

FEA REFRESHER

All the FEA basics so that you ACE
SOLIDWORKS Simulation Associate Exam

1st Edition

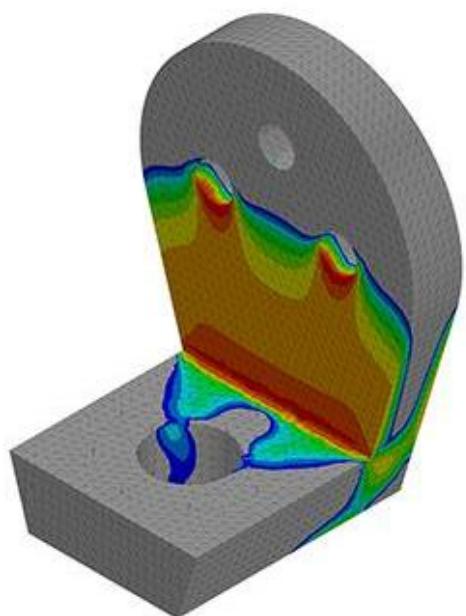
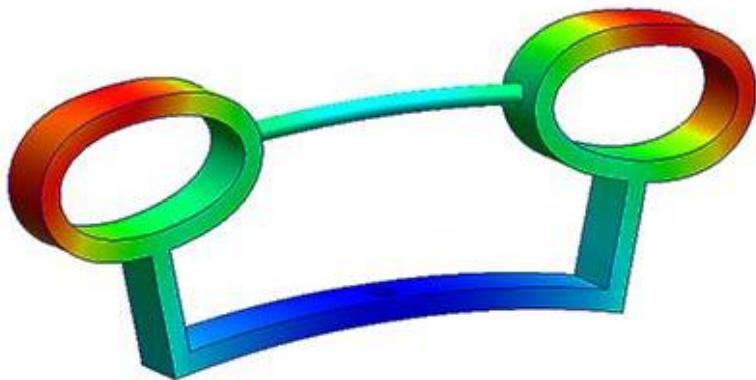


Table of Contents

Table of Contents.....	2
Stress	3
Static Analysis.....	4
Part Type & Materials.....	4
Fixtures.....	5
Loads.....	6
Axisymmetry & Cyclic Symmetry.....	7
Connectors	8
Nodes & Degrees of Freedom (DOF).....	9
Beam & Truss elements	9
Shell element.....	10
Solid element.....	11
Mesh.....	11
Beam Mesh.....	11
Shell Mesh	11
Solid Mesh	11
Mixed Mesh.....	12
Re-meshing	15
Factor of Safety (FOS).....	15
Results.....	16
Conclusion.....	18
More Resources.....	19



SOLIDWORKS Simulation is a powerful tool to virtually test and analyze your designs under real-world conditions, saving time and money on actual prototyping.

This refresher offers a brief overview of key FEA concepts to supplement the learnings of the [TforDesign SOLIDWORKS Simulation program](#). It is NOT a replacement of FEA-related university courses. While not an exhaustive exploration, it provides a solid foundation for understanding SOLIDWORKS Simulation and prepares you for the CSWA-Simulation exam.

Let's begin by familiarizing ourselves with some essential terminology commonly used in FEA.

Stress

Stress refers to the amount of force exerted per unit area.

For a given material in the linear/elastic section, the applied stress is directly proportional to the resulting deformation (strain). Fig 1 shows how different materials react to increasing levels of stress and strain.

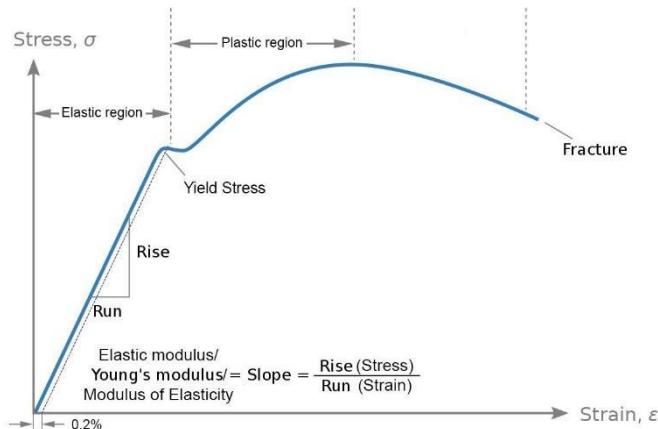


Fig 1: Stress-Strain Curve (Image courtesy: Wikipedia)

Once the stress surpasses the **yield stress**, the material undergoes permanent deformation and does not return to its original shape. At this point, the material becomes plastic and behaves non-linearly.

The yield stress is determined at **0.02% of strain**.

According to SOLIDWORKS Simulation, the structure/material fails at yield stress.

Now, let's delve into the world of **linear static analysis** in SOLIDWORKS Simulation. We will follow the sequence in which static analysis are conducted and discuss the theory behind each step.

Static Analysis

Linear static analysis assumes the following conditions. Those assumptions apply by default to all SOLIDWORKS studies.

- Loads are applied slowly and gradually.
- Loads become constant after reaching their full magnitude. (Static analysis cannot be applied if the load varies with time).
- The response of the system (displacements, stresses, and strains) is directly proportional to the applied loads (except for problems involving no penetration and virtual wall contact conditions).
- Resulting displacements are very small compared to the model geometry
- Inertia and damping effects are neglected.



Note:

In static analysis, non-uniform pressure can also be applied. This pressure is described by a second-order polynomial coefficient in terms of a reference coordinate system.

Part Type & Materials

Different types of models require different treatment based on their type. Let's discuss this further. In a SOLIDWORKS Simulation study, studied entities can have different structural material. For example:

- different components in a solid assembly
- different bodies in a multi-body component
- each shell in a shell model and
- each beam in a beam model

However, when mixed models are present, it is necessary to define the required material properties separately for solids and shells. Also, defining materials in simulation does not update the material assigned to the CAD model in SOLIDWORKS.



The two types of materials to be mentioned here are:



- **Isotropic materials**

They have the same mechanical and thermal properties in all directions, E.g. steel.

- **Orthotropic materials**

They have direction-dependent properties that are unique and independent in three mutually perpendicular directions, E.g. wood, crystals, and rolled metals.



Another important aspect to mention is the treatment of elements as shells. Sheet metal parts with uniform thickness are treated as shells.

Note:

Shells are typically used when the **thickness-to-span ratio is less than 0.05**.

For regular sheet metal parts, a thin shell element formulation is recommended.



Fixtures

After applying the material and understanding the model type, let's discuss fixtures. There are two major types of fixtures:

- **Fixed Geometry** makes all translations and rotations zero.
- **Immovable** makes all translations zero.

For solid models, fixed geometry and immovable have the same meaning: zero translation. Therefore, the immovable option is not available for solid models. However, in shells and beams, fixed geometry and immovable fixtures have different meanings.

Table 1 below shows the allowed orientations for fixed geometries and immovable fixtures for each model type.

	Fixed Geometry	Immovable
Solids	0 translation	N/A
Shells	0 translation 0 rotation	0 translation
Beams		

Table 1: Difference between fixed geometry and immovable fixture for different model types

Loads

Now that we understand fixtures, let's move on to the next step: loads. Basic loads are straightforward.

Force and pressure are the most commonly used types of loads. Force is an external cause that a body experiences as a result of interacting with another body. Whereas, pressure is the force exerted perpendicular to the surface of an object per unit area. By default, both of these are applied with uniform distribution on selected entities.

Now in this section, we will focus on two interesting types of loads: distributed load and remote load. Table 2 shows the differences between each load type and conditions where one should be preferred over the other.

Distributed Load	Remote Load
Used when mass is distributed uniformly. Total mass is distributed proportionally to the area of each face.	If the mass is distributed non-uniformly.
Applied directly to the selected faces.	Applied at a remote location, indirectly at the selected faces.
Transfers no rotational effect	Transfers as a force and equivalent moments(s)
Gravity or centrifugal load must be defined.	-

Table 2: Difference between Distributed Load and Remote Load

Now, let's discuss an interesting topic about types of symmetry.



Axisymmetry & Cyclic Symmetry

If a model is symmetric, we can simulate a pie-section from it instead of simulating the full model. This reduces the size of the problem while still giving accurate results.

The results on the remaining model are deduced from the simulated pie-section.

Let's discuss the two types of symmetry used in simulations.

- **Axisymmetry**

It is present when the model has **rotational symmetry** about an axis. (shown in Fig 2). In this case, the geometry, material properties, loads, fixtures, and contact conditions should also be symmetric throughout the model.



Fig 2: A model symmetric about its axis (Image courtesy: SOLIDWORKS)

- **Cyclic symmetry**

It is present when the model exhibits **circular patterns** about an axis. (shown in Fig 3). In this case, the geometry, restraints, and loading conditions should be similar for all segments (cyclically patterned) making up the model.

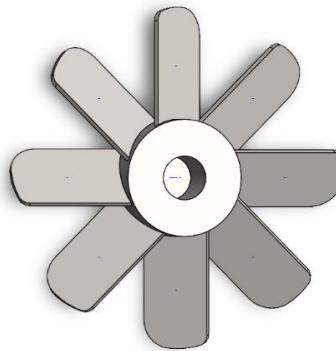


Fig 3: A model having circular pattern (Image courtesy: SOLIDWORKS)

Note: 

If a model with axisymmetry experiences tangential loads, then the pie-sections can deform normal to their planes (as shown in Fig 4). In such cases, cyclic symmetry is used.

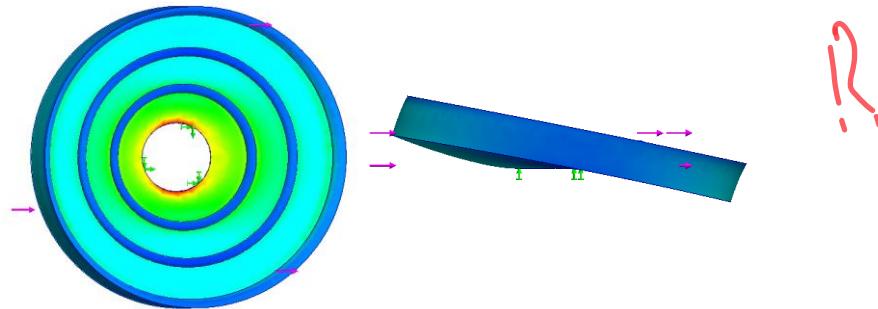


Fig 4: Tangential load on an axisymmetric model

Now let's move on to connectors.

Connectors

Connectors connect two or more parts of an assembly together and help to transfer loads between them. Connectors in SOLIDWORKS Simulation are virtual. We can add a connector without actually modeling it, and it will serve the same purpose as the modeled connector.

Connector types include **spring, pin, bolt, bearing, spot welds, edge welds and links**.

Once a model is simulated, we can see the resulting reaction forces. Similarly, resulting output forces on connectors are available for all types of connectors. With this information, we can go back and modify the connector properties as needed.

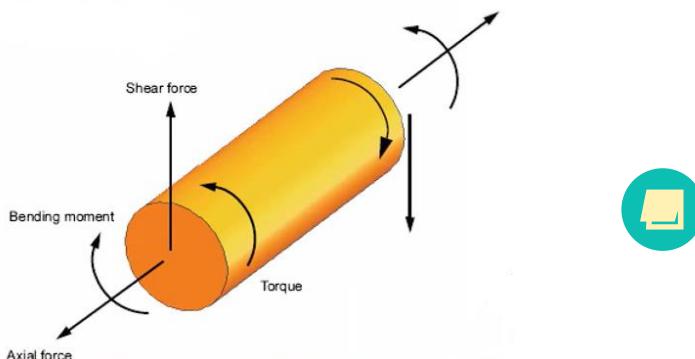


Fig 5: Resulting forces from applied load on a connector (Image courtesy: SOLIDWORKS)

Table 3 shows the available output forces for each type of connector.

	Axial Forces	Shear Forces	Bending Moments	Torque
Spring	✓	✓	✓	✗
Bolt	✓	✓	✓	✗
Bearing	✓	✓	✓	✗
Spot Welds	✓	✓	✓	✗
Edge Weld	✓	✓	✓	✗
Link	✓	✓	✓	✗
Pin	✓	✓	✓	✓

Table 3: Output forces on each type of connector

Next, we'll move on to the concepts of nodes and degrees of freedom (DOF), which are essential for meshing.

Nodes & Degrees of Freedom (DOF)

The smallest unit of a mesh is called an element, which consists of nodes (joints) connected by edges. **Each element has a certain type of movement based on the mesh type.**

Let's discuss the degrees of freedom for each element type:

Beam & Truss elements

- Beam element is defined by 2 end nodes and a cross-section. (Fig 6).
- Each node has 6 DOF (3 translations and 3 rotations).



- They can resist axial, bending, shear, and torsional loads.

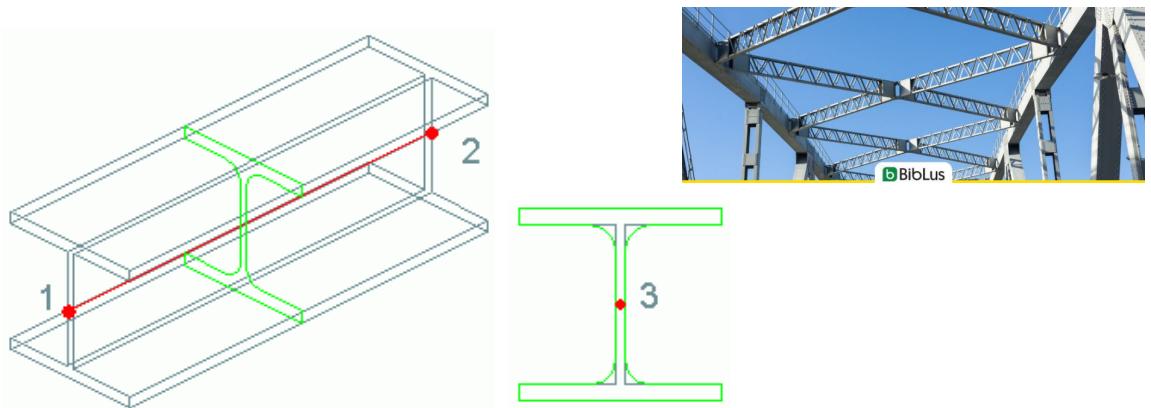


Fig 6: Beam element (Image courtesy: Autodesk)

- A truss is special beam element that can resist axial deformation only.
- Each truss element has 2 nodes, and each node has 3 translational DOF as shown in figure Fig 7.



Fig 7: Truss element (Image courtesy: SOLIDWORKS)

Shell element

- A Draft-quality shell element has 3 nodes. (Fig 8a)
- A High-quality shell element has 6 nodes. (Fig 8b)
- Each node has 6 DOF (3 translations and 3 rotations).

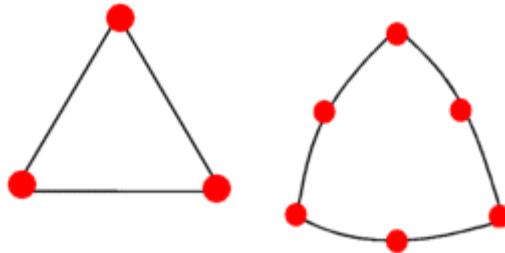


Fig 8: a) Draft-quality shell element, b) High-quality shell element (Image courtesy: SOLIDWORKS)

Solid element

- A Draft-quality solid element is linear tetrahedral with 4 corner nodes. (Fig 9a)
- A High-quality solid element is parabolic tetrahedral with 4 corner nodes and 6 mid-side nodes. (Fig 9b)
- Each node has 3 DOF (3 translations).

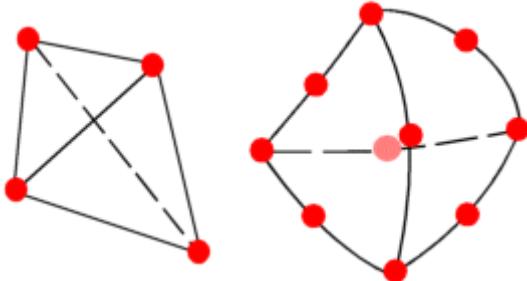


Fig 9: a) Draft-quality solid element, b) High-quality solid element (Image courtesy: SOLIDWORKS)

Mesh

The process of subdividing a model into small pieces is called meshing. We've already talked about the smallest unit of a mesh i.e. an element. Now, let's zoom out and discuss mesh in general.

Creating a mesh is an important step before running an analysis. Different types of mesh are used for different model types in SOLIDWORKS Simulation. Let's explore the various mesh types and when to use each one.

Beam Mesh

- Beam mesh is specifically designed for structural members and weldments.

Shell Mesh

- This type of mesh is suitable for surfaces and sheet metals with uniform thickness.
- It is generated on the surface located at the mid-surface of the shell.

Solid Mesh

- Solid bodies require a solid mesh.

Mixed Mesh

- When there are different geometries present in the model, such as solids, shells, and beams, a **mixed mesh** can be created.
- This type of mesh can be used for both parts and assemblies.

Note: 

A mesh consists of elements, which are composed of nodes and edges. The **global element size** refers to the average length of an element edge.

In a **Standard mesh**, the **global element size** is based on the model's volume and surface area.

In **Curvature-based** and **Blended curvature-based** meshes, **global element size** is not present. Instead, we can set **minimum** and **maximum element sizes**.

To achieve more accurate results, the mesh can be made finer. However, it's important to note that this may increase the time required to create the mesh and compute results.

There are multiple methods to achieve a finer mesh and improve results.

One way is to change mesh density from the **mesh PropertyManager**. Here, we can drag the **slider between Coarse and Fine** which changes the **global element size**. At advanced levels, we can adjust the mesh density locally within a model.

Another method of refining the mesh is by adaptive meshing methods. They improve result accuracy based on error estimation. Once you select a certain type of adaptive setting and run the study, **SOLIDWORKS** automatically and iteratively refines the mesh where needed.

To access these meshing methods, right-click on the name of study > click on Properties > and select Adaptive tab (shown in Fig10).



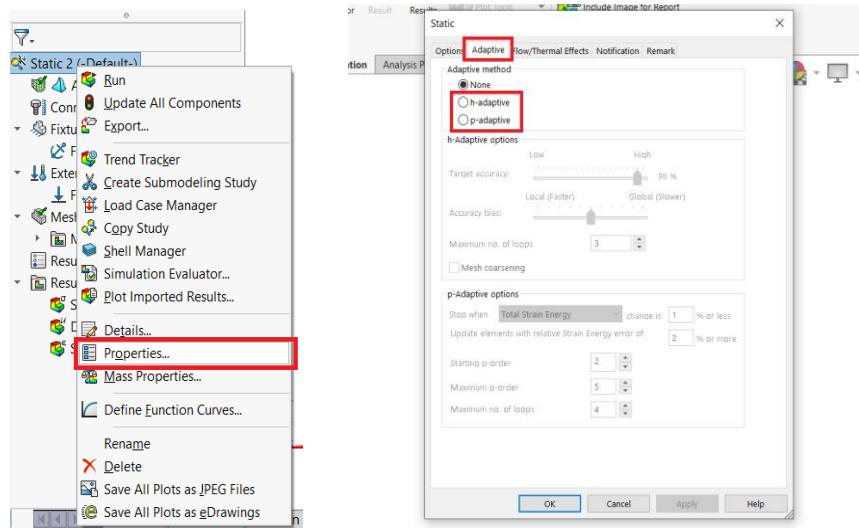


Fig 10: Steps to enable Adaptive meshing

The 2 adaptive meshing methods are:

- **H-Adaptive Method**

It increases the mesh density by using smaller elements in regions with high relative errors. (Fig 11)

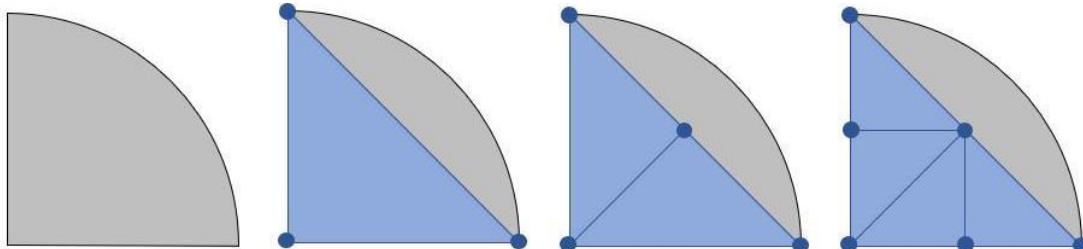


Fig 11: H-adaptive method increases mesh density

- **P-Adaptive Method**

It increases the polynomial order of elements with high relative errors, which increases the number of nodes and improves accuracy. (Fig 12)

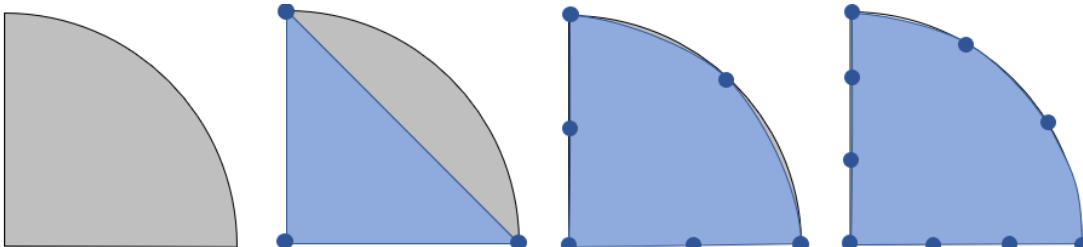


Fig 12: P-adaptive method increases polynomial order of elements

When working with assemblies and multibody parts, two types of meshes can be created for bodies that touch each other: compatible and incompatible.

- **Compatible Mesh**

In a compatible mesh, the elements of touching bodies share the same nodes, resulting in a cohesive mesh without abrupt breaks as shown in Fig 13.

This type of mesh provides more accurate results compared to the incompatible mesh.

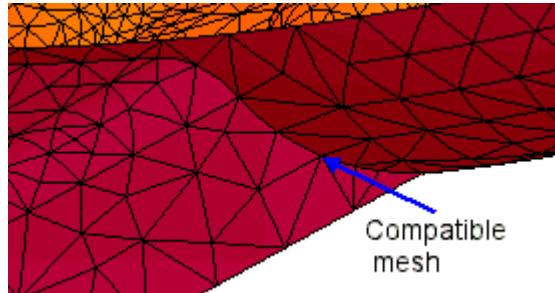


Fig 13: Compatible mesh with continuity (Image courtesy: SOLIDWORKS)

- **Incompatible Mesh**

In an incompatible mesh, each body is meshed separately, and nodes between touching bodies are not shared. This can result in abrupt breaks as illustrated in Fig 14.

This is generated when compatible meshing fails at a common interface between bodies. Failing of a compatible mesh is the software's algorithm-based thing and is not relevant to us.

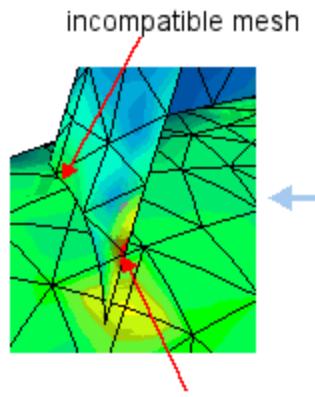


Fig 14: Incompatible mesh with abrupt breaks (Image courtesy: SOLIDWORKS)

As we're discussing touching bodies here, let's touch on a few aspects linked to it.

In cases where an assembly has interferences, a solid mesh can only be applied if the **shrink-fit contact condition** is defined. If there is no shrink-fit, interferences in the model must be eliminated before meshing.

In another scenario, if you need to specify a bonded contact condition between bodies, they should be touching.

Now let's discuss adjustments and re-meshing.

Re-meshing

When working in the Simulation environment, it is common to make changes to the model and applied conditions, at times even after running the study and obtaining results.

So, let's discuss re-meshing the model. There are certain circumstances that require re-meshing to accommodate applied changes, while in other cases, re-meshing is not necessary.

Table 4 below shows which conditions require a re-mesh and which do not.

Re-mesh before running study	Do not Re-mesh
Change in geometry	Change in material properties
Change in contact conditions	Adding/deleting/modifying loads or restraints
Change in meshing options 	Change in properties of the study
Change in mesh control	Change in options in the Mesh PropertyManager 

Table 4: Conditions which require re-meshing and which do not

Factor of Safety (FOS)

The factor of safety (FOS) is a measure that indicates when a model may fail under an applied load. It is defined as:

$$\text{Factor of Safety (FOS)} = \frac{\text{Max Stress}}{\text{Actual Stress}}$$

The default criterion of FOS is specified for each material and can be checked from material properties.



Note: 



By default, the max von Mises stress criterion is applied to measure the FOS of ductile materials.

By default, the Mohr-Coulomb stress criterion is applied to measure the FOS of brittle materials.

For the failure of ductile materials, the maximum von Mises stress (more accurate) and maximum shear stress are used as failure criteria. For the failure of brittle materials, the Mohr-Coulomb and maximum normal stress are used.

For beam materials, SOLIDWORKS Simulation uses the following allowable stress criteria:

- **Yield stress** if the maximum von Mises or maximum shear stress (Tresca) criterion is selected.
- **Tensile stress** if the maximum normal or Mohr-Coulomb criterion is selected.

Finally, let's move on to the last topic about results.

Results

Here are few notes you should be aware of regarding the study results:

- Displacement at a specific location in a specific direction is always smaller than the resultant displacement at that location (Fig 15).

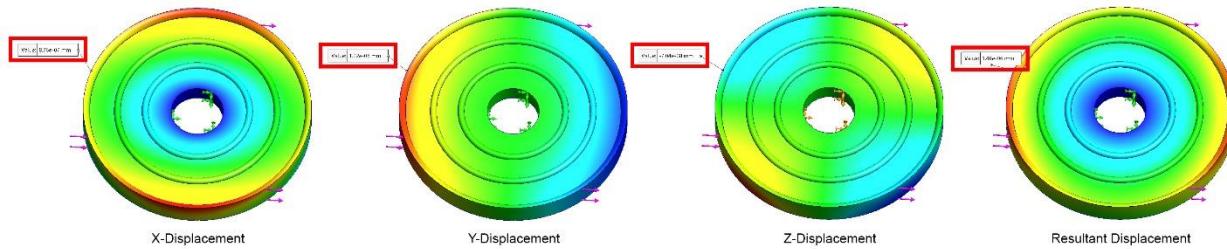


Fig 15: Displacement values at a specific location

- The distance between two nodes can be calculated using mesh and displacement plots (Fig 16). Right-click on Mesh/Displacement plot > select Problr > select “Distance” option > select the 2 nodes from the graphics area > and the distance between the nodes is shown on the model and PropertyManager.

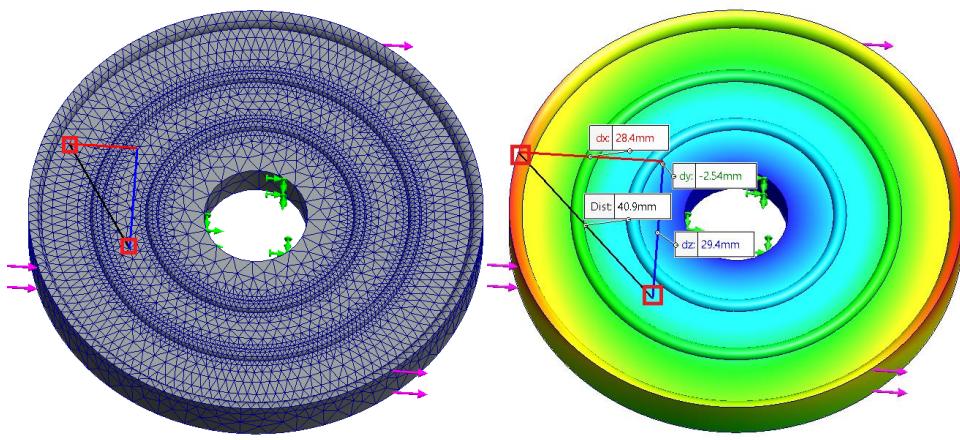


Fig 16: Distance between two nodes

- For rigid bodies, only displacement results are computed.
- The probe tool is used to determine result values at specific locations (nodes) in a model.
- Maximum and minimum value indicators can be displayed on stress and displacement plots (Fig 17). Right-click on the plot > go to Chart Options > check “Show min annotation” and “Show max annotation” options

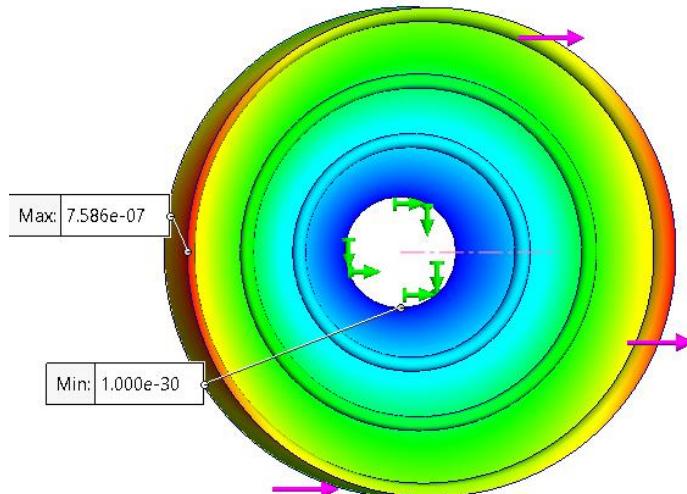


Fig 17: Min and Max value indicators on displacement plot

- To examine portions of the model within a desired stress range, an iso plot with two iso values referencing the lower and upper stress limits can be created (Fig 18). Right-click on the plot > select iso Clipping > select range.



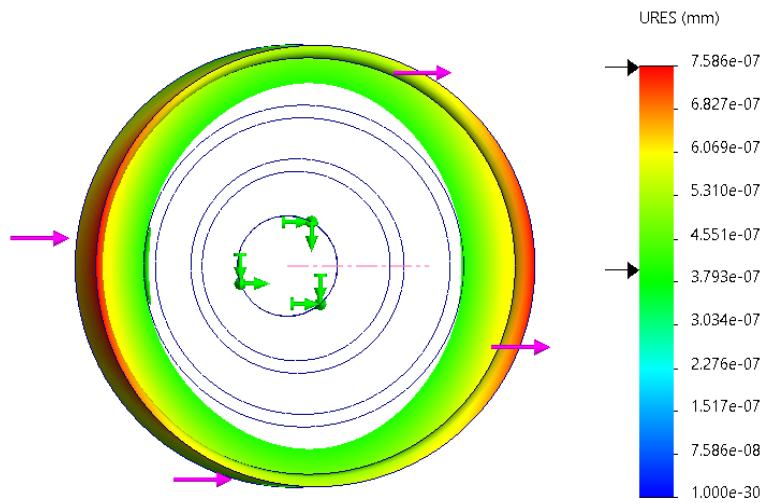


Fig 18: iso plot within defined range

Conclusion

Congratulations on completing the SOLIDWORKS Simulation refresher! In this brief refresher, we explored the basic concepts of Finite Element Analysis (FEA) and their utilization in SOLIDWORKS Simulation.

We covered essential topics such as stress and strain, linear static analysis, fixtures, loads, connectors, Factor of Safety and understanding results.

We also emphasized the importance of understanding different types of degrees of freedom and meshing and remeshing techniques to ensure accurate simulation results.

You're almost there to conquer the CSWA-Simulation exam. Coming up is a case study guide of real-life examples to enhance your understanding regarding SOLIDWORKS Simulation



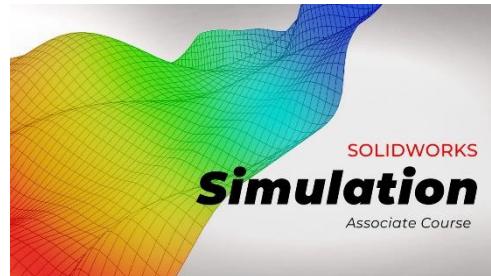
More Resources

TforDesign offers many other design and innovation resources. Here are a few:

SOLIDWORKS Simulation Associate Program:

This refresher is made specifically as supplementary material to our comprehensive Simulation program. This program provides comprehensive training in using SOLIDWORKS simulation tools and Finite Element Analysis, focusing on optimizing designs virtually before prototyping and preparing students for the CSWA-Simulation certification exams.

[Get the full program in TforDesign school](#)
[Get the program in Udemy](#)



3D Printing Opportunities and Applications:

This program offers an extensive understanding of 3D printing technology, including its processes, advantages, developing trends, and applications across various industries. Designed for beginners, it aims to familiarize students with the potential of additive manufacturing in both prototyping and production, empowering them to create innovative solutions and capitalize on 3D printing to generate value in their respective fields.



[Get the program in TforDesign school](#)
[Get the program in Udemy](#)

TforDesign's Design Purrs:

Our Design Purrs contain the latest writings we publish on various topics, including technology, design, and business. Check them out here: <https://www.tfordesign.com/design-purrs.html>

Explore all our programs here: <https://school.tfordesign.com/courses>

Connect with us:

TforDesign School LinkedIn: <https://www.linkedin.com/showcase/tfordesign-school/>

Instagram: <https://www.instagram.com/tfordesign.school/>

Drop us a line: <https://www.tfordesign.com/contact.html>

