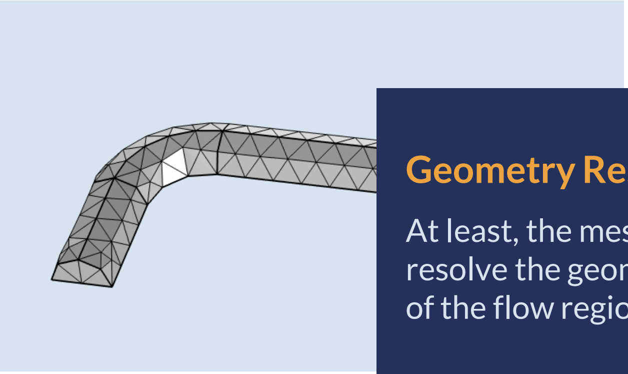
- Mapped (2D) or Swept Mesh

+ Low number of mesh elements.

+ Efficient to generate

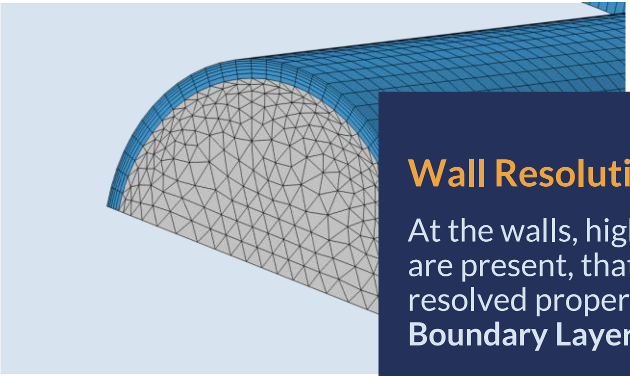
- Restrictions on Geometry

Accuracy



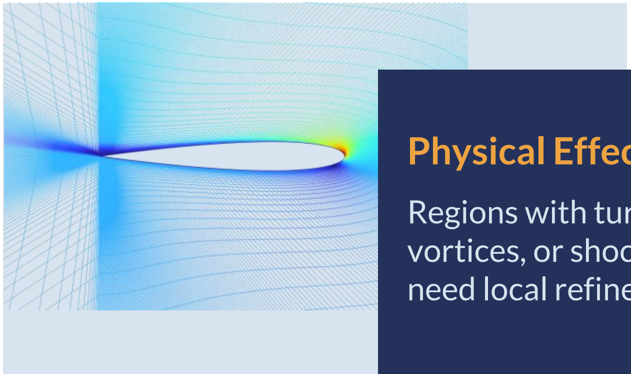
Geometry Resolution

At least, the mesh should resolve the geometric shape of the flow region



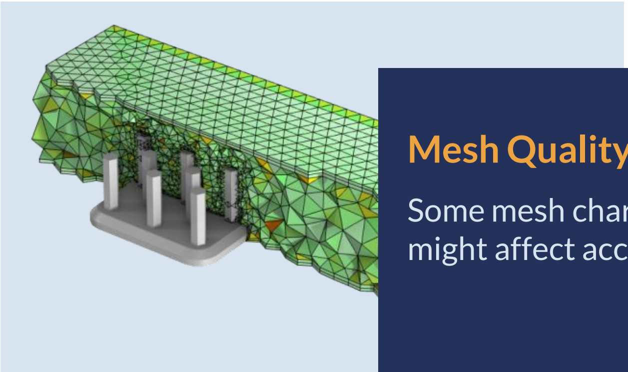
Wall Resolution

At the walls, high gradients are present, that need to be resolved properly using **Boundary Layer Mesh**



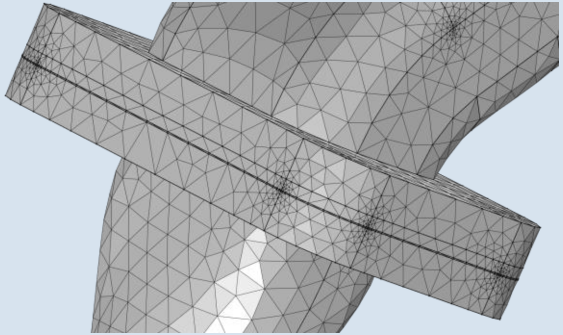
Physical Effects

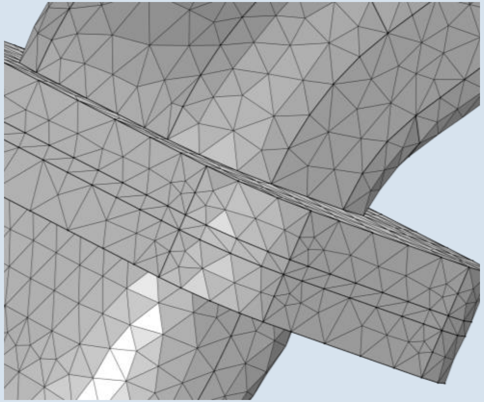
Regions with turbulence, vortices, or shock waves need local refinement.

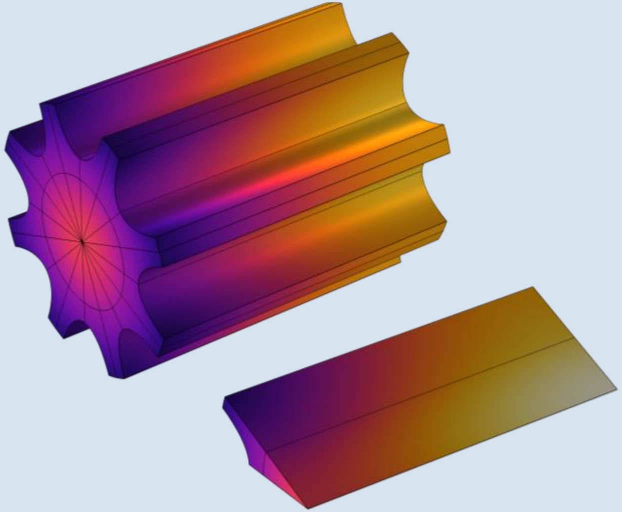


Mesh Quality

Some mesh characteristics might affect accuracy.





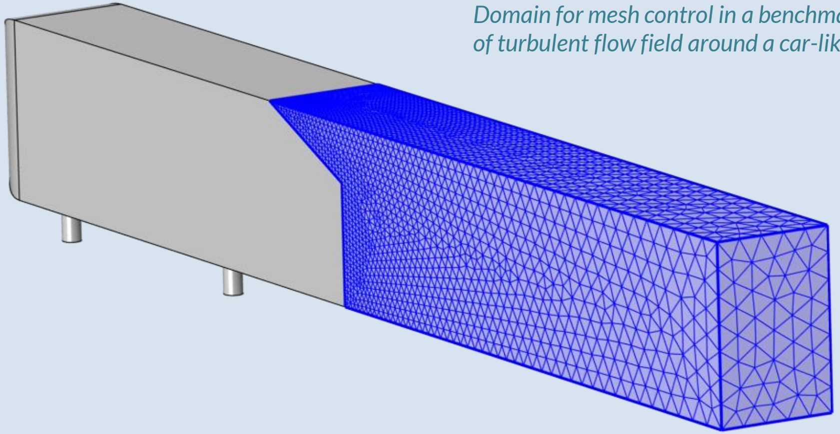


Geometry Considerations

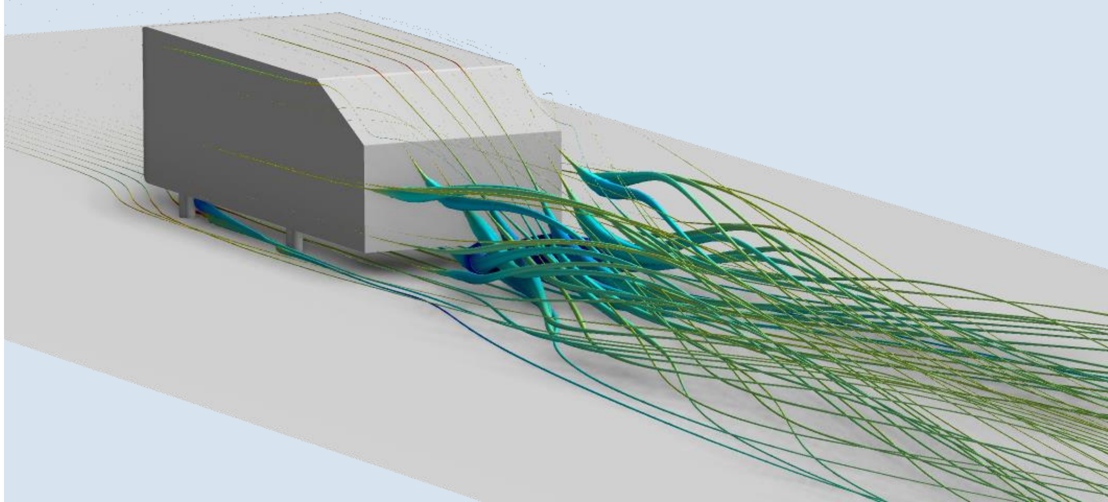
- Make use of symmetries!
- Make use of symmetries!
- Remove unimportant details from the geometry!
- Introduce additional domains/boundaries only for the purpose of building a good mesh

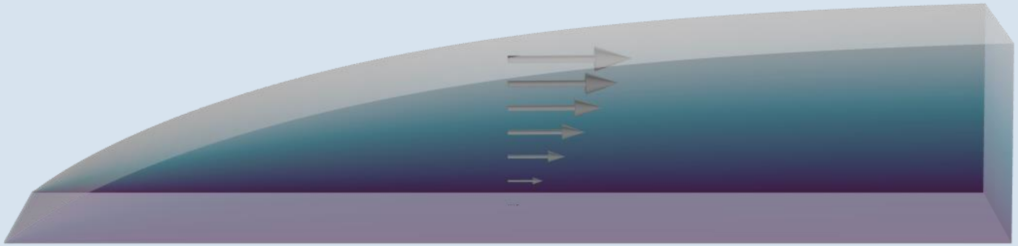
Mesh Control Operations

- Gain precise control over mesh layout and density in regions of known rapid changes or steep gradients.
  - Common in CFD simulations.
- The selected geometric entities:
  - Available exclusively to control local mesh properties .
  - Do not affect the subdivision of edges, faces, or domains when applying physics settings.
- The used geometric entities are removed from the geometry after the mesh is created.



Domain for mesh control in a benchmark model of turbulent flow field around a car-like object.





Laminar Flow

Characteristics

Relatively thick boundary layers where velocity changes gradually from wall to bulk.

Sensitive to disturbances.

Estimation

Use asymptotic expressions[1] to estimate the width of boundary layers, wakes and mixing layers

Free Flow Domains

Boundary layer over flat plate:

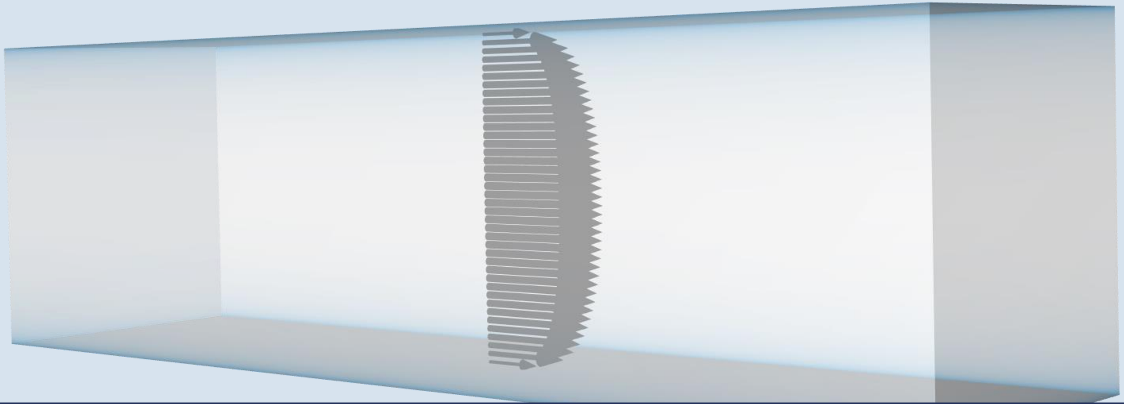
- Reynolds number: $Re_x = \frac{Ux}{\nu}$
- Thickness: $\delta(x) \approx \frac{5x}{\sqrt{Re_x}}$

Porous Domains

For Brinkman:

$$\delta \approx 5\sqrt{\frac{\kappa}{\varepsilon}} \text{ for } \frac{U}{\nu}\sqrt{\frac{\kappa}{\varepsilon}} \ll 1$$

[1]Schlichting H, "*Boundary-Layer Theory*", McGraw-Hill, 1987



Turbulent Flow

Characteristics

Thin boundary layer with much steeper velocity gradients and enhanced mixing.

Relatively stable to disturbances.

Estimation

Use asymptotic expressions[1] to estimate the width of boundary layers, wakes and mixing layers.

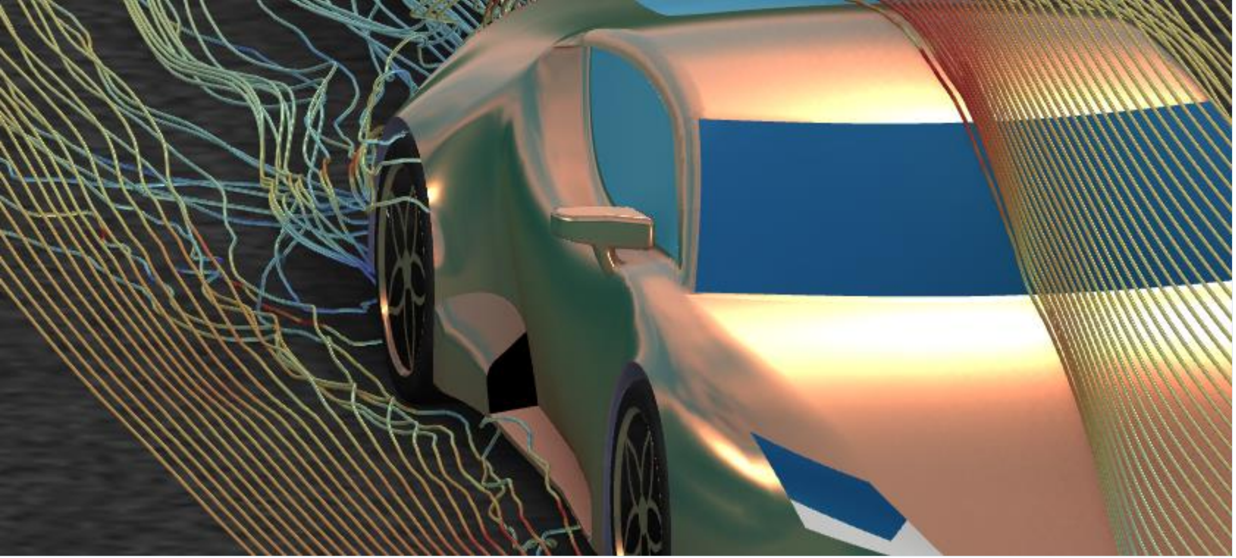
Free Flow Domains

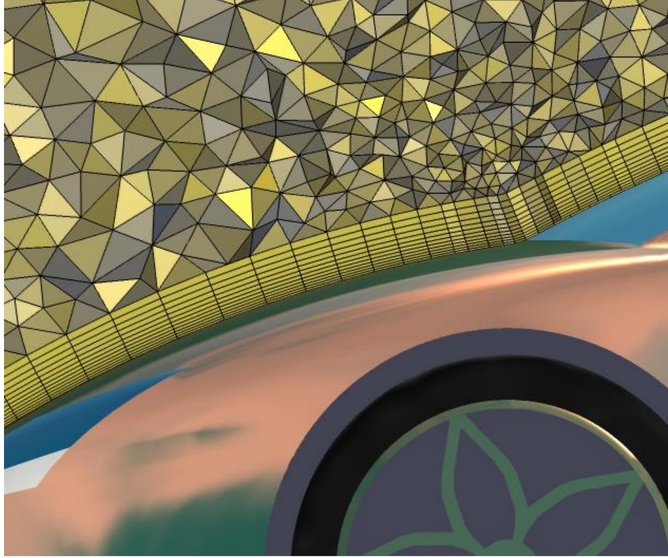
Boundary layer over flat plate:

- Reynolds number: $Re_\delta = \frac{U\delta}{\nu}$
- Thickness: $\delta(x) \approx \frac{0.37x}{Re_x^{1/5}}$
- Viscous units: $y^+ = \frac{u_\tau y}{\nu},\ u_\tau = \sqrt{\frac{\tau_w}{\rho}}$

Porous Media Domains

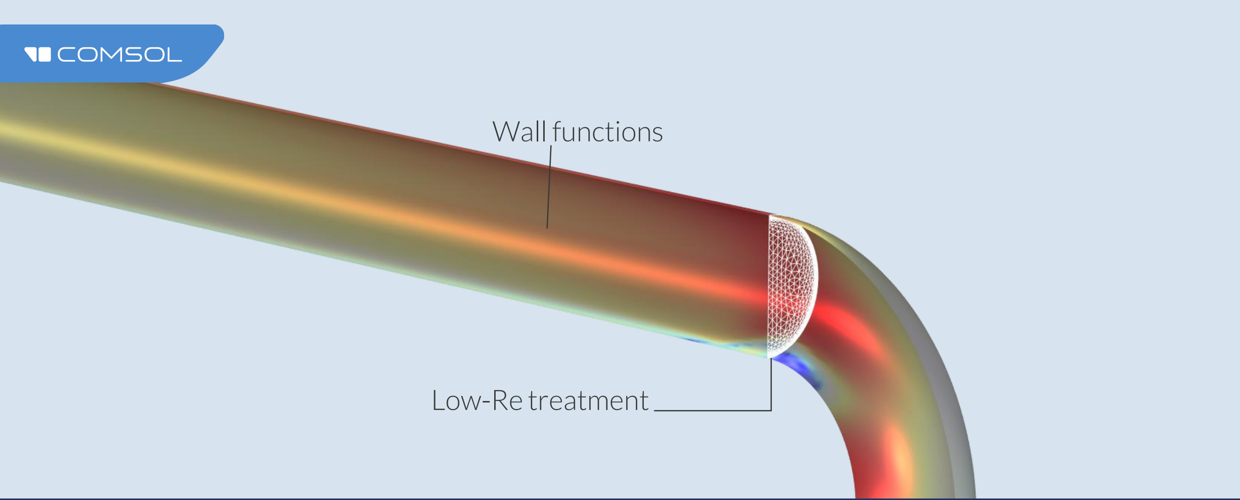
As a first approximation: replace U by U/ε in the expressions to the left.

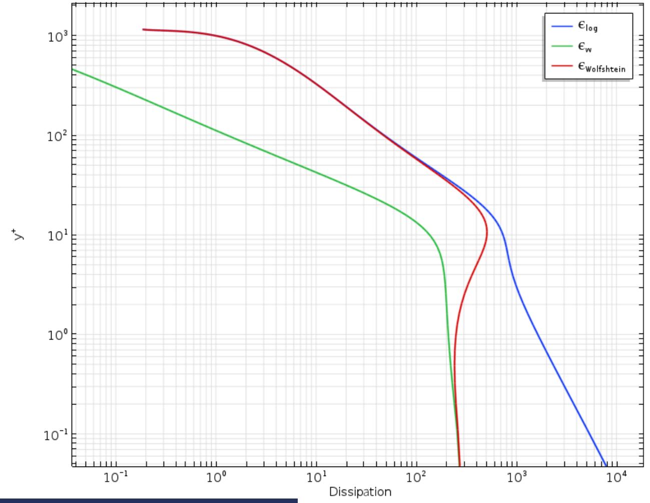




Generating Boundary Layers Meshes

- Consists of layered quadrilateral elements in 2D or layered prism or hexahedron elements in 3D.
- Automatic detection and treatment of sharp corners.
- Manual control of boundary layer properties.
- Can be added for any domain mesh or by sweeping a face mesh with boundary layers.
- Smooth transition to the interior mesh.
- Support for boundary layer mesh on isolated boundaries.
- Application Library Tutorials: [Boundary Layer Meshing Tutorial Series]





Wall Treatment Functionality

Wall Functions

- Applies a wall stress based on the log-law
- Resolution: $25 < \delta_w^+, \ \delta_w < 0.3\delta$, where δ_w is the wall lift-off
- Not available for LES/DES

Low-Re-Number Treatment

- Viscous sublayer is resolved
- $\boldsymbol{u} = \boldsymbol{u}_{BND}$
- Resolution: $d_w^+ \leq 1$, where $d_w = h_\perp$

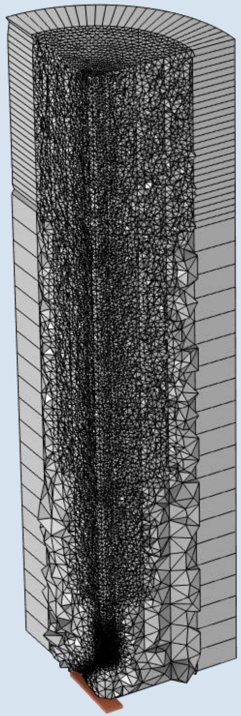
Automatic Wall Treatment

Wall stress based on a blending between the log-law and the viscous sublayer

Resolution:

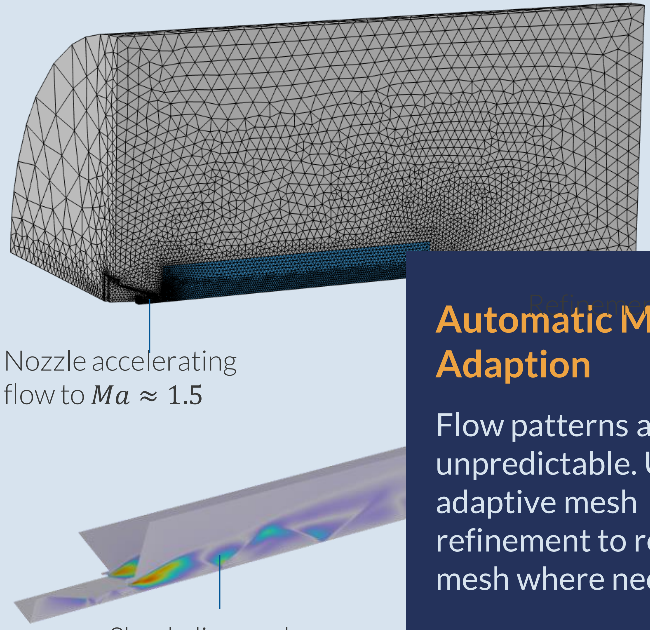
- RANS: $\delta_w < 0.3\delta$ LES: $d_w^+ \leq 20$

Physical Effects



Manual Mesh Modification

In certain problems it is known where e.g. vortexes appear, e.g. flow past an obstacle. Then adjust the mesh manually.



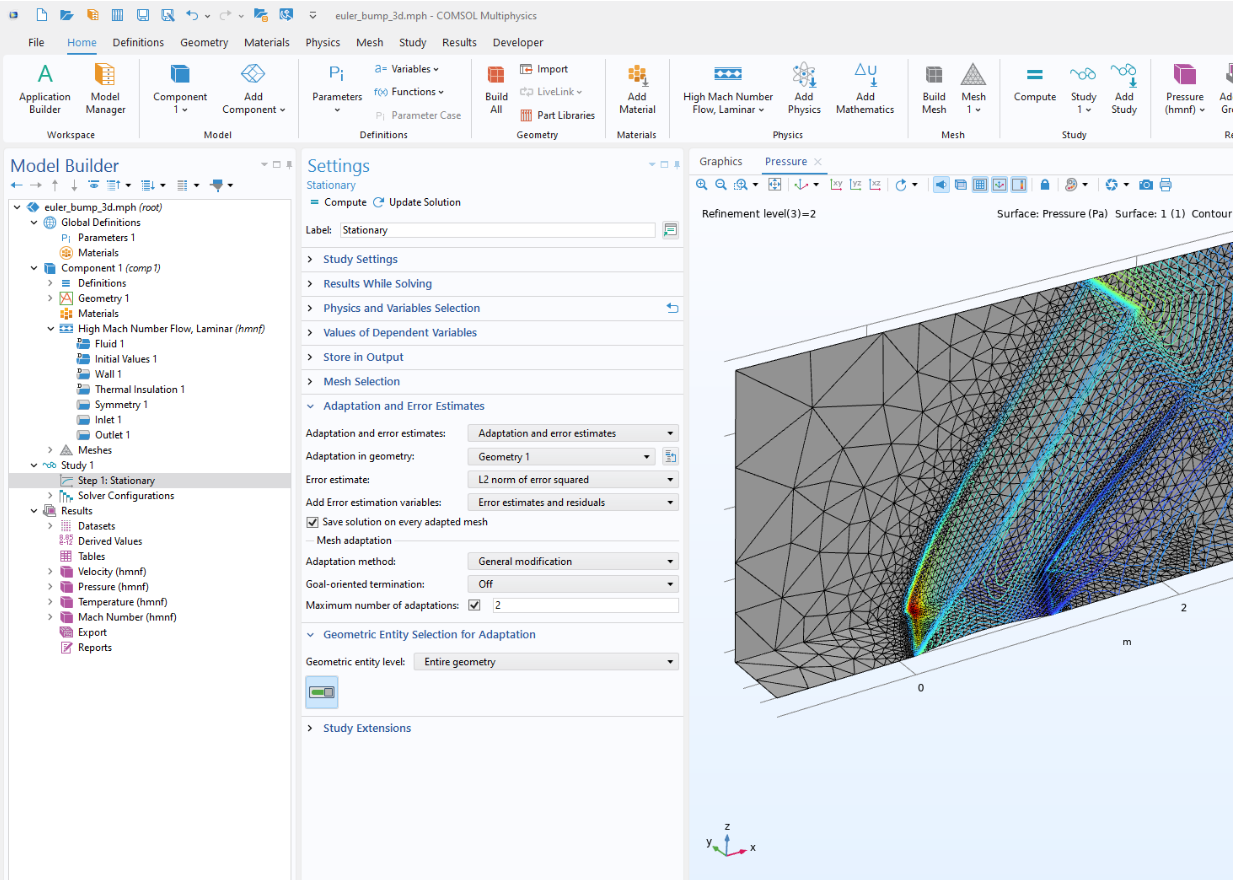
Automatic Mesh Adaption

Flow patterns are often unpredictable. Use adaptive mesh refinement to refine mesh where needed.

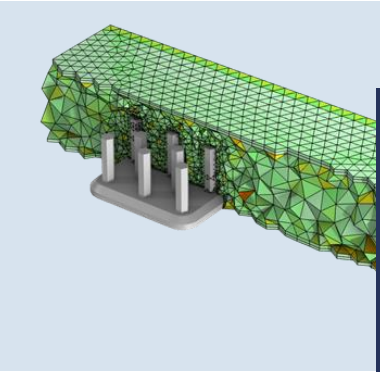
COMSOL

Automatic Mesh Adaption

- Based on error, functional or user defined expression

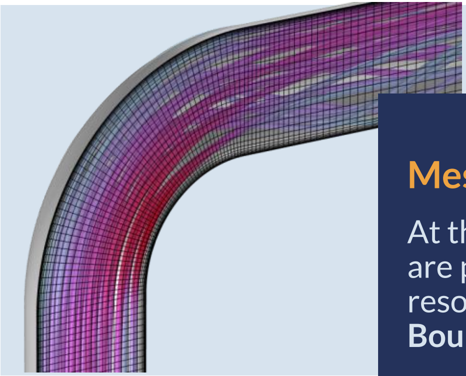


Stability



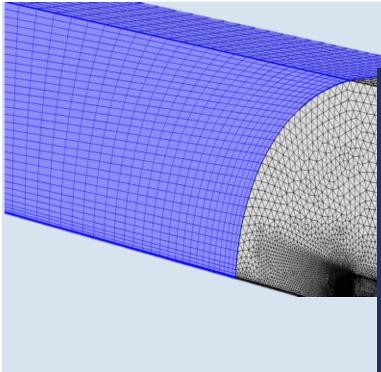
Mesh Quality

Low mesh quality leads to numerical instability.



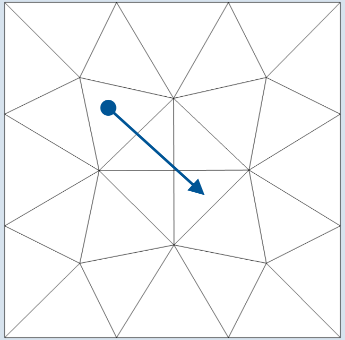
Mesh Alignment

At the walls, high gradients are present, that need to be resolved properly using **Boundary Layer Mesh**



Mesh Transitions

Avoid large differences in mesh element sizes next to each other.

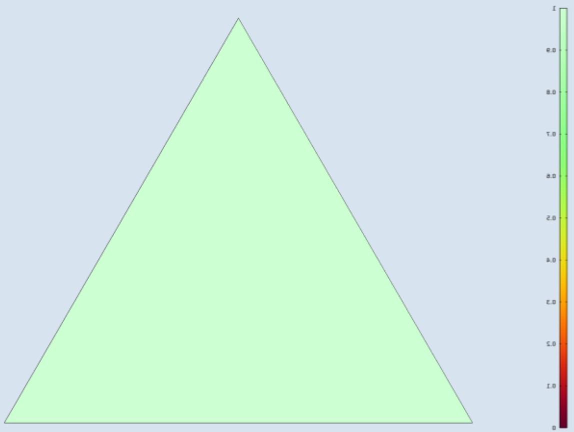


CFL Condition

For time dependent models CFL criteria needs to be fulfilled

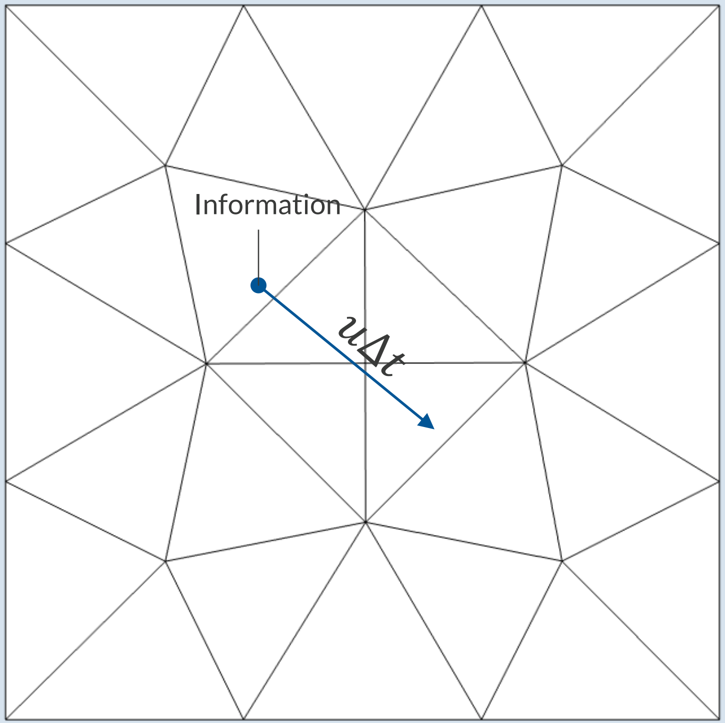
Mesh Quality

- Skewness (deviation from ideal element angles):
  - Distorts gradient calculations, which reduces accuracy and can make the simulation unstable.
  - The resulting matrix system has a higher condition number, meaning it's harder to solve accurately and quickly. Small errors get amplified.
- Growth rate
  - Solver tries to estimate what happens from one cell to another
  - Gradients might be overlooked
  - Solutions can be smoothed out
- Avoid low quality elements in critical regions (walls, flow separation)



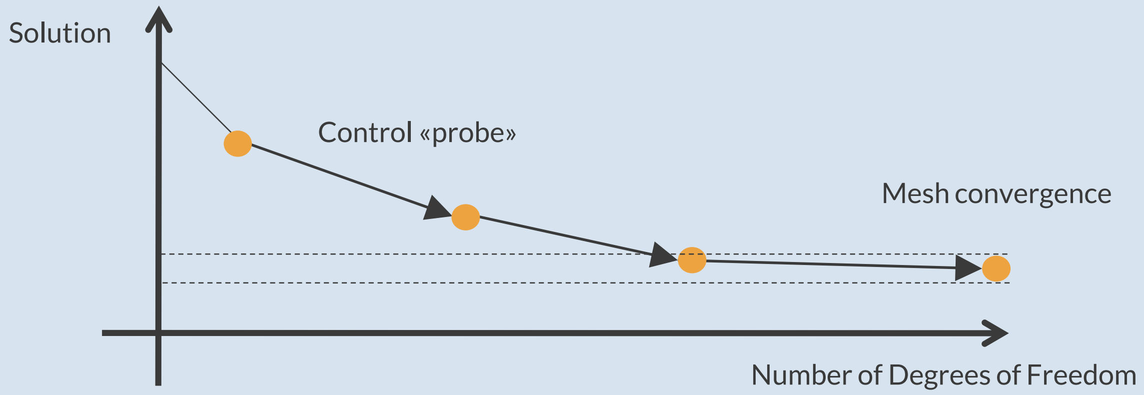
Time-Dependent Models

- CFL Condition:
  - Information (fluid) must not travel more than one cell per time step
    $$CFL = \frac{u\Delta t}{\Delta x}$$
  - CFL<1 (explicit solver)
  - Sometimes manual limit is needed: Use fraction of built-in variable spf.dt_CFL



Information must not travel more than one cell per time step.

How Many Elements Do We Need?



Rarely Known at the Beginning

Fine enough to represent the geometry adequately

Fine enough to resolve all gradients of the solution

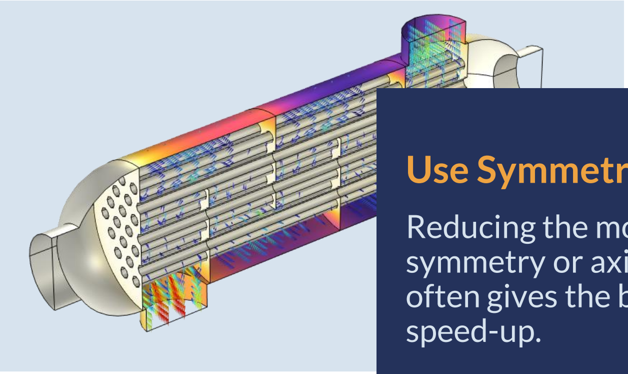
Mesh Refinement Study

Refine gradually the mesh until the results are not affected

Gain confidence in the accuracy of your model

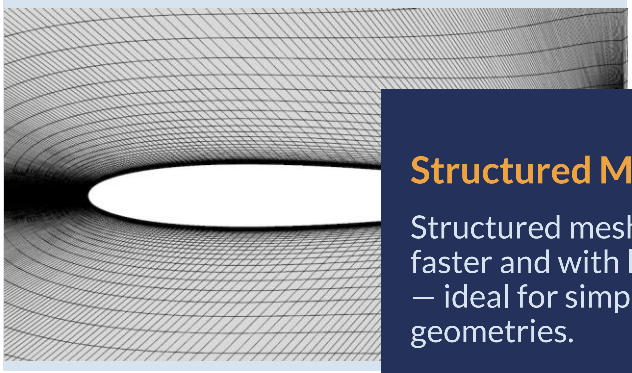
Knowledge Base: [Performing a Mesh Refinement Study]

Performance



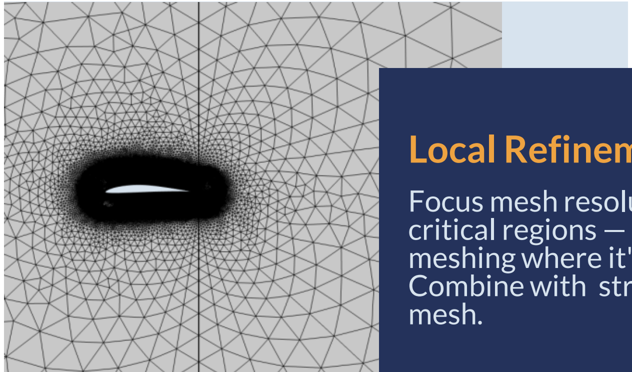
Use Symmetries

Reducing the model using symmetry or axisymmetry — often gives the biggest speed-up.



Structured Mesh

Structured meshes solve faster and with less memory — ideal for simple geometries.



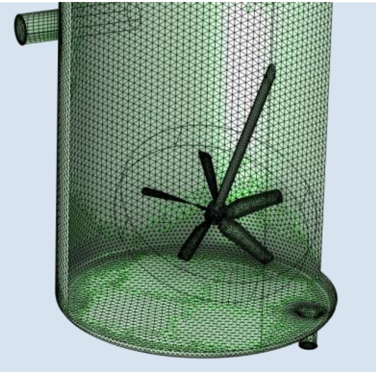
Local Refinement

Focus mesh resolution in critical regions — avoid over-meshing where it's not needed. Combine with structured mesh.

Adaptive Meshing

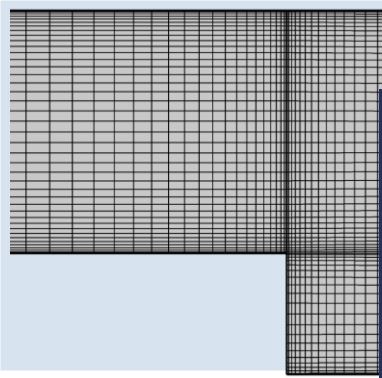
Refine and coarsen mesh automatically during computation based on error.

Performance



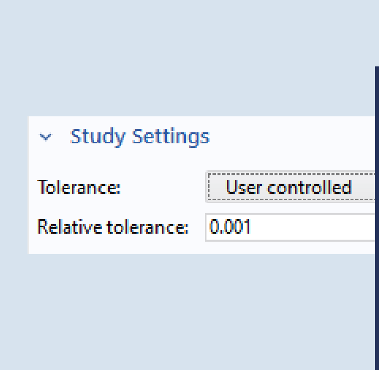
Improve Element Quality

Spend time on preparing the mesh. Good element quality means faster convergence.



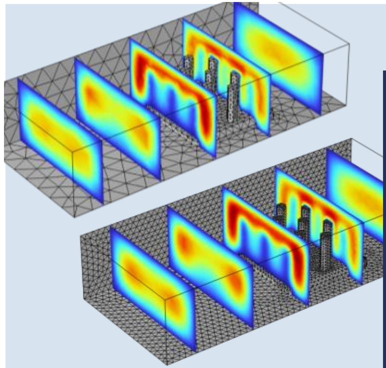
CFL Number

If the mesh is unnecessarily too fine (locally) also the time step must be reduced unnecessarily.



Solver Tolerance

A coarse mesh and tight solver tolerance does not mean a more accurate solution.



Initial Solution

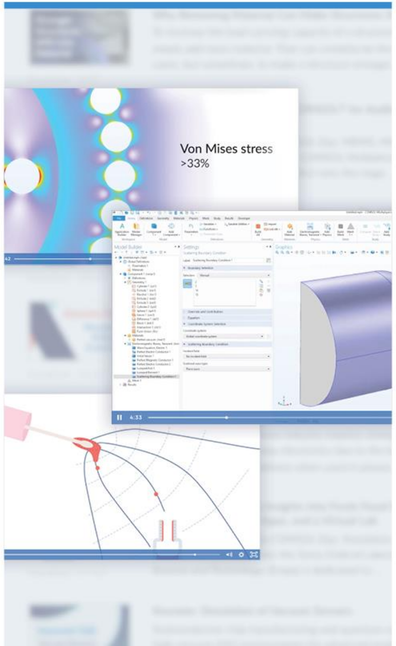
Compute on a coarse mesh with simplified conditions and use this as starting point.

Further Resources for Inspiration

comsol.com

BLOG POSTS

VIDEOS



USER STORIES

MODELS & APPLICATIONS