

SolidWorks Simulation Finite Element Analysis Overview





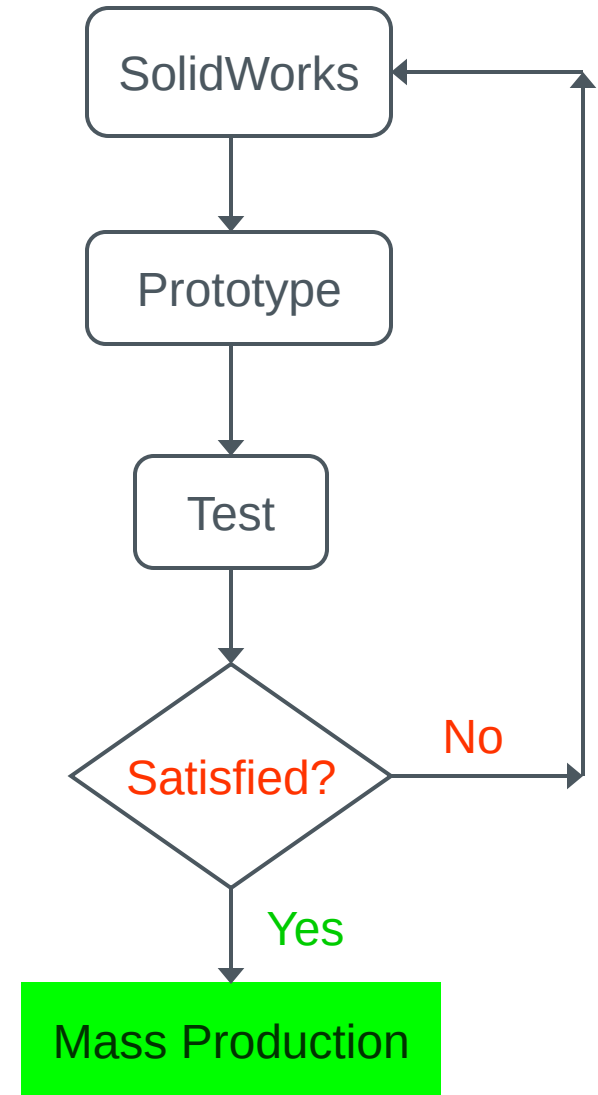
What is SolidWorks Simulation?

- SolidWorks Simulation is a design analysis software that is fully integrated in SolidWorks.
- SolidWorks Simulation simulates the testing of your model's prototype in its working environment. It can help you answer questions like: how safe, efficient, and economical is your design?
- SolidWorks Simulation is used by students, designers, analysts, engineers, and other professionals to produce safe, efficient, and economical designs.



Traditional Design Cycle

- Use SolidWorks to build the model.
- Manufacture a prototype.
- Test the prototype under various loading conditions. Instrumentation is needed in most cases.
- Based on results, modify the model in SolidWorks, build a new prototype, and test it again until you are satisfied.





Benefits of Analysis

- Design cycles are expensive and time-consuming.
- Analysis reduces the number of design cycles.
- Analysis reduces cost by testing your model using the computer instead of expensive field tests.
- Analysis reduces time to market.
- Analysis can help you optimize your designs by quickly simulating many concepts and scenarios before making a final decision.



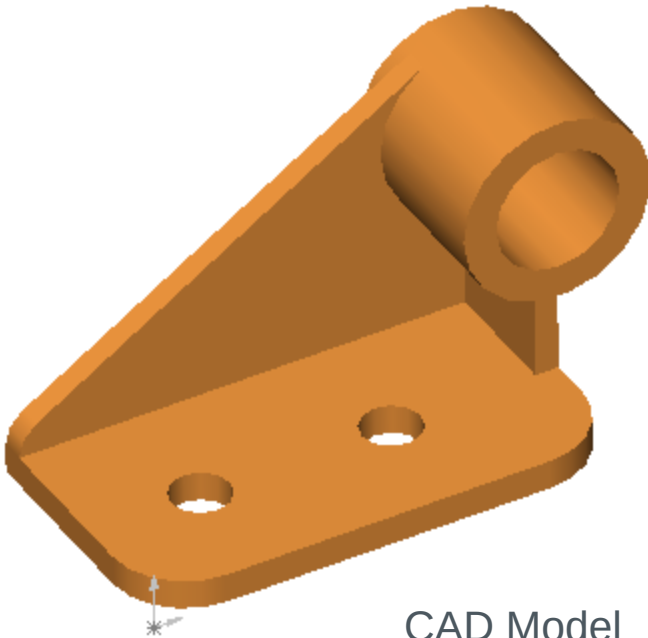
The Finite Element Method

- Analytical solutions are only available for simple problems. They make many assumptions and fail to solve most practical problems.
- SolidWorks Simulation uses the Finite Element Method (FEM). Analysis using the FEM is called Finite Element Analysis (FEA) or Design Analysis.
- FEA is very general. It can be used to solve simple and complex problems.
- FEA is well-suited for computer implementation. It is universally recognized as the preferred method of analysis.

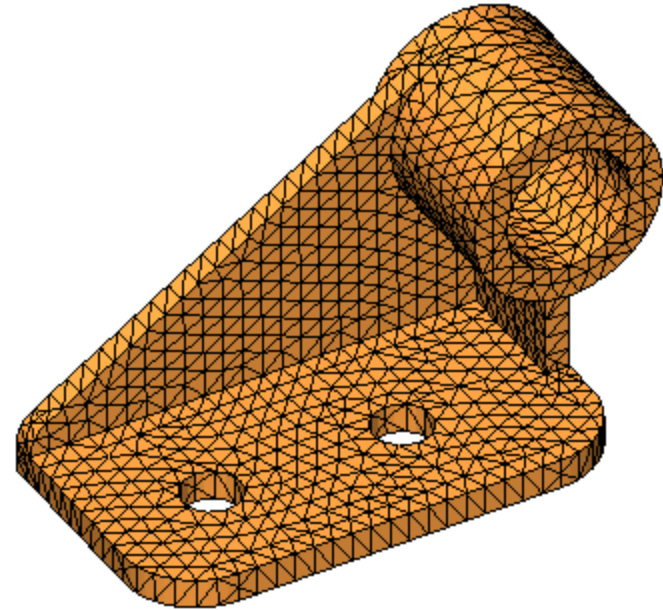


Main Concept of Design Analysis

The FEM replaces a complex problem by many simple problems. It subdivides the model into many small pieces of simple shapes called elements.



CAD Model

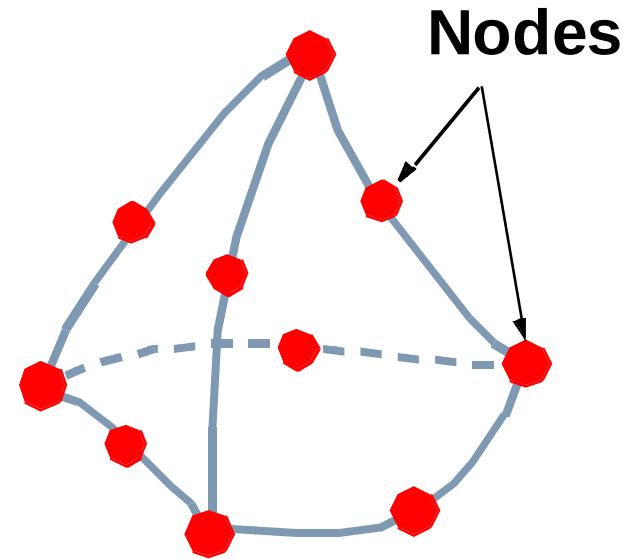


CAD Model Subdivided into Small Pieces



Main Concept of Design Analysis

- The elements share common points called nodes. The behavior of these elements is well-known under all possible support and load scenarios.



Tetrahedral Element

- The motion of each node is fully described by translations in the X, Y, and Z directions. These are called degrees of freedom (DOF). Each node has 3 DOF.



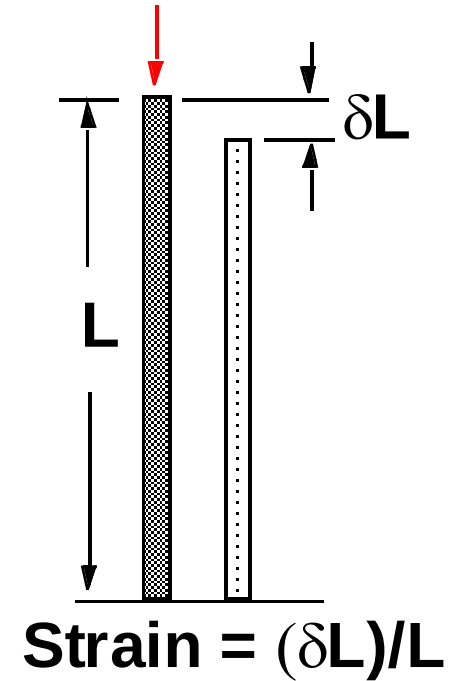
Main Concept of Design Analysis

- SolidWorks Simulation writes the equations governing the behavior of each element taking into consideration its connectivity to other elements.
- These equations relate the unknowns, for example displacements in stress analysis, to known material properties, restraints and loads.
- Next, the program assembles the equations into a large set of simultaneous algebraic equations. There could be hundreds of thousands or even millions of these equations.



Main Concept of Design Analysis

- In static analysis, the solver finds the displacements in the X, Y, and Z directions at each node.
- Now that the displacements are known at every node of each element, the program calculates the strains in various directions. Strain is the change in length divided by the original length.
- Finally, the program uses mathematical expressions to calculate stresses from the strains.





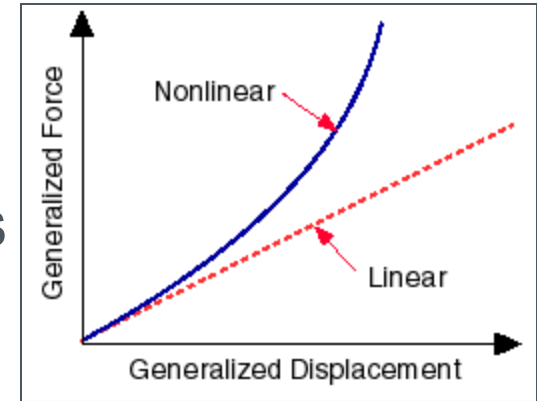
Types of Analysis: Static or Stress Analysis

- This is the most common type of analysis. It assumes linear material behavior and neglects inertia forces. The body returns to its original position when loads are removed.
- It calculates displacements, strains, stresses, and reaction forces.
- A material fails when the stress reaches a certain level. Different materials fail at different stress levels. With static analysis, we can test the failure of many materials.



Types of Analysis: Nonlinear Static Analysis

- Use nonlinear analysis, when at least one of the following conditions applies:
 - a) The stress-strain relationship of the material is not linear.
 - b) Induced displacements are large enough to change the stiffness.
 - c) Boundary conditions vary during loading (as in problems with contact).
- Nonlinear analysis calculates stresses, displacements, strains, and reaction forces at all desired levels of loading.





Types of Analysis: Buckling Analysis

- Slender models subjected to compressive axial loads tend to undergo sudden large lateral deformation. This phenomenon is called buckling.
- Buckling could occur before the material fails due to high stresses.
- Buckling analysis tests failure due to buckling and predicts critical loads.

Axial Load



This slender bar subjected to an axial load will fail due to buckling before the material starts to fail due to high stresses.



Types of Analysis: Frequency Analysis

- Each body tends to vibrate at certain frequencies called natural frequencies.
 - For each natural frequency, the body takes a certain shape called a mode shape.
- 
- Frequency analysis calculates the natural frequencies and associated mode shapes.
 - In theory, a body has an infinite number of modes. In FEA, there are as many modes as DOF. In most cases, the first dominant modes are considered for the analysis.



Types of Analysis: Frequency Analysis

- Excessive stresses occur if a body is subjected to a dynamic load vibrating at one of its natural frequencies. This phenomenon is called resonance.
- Frequency analysis can help you avoid resonance and solve dynamic response problems.





Types of Analysis: Thermal and Thermal Stress Analysis

Thermal Analysis

Calculates the temperature at every point in the model based on thermal loads and thermal boundary conditions. The results include thermal flux and thermal gradients.



Thermal Stress Analysis

Calculates stresses, strains, and displacements due to thermal effects and temperature changes.





Types of Analysis: Optimization Analysis

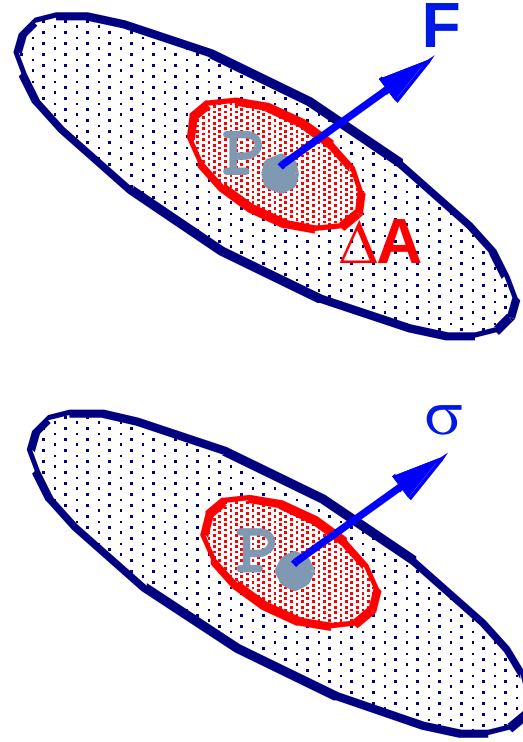
Calculates the optimum solution to a problem based on the following:

- Objective: Sets the goal of the analysis, like minimizing the material of the model.
- Design variables: Specifies acceptable ranges for dimensions that can change.
- Constraints: Sets the conditions that the optimum design should meet, like specifying a maximum value for stresses.



What is Stress?

- When a load is applied to a body, the body tries to absorb the effect by generating internal forces that vary from one point to another.
- The intensity of these forces is called stress. Stress is force per unit area.
- Stress at a point is the intensity of force on a small area around that point.



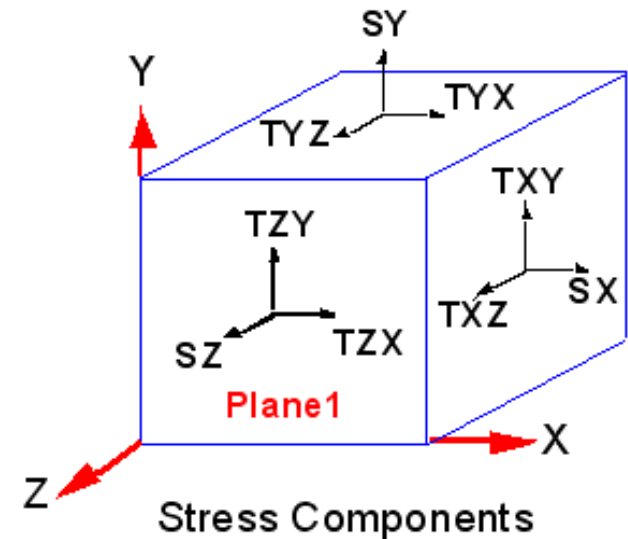
$$\sigma = \lim_{\Delta A \rightarrow 0} F/\Delta A$$



What is Stress?

- Stress is a tensor quantity described by magnitude and direction in reference to a certain plane. Stress is fully described by six components:

- S_X : Normal stress in the X-direction
- S_Y : Normal stress in the Y-direction
- S_Z : Normal stress in the Z-direction
- T_{XY} : Shear stress in the Y-direction on YZ-plane
- T_{XZ} : Shear stress in the Z-direction on YZ-plane
- T_{YZ} : Shear stress in the Z-direction on XZ-plane



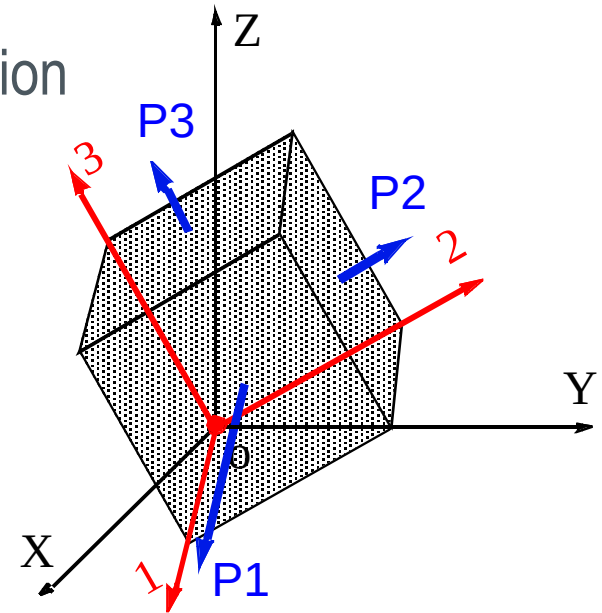
- Positive stress indicates tension and negative stress indicates compression.



Principal Stresses?

Shear stresses vanish for some orientations. Normal stresses at these orientations are called principal stresses.

- P1: Normal stress in the first principal direction (largest).
- P2: Normal stress in the second principal direction (intermediate).
- P3: Normal stress in the third principal direction (smallest).



Axes 1, 2, and 3 are called principal directions and the normal stresses P1, P2, and P3 are called principal stresses.

von Mises Stress

- von Mises stress is a positive scalar number that has no direction. It describes the stress state by one number.
- Many materials fail when the von Mises stress exceeds a certain level.
- In terms of normal and shear stresses, von Mises stress is given by:

$$VON = \left\{ \left(\frac{1}{2} \right) [(SX - SY)^2 + (SX - SZ)^2 + (SY - SZ)^2] + 3(TXY^2 + TXZ^2 + TYZ^2) \right\}^{1/2}$$

- In terms of principal stresses, von Mises stress is given by:

$$VON = \left\{ \left(\frac{1}{2} \right) [(P1 - P2)^2 + (P1 - P3)^2 + (P2 - P3)^2] \right\}^{1/2}$$



Stress Intensity

- Stress intensity (INT) is defined as the difference between the largest and smallest principal stresses:

$$\text{INT} = P1 - P3$$

- Stress intensity is twice the maximum shear stress.



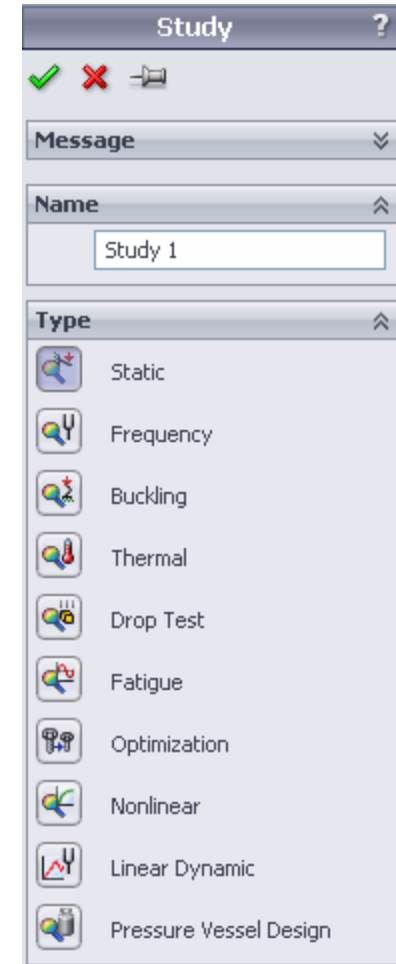
Analysis Steps

1. Create a study to define the type of analysis.
2. Define material for each component.
3. Apply restraints and loads.
4. Mesh the model. This is an automatic step in which the program subdivides the model into many small pieces.
5. Run the analysis.
6. View the results.
 - Steps 2, 3, and 4 can be done in any order.



Creating a Study

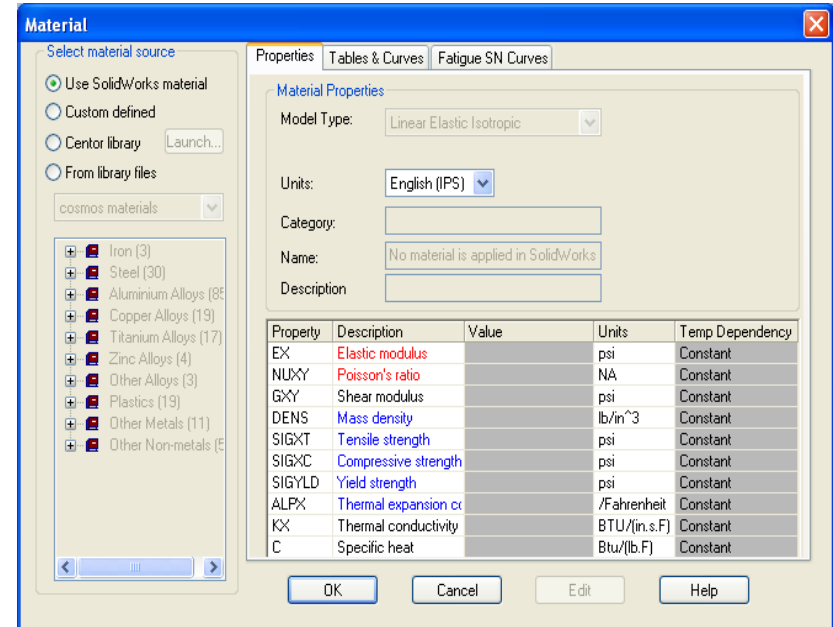
- The first step in analysis using SolidWorks Simulation is to create a study.
- A study simulates a test case or a what-if scenario. It defines analysis intent (type), materials, restraints, and loads.
- You can create many studies and the results of each study can be visualized at any time.





Defining Materials

- Results depend on the material used for each component.
- You can select a material from the library or you can define material properties manually.
- You can also add your own material properties to create customized material libraries.

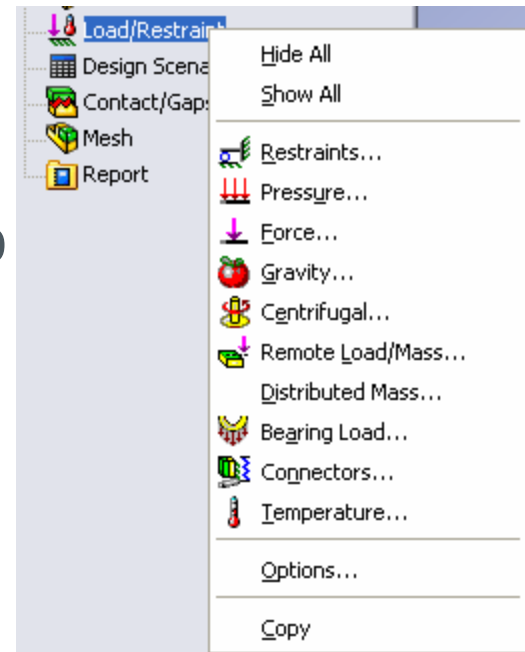


- Materials can be isotropic or orthotropic. Isotropic materials have the same properties in all directions. Orthotropic materials have different properties in different directions (like wood).



Defining Restraints and Loads

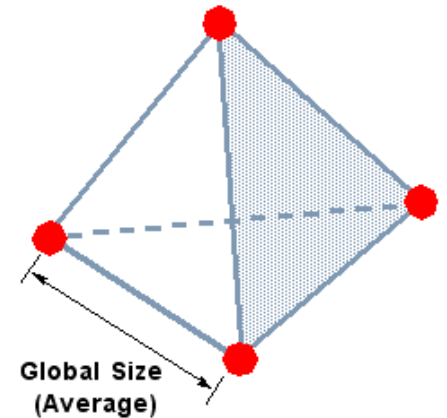
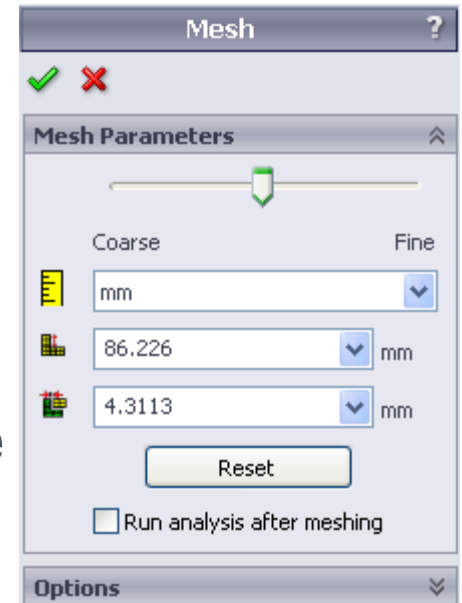
- Restraints define how the model is supported. A body that is not restrained may move indefinitely as a rigid body.
- Adequate restraints should be applied to prevent rigid body motion.
- Loads include forces, pressure, torque, centrifugal, gravitational, prescribed nonzero displacements, and, thermal loads. Special options for bearing and remote forces are also available.





Meshing

- Meshing subdivides the model into many small pieces called elements for mathematical simulation.
- Smaller elements give more accurate results but require more computer resources.
- The program suggests an average global element size for meshing. This is the average length of an element side.
- In critical regions (concentrated loads, irregular geometry) you can apply **Mesh Control** to reduce the element size and improve the accuracy of results.





Meshing Types

- You choose the Mesh Type when you create a study. You can choose: Solid Mesh, Shell Mesh Using Mid-Surfaces, Shell Mesh Using Surfaces, Mixed Mesh, and Beam Mesh.
- Use Solid Mesh for bulky models.
- Use Shell Mesh Using Mid-Surfaces for thin simple models with constant thickness.
- Use Shell Mesh Using Surfaces to create shells with different thicknesses and materials on selected faces.
- Use Mixed Mesh when you have bulky as well as thin bodies in the same model.
- Use Beam Mesh to model structural members.





Meshing

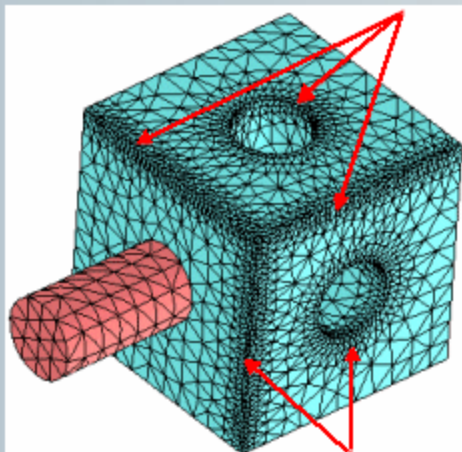
- Based on the element size, the program places points (nodes) on the boundaries and then it fills the volume with 3D tetrahedral elements for solid mesh or 2D triangular elements for shell mesh.
- You must mesh the model after any change in geometry. Material, restraint, and load changes do not require remeshing.



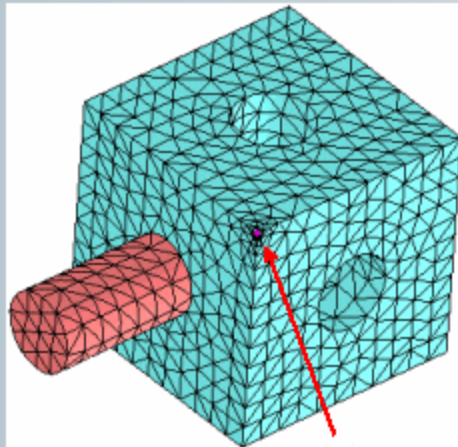
Mesh Control

Examples of Mesh Control:

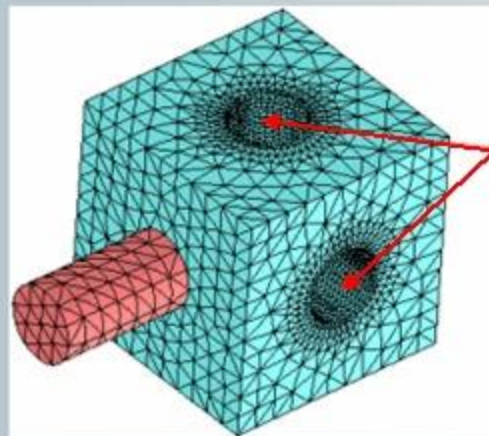
Mesh control applied to selected edges



Mesh control applied to a vertex



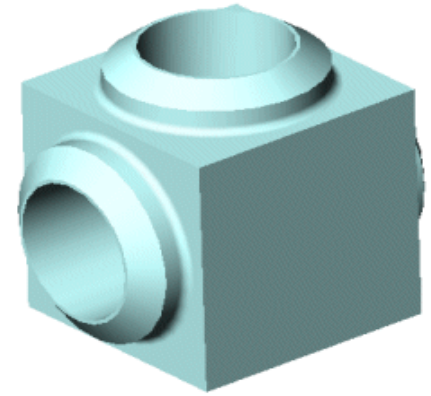
Mesh control applied to selected faces



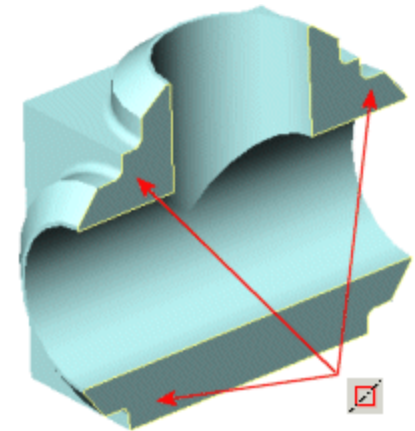


Using Symmetry

- Using symmetry reduces the problem size and improves results.
- Symmetry requires that geometry, loads, material properties, and restraints are symmetrical.
- Requirements of symmetry restraints:
 - Solid models: All faces that are coincident with a plane of symmetry are prevented from moving in the normal direction.
 - Shell models: All edges that are coincident with a plane of symmetry should be prevented from moving in the normal direction and rotating about the other two orthogonal directions.
- Symmetry restraints should be avoided in frequency and buckling studies.



Model symmetrical with respect to one plane.

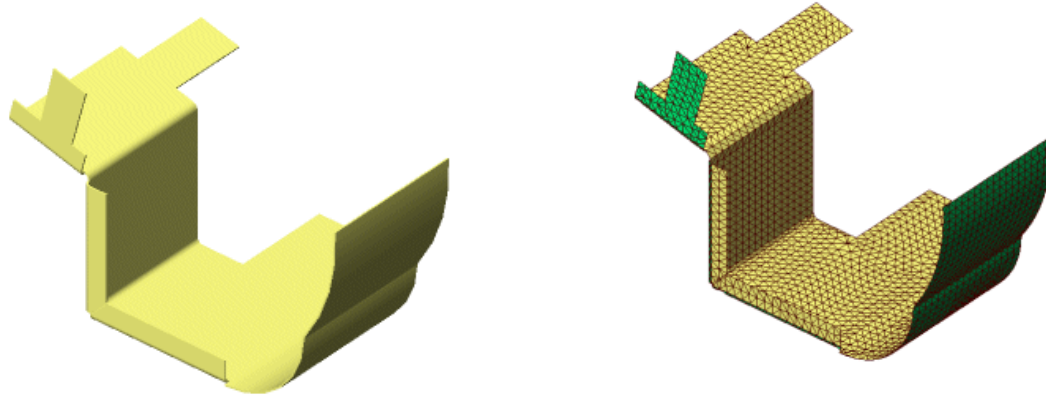


Half of the model with symmetry restraints applied.



Shell Mesh

- You can use shell mesh instead of a solid mesh to model thin parts.

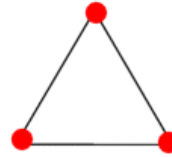


Shell elements resist membrane and bending forces.

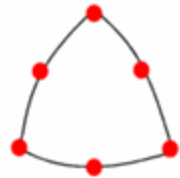


Shell Mesh

- The draft quality shell mesh uses linear triangular elements with three corner nodes.



- The high quality shell mesh uses parabolic triangular elements with three corner and three mid-side nodes.

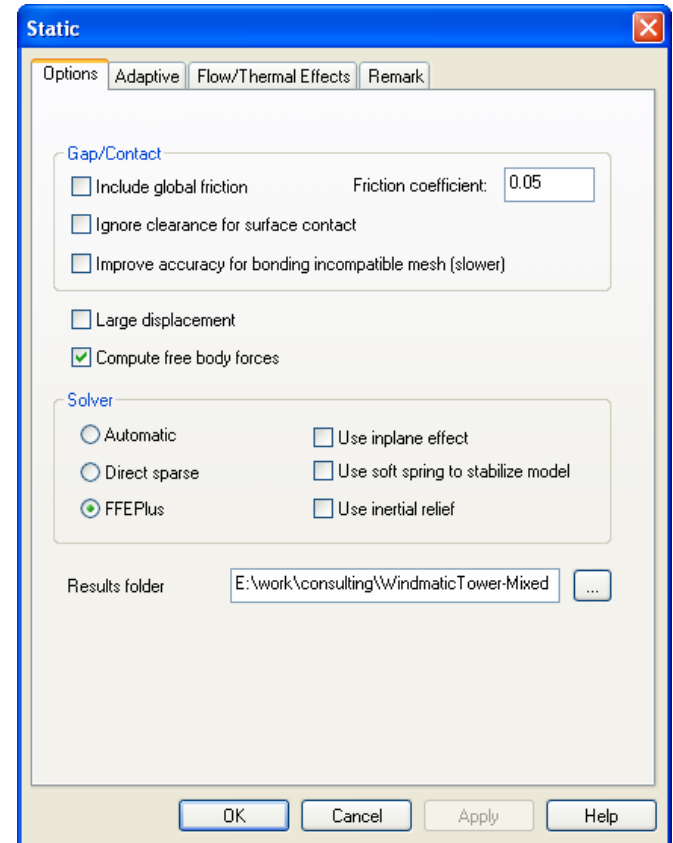


- Each node in a shell element has six DOF: three translations (global X, Y, Z dir) and three rotations (about the global X, Y, and Z axes)



Adaptive Methods for Static Studies

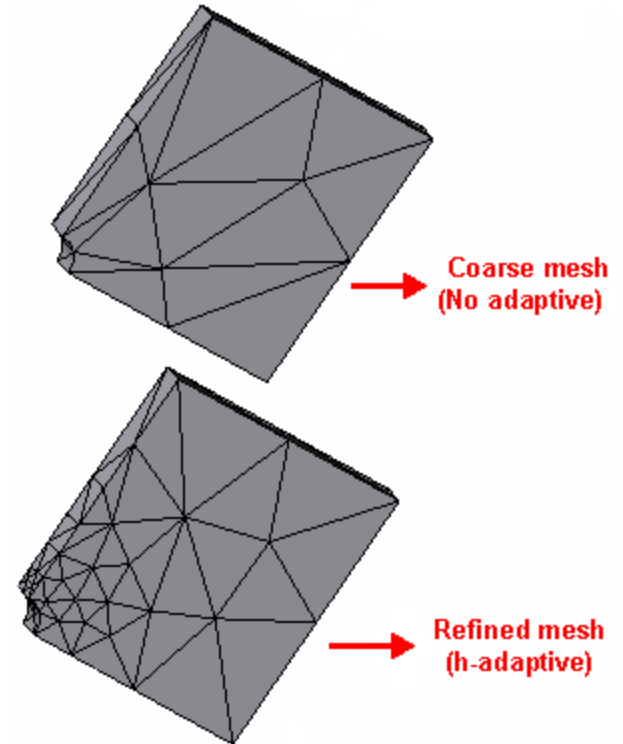
- Critical regions are locations of concentrated forces and irregular geometries (sharp corners) where stress concentrations occur.
- In these critical regions, the results on common nodes of adjacent elements diverge significantly. Adaptive methods can improve the accuracy of the results automatically.
- Adaptive methods are based on error estimation and curve-fitting techniques.





Adaptive Methods for Static Studies

- The **h-adaptive** method refines the mesh (smaller element sizes) by using more elements in critical regions.
- The **p-adaptive** method uses higher element order (polynomial order of the displacement field) to reduce the errors. It does not change the element size.





Running Analysis

- After defining materials, applying restraints and loads, and meshing your model, you run the analysis.
- During analysis, the program calculates the results. This step includes intensive number crunching. In many cases the program will be solving hundreds of thousands of simultaneous algebraic equations.
- SolidWorks Simulation has state-of-the art, fast and accurate solvers.



Visualizing Results

- After completing the analysis, you can visualize the results.
- SolidWorks Simulation provides advanced easy-to-use tools to visualize the results in few clicks.
- Use section and iso plots to look inside the body.
- The Design Check Wizard checks the safety of your design for static studies.
- SolidWorks Simulation generates a structured Internet-ready report for your studies.