Intelligent Automatic Meshing

The meshing technology in CFdesign readily adapts complex 3D assemblies for simulation. This workhorse functionality automatically does the work most engineers want to avoid. By automating this process CFdesign can be used by more people to get more done on every project.

These are the key components of Automated Meshing:

1. Automatic Mesh Sizing
2. Local Size Adjustment
3. Geometry Mesh Diagnostics
4. Boundary Layer Mesh Enhancement
5. Interactive Mesh Refinement Regions
6. Extrusion
7. Volume mesh growth rate specification
8. Surface-based mesh distribution and refinement
9. Gap and Thin Solid Refinement
10. Mesh generation flexibility

1. Automatic Mesh Sizing

The heart of CFdesign Meshing, Automatic Mesh Sizing defines the mesh by performing a comprehensive topological interrogation of the analysis geometry and determining the mesh size and distribution on every edge, surface, and volume. Automatic Mesh Sizing considers geometric curvature, gradients, and proximity to neighboring geometry when computing all element sizes of the mesh distributions.

Benefits:

- Greatly simplified set-up of analysis models resulting in less time spent assigning mesh sizes.
- Efficient use of computing resources--the mesh is fine where required and coarser where appropriate.

- Improved solution accuracy due to better mesh quality and mesh transitions.
- Improved solution robustness--good mesh transitions lead to a well-posed mathematical model.

2. Local Size Adjustment

Automatic Mesh sizing creates a mesh distribution based on multiple geometric factors, and Local Size Adjustment provides a graphical way to dynamically customize the mesh as needed.

Benefits:

- Customize the mesh based on the physics and material models in areas of the model that do not have a lot of geometric curvature.
- Maintain smooth transitions across the entire mesh and eliminate discontinuities between modified and existing settings using the Spread Changes function.
- In some cases, an entity should have a uniform mesh. The “Use Uniform” command modifies the length scales on an entity to be the same, thus ensuring the entity’s mesh does not vary.

3. Geometry Mesh Diagnostics

The Diagnostics functions identify extremely thin surfaces and extremely small edges, relative to the rest of the model. In many cases, these are caused by poor CAD practices, a lack of design intent, or multiple file format conversions.

The Minimum Refinement Length is an important tool that allows fine-tuning of Automatic Mesh Sizing to either include or exclude small edges from the model, based on their relevance to the simulation.

Benefits:

- Identify and fix geometric problems including disproportionately small edges and problematic surfaces (such as cusps, slivers, and thin annuli) before meshing.
- Customize the amount of influence that small edges have on the mesh, without having to modify the CAD model.

4. Boundary Layer Mesh Enhancement

Mesh Enhancement automatically adds layers of elements to the fluid mesh along all fluid-wall and fluid-solid interfaces. Mesh heights vary gradually across adjacent surfaces to ensure even transitions.

Benefits:

- The element layers generated by Mesh Enhancement produce a smooth distribution along all walls, which is critical for accurate flow and temperature prediction and results visualization.
- Mesh Enhancement ensures an adequate mesh within tight gaps, which is essential for capturing detail in and around small features.

Automatic Layer Adaptation

- An extension of Mesh Enhancement that automatically varies the distance between the nearest fluid nodes and the wall to ensure the Y+ value is maintained within the optimal range for turbulent flow.
- A proper Y+ value is essential for accurately calculating shear in external flows such as vehicle aerodynamics.

5. Mesh Refinement Regions

Local Size Adjustment is extremely useful for modifying the mesh distribution on geometric entities. If, however, energetic flow is anticipated within a region that does not contain selectable geometric entities, use a Mesh Refinement Region to focus the mesh.

Mesh Refinement Regions are available in several shapes, and are navigated interactively to ensure exact coverage of the refined mesh. The mesh distribution can either be uniform or spatially-varying, based on the length scales of surrounding geometry (for uniformity). They are features of the simulation model, and are not part of the CAD geometry.

Benefits:

- Improve solution accuracy and efficiency by refining the mesh only in critical areas.

- Avoid adding superfluous parts to the CAD model that are only used for mesh refinement. Such volumes typically have no physical meaning, and can interfere with PLM initiatives.

6. Extrusion Meshing

Extrusion Meshing creates layers of prism elements through the length of uniform cross-section parts. Automatic Extrusion Sizing determines the growth to ensure a smooth transition between the surface mesh and extrusion layers. Based on a user-specified Growth parameter, CFdesign automatically computes the number of layers and the amount of transitional growth from one layer to the next.

A manual mode is also available which leaves the control of layer growth and the number of layers to the user.

Benefits:

- Automatic Extrusion Sizing simplifies the setup process by ensuring both continuity with the surrounding mesh and efficient growth of extrusion mesh layers.
- Greatly reduces the element count in high aspect-ratio parts, resulting in reduced simulation times.
- Improves flow accuracy in models dominated by form drag, such as pipe flow.

7. Volume Growth Rate

Part of the next generation of Automatic Mesh Sizing, use the Volume Growth Rate to control how much (or little) the mesh can grow in large, sparsely detailed regions. Refinement Regions are great for focusing the mesh in a particular area, but specify a Volume Mesh Growth Rate for additional control of the volume mesh throughout the entire model.

Benefits:

- Managing element growth in open areas improves the accuracy of the flow and temperature results.
- An intuitive, percentage-based growth parameter makes it easy to specify how much the mesh will grow, ensuring a predictable model size.

8. Automatic Surface Refinement

Part of the next generation of Automatic Mesh Sizing, Surface Refinement provides direct control over isotropic length scale variations across model surfaces. It supports explicit control over transition rates, and allows specification of the growth rate for surface meshes.

Benefits:

- Produce a better mesh on surfaces that have little curvature or few edges.
- Allow the Boundary Mesh Enhancement layer thickness to vary throughout the entire mesh.
- Enables Gap and Thin Solid Refinement…

9. Gap and Thin Solid Refinement

Gaps and thin solids are a fact of life in many mechanical devices, but they can make creating a mesh more complex than it should be. This tool tackles the problem by focusing specifically on small gaps (even if they belong to different parts) as well as long, skinny solid parts.

In many devices, flow through the small gaps is the key to their operation, and well meshed gaps are essential. Simply reduce the “Gap refinement length” to be smaller than these gaps, and specify how many elements you want within the gap. The result is a gap with a mesh fine enough to resolve the flow through it.

Conversely, some gaps just aren’t that important, and removing them from the CAD model can be challenging and disruptive. Instead, simply set the “Gap refinement length” to be larger than the gap size, and prescribe a single element in the gap. The result is a gap meshed with a single element layer down its length. It is part of the model, but is effectively “ignored” by the flow.

Benefits:

- Control the mesh in gaps, whether they are critical or not.
- Improve the accuracy in small clearances and the heat transfer in thin solids.
- Improve solution performance by focusing mesh only in essential gaps, and ignoring the rest.

10. Mesh generation flexibility

The model-centric workflow provides the flexibility to create the mesh when you want to, on your own terms.

In some situations, it's a good idea to generate the mesh, and then check it over before running the analysis. This is a great strategy for complicated systems, especially for the first design iteration. Use the "Generate Mesh" command.

In other situations, hit the "Start" button, and don't look back until the solution is complete. This is ideal for simpler devices and for design iterations where the mesh is "dialed in" and doesn't need a lot of checking. Simply specify the number of iterations, and click the "Start" button. The mesh is generated and the solution starts automatically.

Benefits:

- Mesh the model when you want to, on your terms.
- Inspect the mesh for the first design.
- Automatically create the mesh for each scenario when scheduling analyses with the Solver Manager.