Technical Paper

Introduction to Meshing in Ansys Discovery Software

Alfred Oti

Developed and curated by the Ansys Academic Development Team

education@ansys.com

Ansys Software Used

This case study uses Ansys Discovery™, the 3D product simulation software.

Summary

Ansys Discovery software democratizes the practice of meshing by making it accessible to all users regardless of their prior level of knowledge and skill. Ansys Discovery Software achieves democratization by automating many of the processes associated with meshing. Thereby enabling users to create meshes efficiently, effectively and with greater complexity. In this case study we explain the general concepts of meshing. By the end of this paper, users should have an introductory understanding of the types meshing available in the Ansys Discovery software and the contexts in which they are applied. Informing users of these concepts will enable them to make decisions about their designs more efficiently and effectively and thereby increase confidence in their skills and the tool.

Table of Contents

1. The role of meshing in simulation

In Engineering design, the term “Simulation” describes the processes of using computer models that imitate the physical forms, conditions, behaviors and performances of products, structures, mechanisms and systems. Simulation enables engineers to generate data by which they can analyze, understand, optimize and evaluate the efficacy of their designs without building physical prototypes.

To successfully conduct a simulation the computer models must be split into smaller sections using a grid. Numerical analysis (partial differential equations) can then be performed on the elements and the grid to generate simulation data. The grid is referred to as a Mesh. In this case study we will discuss the techniques used to generate meshes in Ansys Discovery Software.

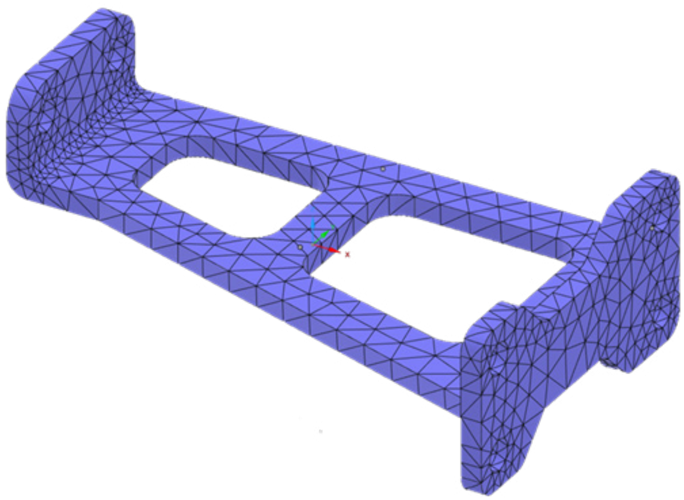

Figure 1: Model of a bracket with a Tetrahedral mesh

2. What is a mesh?

The computer models used in simulations are built using complex 3D geometry that resemble real-world objects. The complexity of the geometry may be increased by asymmetric and dis-continuous surfaces (with gaps or holes) of non-uniform lengths, widths, depths and thickness, angles and curvatures. Even if geometry were symmetrical the difficulty performing calculations on whole objects would not be much reduced. Before computer simulations, engineers performed manual analytical solutions to approximate object behavior. These solutions were limited and time-consuming.

A Mesh divides the entirety of the model’s surface and interior into small sections that enable numerical analysis and analytical solutions to be performed more efficiently and accurately. The greater the number of divisions, the greater the accuracy of the numerical analysis and the closer to the analytic solution.

Visually a mesh resembles a 3D grid, but it is actually a collection of connected points, sections and lines that represent the data in 3D space as a mathematical model. In Ansys Discovery software, meshes are observable in the Explore and Refine modes. In the next section we will discuss the methods of meshing used in each mode in further detail.

3. How does meshing work in Ansys Discovery Tool: Explore Mode?

The Explore mode uses small discrete 3D cubes of uniform size called voxels to represent CAD model geometries. Each voxel contains data regarding its co-ordinates in space, orientation, size and other attributes.

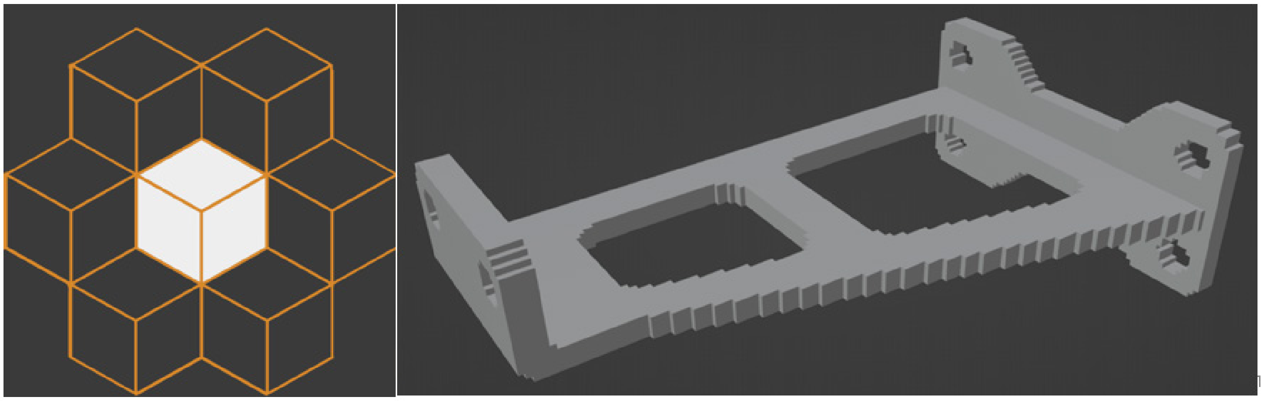

Figure 2: (Left) A single voxel highlighted in a mesh. (Right) A voxel model[1] of a bracket – (the model is exaggerated for the purpose of illustration).

A voxel mesh is an array of voxels that form a 3D object (or volume). The space between the voxels forms a 3D Cartesian grid which is not part of the mesh. Numerical analysis is performed on the voxels not the grid.

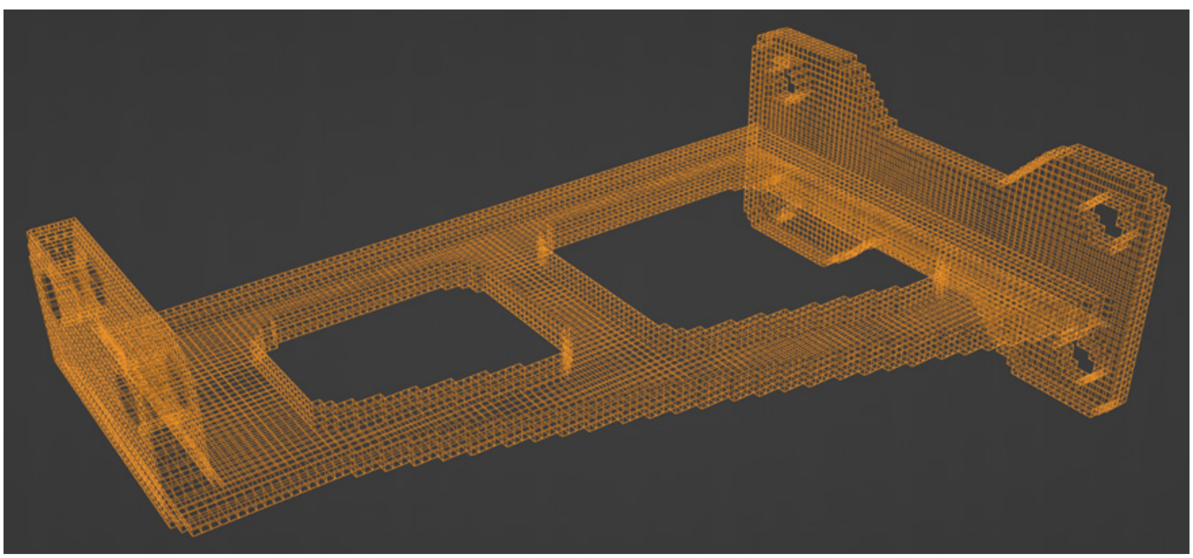

Figure 3: A voxel mesh[2] of a bracket shown in wireframe

The fidelity of the voxel mesh can be increased or decreased by adjusting the fidelity scale found in the state navigator. Manually altering the size of the voxels does not change the appearance of the mesh but rather the density of the voxels within. Increasing the fidelity reduces the size of the voxels and increases the number within the mesh. Decreasing fidelity does the opposite. Alternatively, a set fidelity approach can be selected using the global solution fidelity tool (for more information please see the Ansys Discovery software documentation).

1 Figure 2 (right) image produced in Blender 3.1.2

2 Figure 3 image produced in Blender 3.1.2

The size of any voxel in the mesh can be viewed using the size preview tool. When selected a red cube appears within the mesh the cube can be moved to show the location and size of any voxel including those at the boundary curves and edges of the mesh.

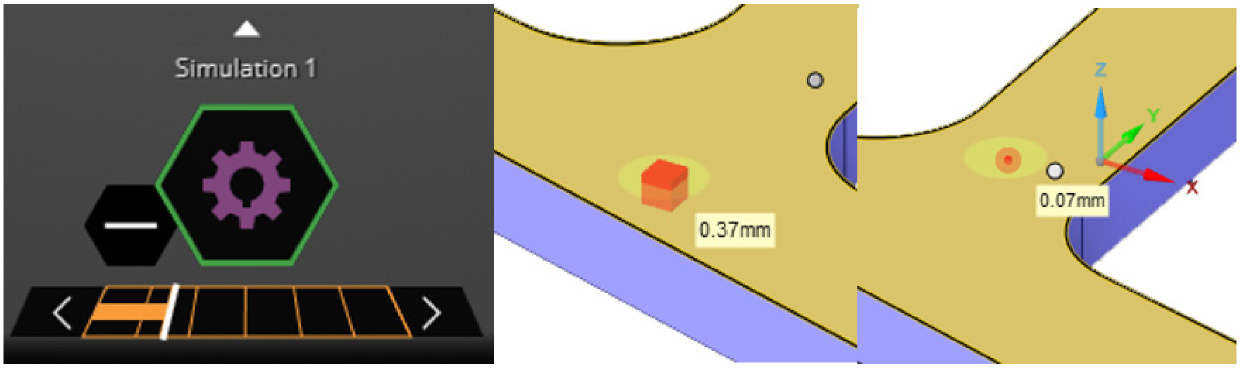

Figure 4: (Left) Fidelity Scale (Center) A voxel at minimum fidelity. (Right) A voxel at maximum fidelity

Adjusting the fidelity of the voxel mesh also affects its resolution, once again the appearance of the voxel mesh is not affected. Here resolution refers to areas of the voxel mesh that are excluded from the numerical analysis. These areas are typically displayed in red and are located at the boundary curves and edges of the mesh. Voxels within the red areas only partially rest within the extents of the mesh.

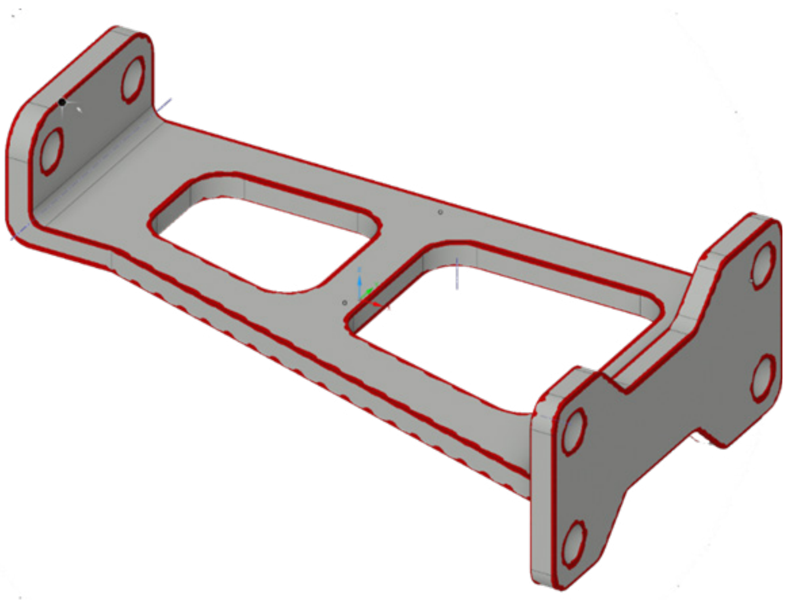

Figure 5: Model of a bracket with low fidelity resolution

4. How does meshing work in Ansys Discovery Tool: Refine Mode?

The complexity of computer (or CAD) models and meshes may be further complicated by the inclusion of small details such as indented logos, threads in holes or on bolts and indented walls etc. Such details often prolong the time needed for numerical analysis without offering greater accuracy. Therefore, engineers simplify their CAD models to improve efficiency, this is known as Defeaturing.

During defeaturing engineers may also find errors in their construction of their CAD models and volume meshes such as unintended gaps or misaligned edges etc. that would cause errors in calculation. The act of fixing errors is known as Model Preparation.

Defeaturing and model preparation are time consuming activities that require skill. To prevent both activities being a barrier to entry Ansys Discovery tool automatically performs defeaturing and model preparation without disrupting the user experience. Mesh generation is also performed automatically once the engineer has specified their desired boundary and loading conditions and the solve icon is clicked.

During the solve process Ansys Discovery software will perform a sweep of the CAD model geometry searching for areas with identical start and end faces and constant cross-sections where a simple uniform volume mesh can be applied. The sweep will apply a non-uniform mesh if such areas are not found. So far, we have discussed meshing in very general terms in the next sections we will continue our discussion in much greater detail highlighting the differences between the types of meshing in the Refine mode of Ansys Discovery tool.

4.1 Hexahedral Meshing in Refine Mode

A hexahedral mesh is 3D network consisting of nodes, elements and edges. Each hexahedral element is arranged in the form of a prism with 12 edges, 8 nodes and 6 faces. The elements fit together in a Cartesian grid similar to a voxel mesh. However hexahedral elements are not necessarily of the same uniform size and shape, both of which may vary across a mesh. When performing the initial sweep of geometry, it is the hexahedral mesh that Ansys Discovery software seeks to apply. Due to the quadrilateral shape of the elements, Hexahedral meshes need fewer elements which offers great efficiency in numerical analysis. Hexahedral meshes can be applied to structures and fluid enclosures.

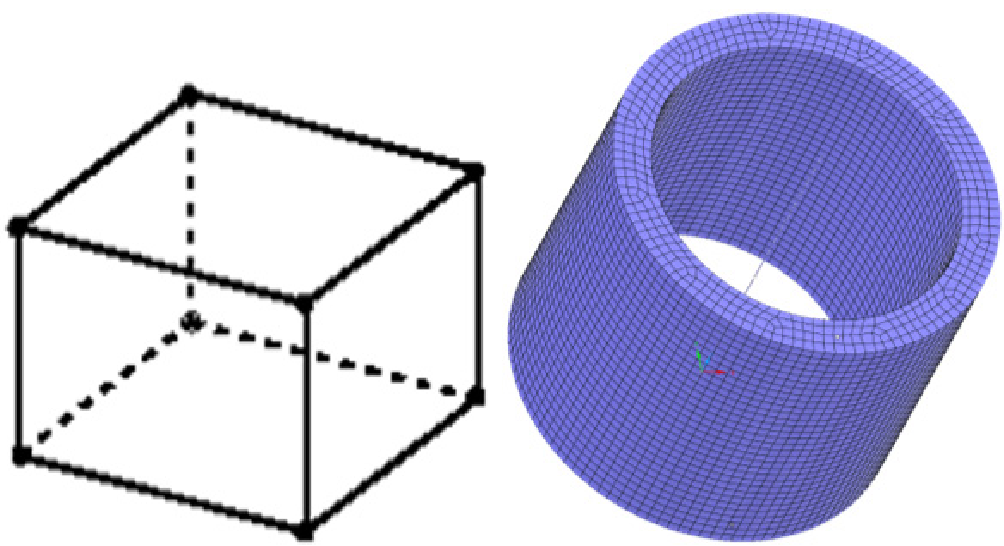

Figure 6: (Left) A Hexahedral element with 12 edges, 8 nodes and 6 faces. (Right) A pipe with a hexahedral mesh

4.2 Tetrahedral Meshing in Refine Mode

A tetrahedral mesh is also a network of nodes, elements and edges. A tetrahedral element has 6 edges, 4 nodes and 4 faces in the shape of a 3D tetrahedron. Tetrahedral meshes can also be applied to structures and fluid enclosures. A tetrahedral mesh is particularly useful when geometry is non-uniform or asymmetric. Tetrahedral meshes offer great flexibility which makes them well suited to geometries with complex curvatures, however more elements may be required in the mesh.

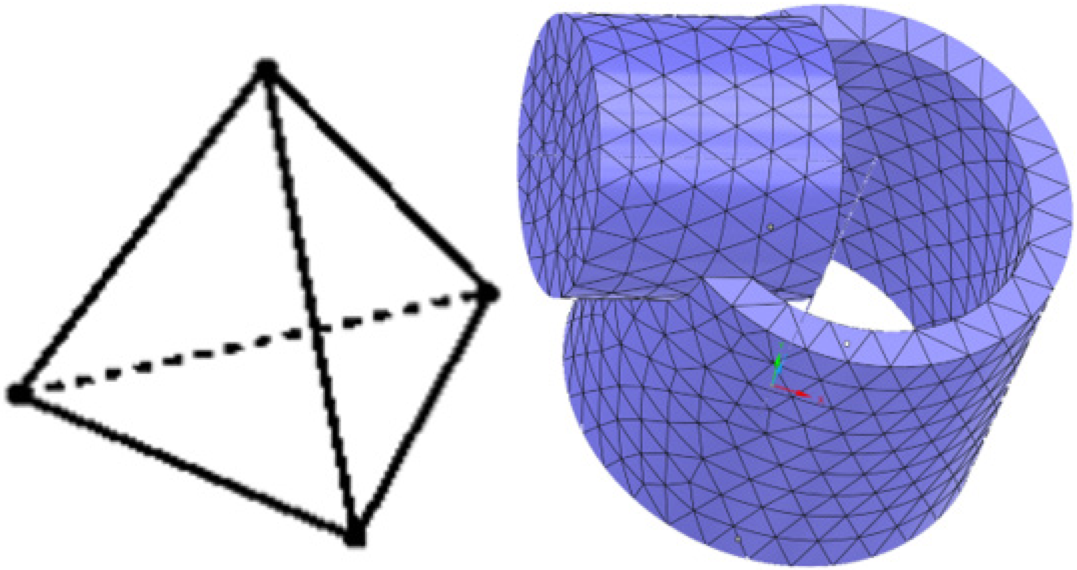

Figure 7: (Left) A Tetrahedral element with 6 edges, 4 nodes and 4 faces. (Right) An symmetric pipe with a tetrahedral mesh

4.3 Polyhedral Meshing (Fluids only)

A polyhedral mesh uses multiple triangles, quadrilateral and polygonal elements to provide a highly efficient and accurate mesh compared to a Tetrahedral mesh. Polyhedral meshes are very adaptable and can more closely represent the shape of geometry using fewer elements; which enables faster and more accurate numerical analysis. In Ansys Discovery software users may choose between creating a Tetrahedral or Polyhedral mesh for more details please see the Ansys Discovery Documentation.

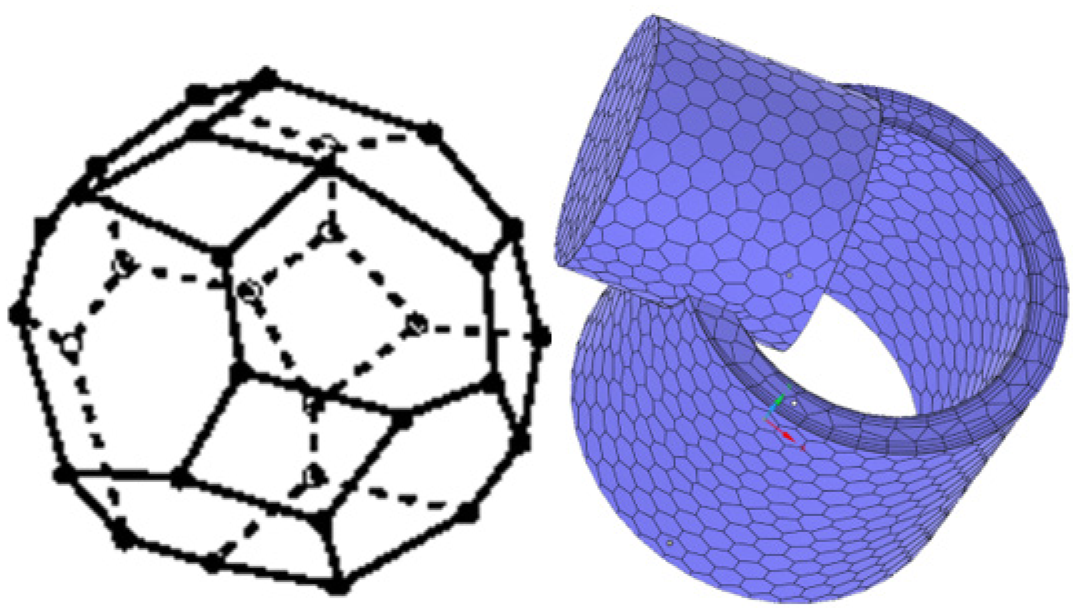

Figure 8: (Left) A Polyhedral element with multiple edges, nodes and faces. (Right) An asymmetric pipe with a polyhedral mesh. The rim of the pipe contains hexahedral and tetrahedral elements.

5. Mesh refinement

Further refinement of meshes in the Refine mode may be made using the Global and Local fidelity tools. Global fidelity affects the entire mesh and works as described earlier. Local fidelity enables the user to select regions of the mesh that may need enhancement, any changes made only affect the selected areas.

The quality of meshes can be evaluated and adjusted using statistical information such as node and element counts. The size of elements in a mesh is important, if too large results may not be accurate, if too small the time for calculation may be prolonged. The number of nodes and elements in a mesh may give an indication regarding the ratio of nodes to elements. More detailed information about mesh quality like element quality, aspect ratio, Jacobian ratio, Warping factor, Parallel deviation, Maximum corner angle, Skewness and Orthogonal quality can be obtained by creating monitor charts (for more information about monitors please see the Ansys Discovery Space).

6. Conclusion

In this case study users of Ansys Discovery software were introduced to some of the fundamental concepts of meshing. While this was not an exhaustive investigation, all users should be able to confidently create and adjust meshes in Ansys Discovery tool. All users should be confident in recognizing and analyzing the anatomy of meshes discussed in other models. Some users may even feel confident enough to manually create basic meshes in tools like Ansys Mechanical and Ansys Fluent.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to structures, fluids, or heat transfer (physics areas supported by Ansys Discovery).

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com