BEACON Newsletter - May 2024

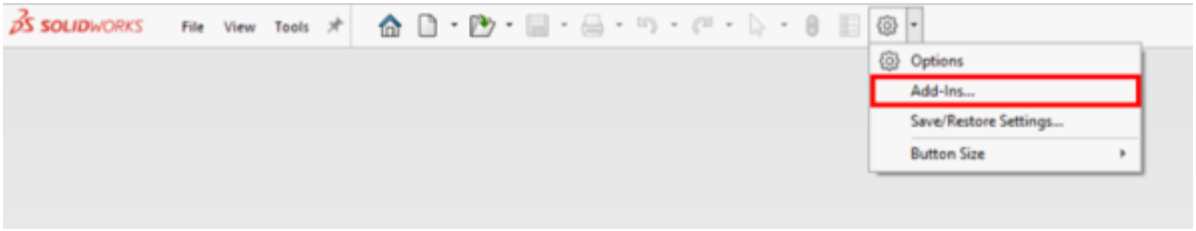
How to Setup your first Static Simulation in SOLIDWORKS

SOLIDWORKS Simulation serves as an effective Finite Element Analysis (FEA) solution, enabling the validation of designs prior to production.

Among the various studies offered by SOLIDWORKS Simulation, static analysis stands out as a widely favored option. This type of analysis operates on the assumption of constant loads and boundary conditions, making it particularly applicable to structures experiencing steady-state conditions. Users utilize static analysis to forecast the response of a structure to applied loads. This guide will outline the fundamental procedures for configuring and executing finite element analysis utilizing SOLIDWORKS Simulation.

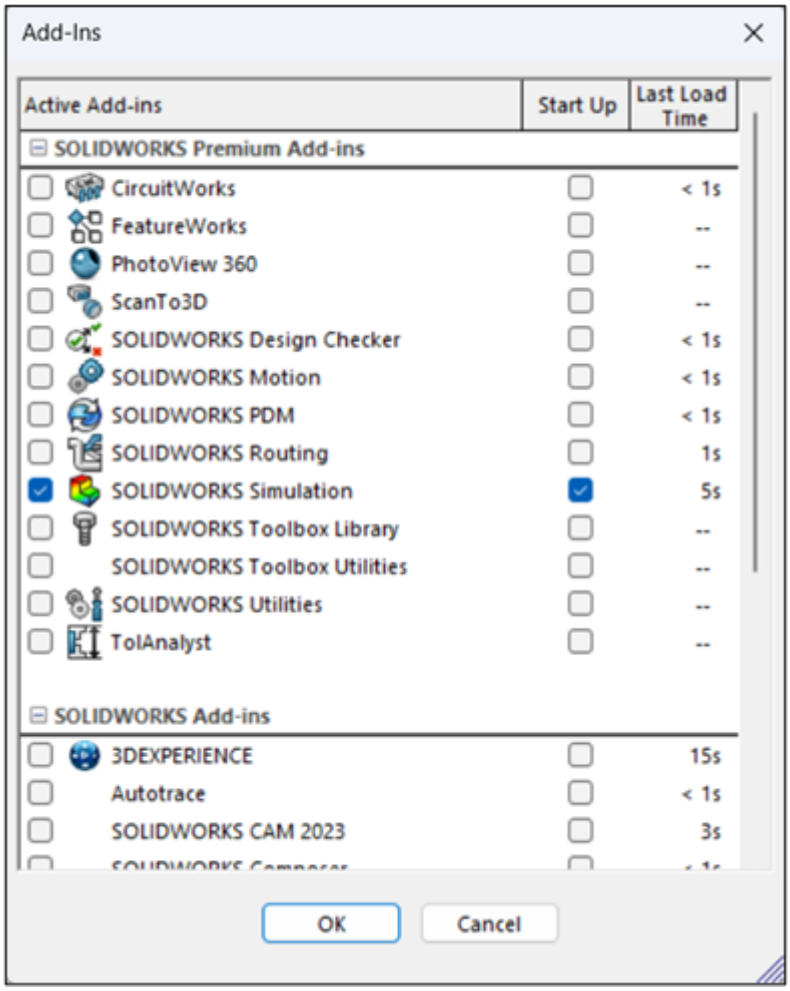
Enable the SOLIDWORKS Simulation FEA Add-In

Before initiating a simulation, it's crucial to ensure that the add-in is enabled. Failure to do so will result in the absence of the simulation tab and its associated tools from the Command Manager. If SOLIDWORKS Simulation is installed, users can activate this feature by navigating to Tools > Add-Ins or by accessing the Add-Ins option from the drop-down menu next to the gear icon at the top of the SOLIDWORKS window.



Add-in selection from the down arrow next to the Options icon (Picture 1)

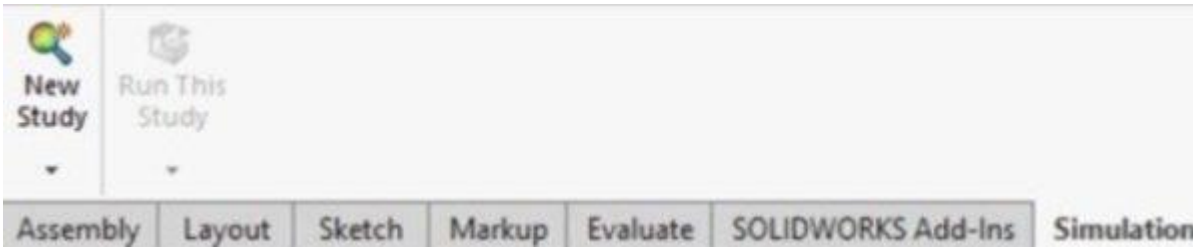
Within the Add-Ins window, SOLIDWORKS Simulation can be enabled by selecting the corresponding checkbox on the left. This action will activate the product for the current SOLIDWORKS session only. To streamline future usage, users can also activate the checkbox in the Start-Up column, ensuring that the add-in is automatically enabled each time SOLIDWORKS is launched, thereby eliminating the need for repetitive manual activation.



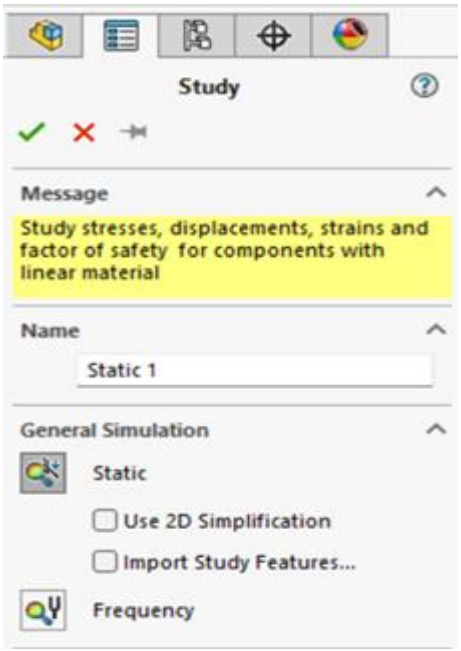
SOLIDWORKS Simulation add-in enabled for active session and at SOLIDWORKS (Picture 2)

Starting the Study

Once the add-in has been enabled and a part or assembly is open, proceed to the Simulation tab and select "New Study." In the Property Manager, activate a Static study and confirm by clicking the green checkmark.



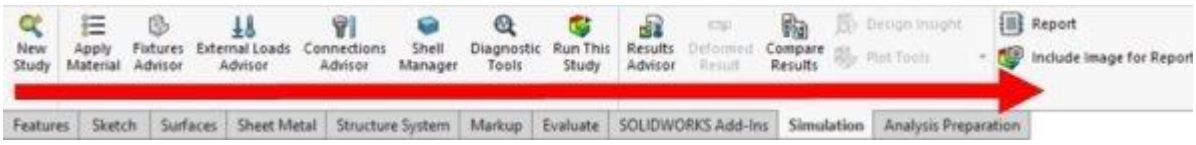
New Study selection from the Simulation tab in the Command Manager (Picture 3)



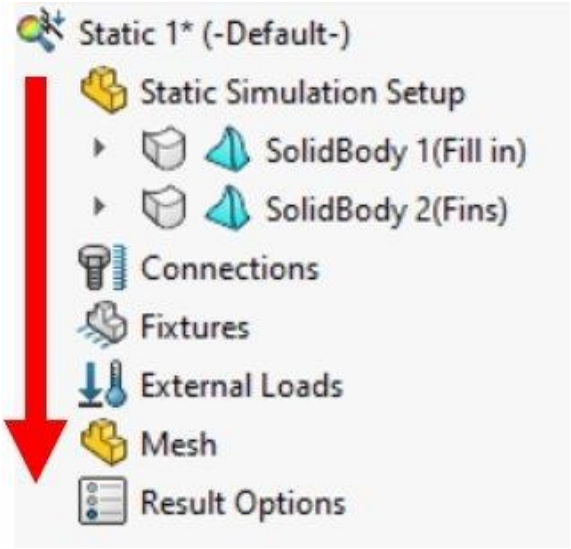
Static study selected from Property Manager (Picture 4)

Study Setup

After the creation of the static study, users can begin defining conditions for the study. Conditions can be applied using commands in the Simulation tab or by right-clicking on a condition in the design tree and selecting from the flyout menu that appears.



Work across the Simulation tab to set up study (Picture 5)



Work down the Simulation tree to set up the study (Picture 6)

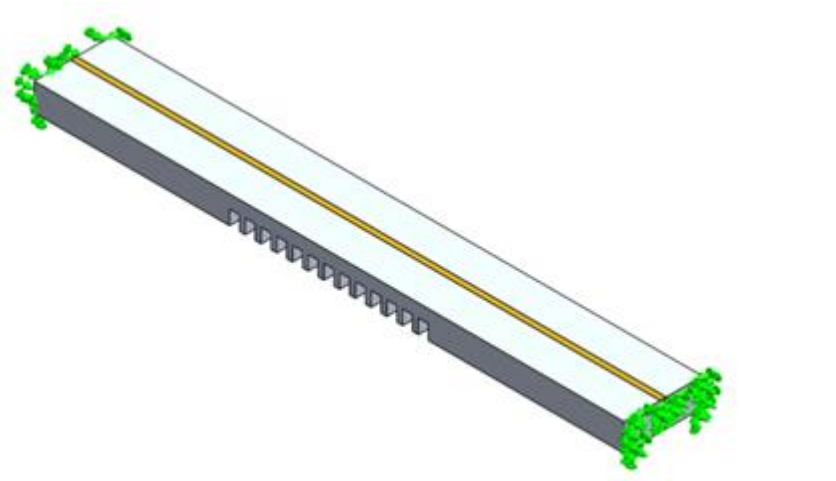
SOLIDWORKS Simulation streamlines the process of applying boundary conditions, providing users with intuitive guidance throughout simulation setup. Here's how to get started:

1. **Assign Material Properties:**
   Begin by assigning material properties to each component in the model. Accurate material data is essential for achieving realistic simulation results. SOLIDWORKS offers an extensive library of pre-defined materials and allows users to create custom materials as needed.

Materials applied to bodies in a study (Picture 7)

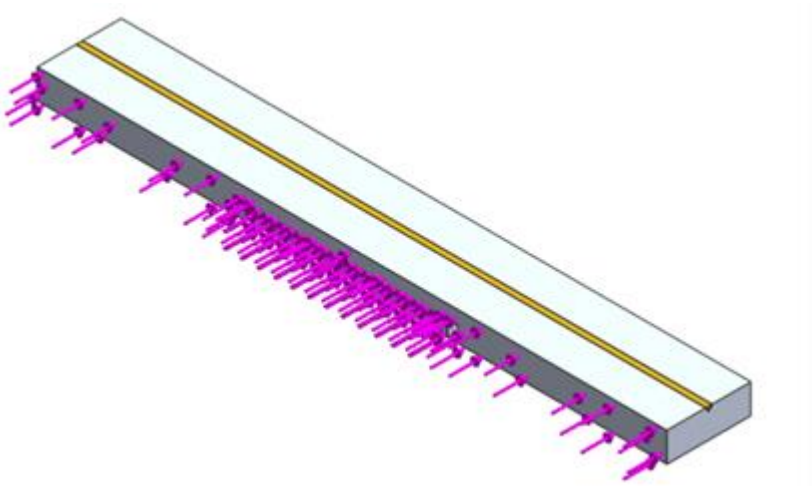
2. **Set Up Connections and Interactions (If Necessary):**
   Connections define how components are linked to one another. Various connector types, such as bolts, bearings, pins, links, and springs, facilitate this process. Interactions govern how components interact when they come into contact under loading conditions.

3. **Define Fixtures:**
   Ensure that the constraints applied accurately reflect the physical behaviour of the model. Use various fixture methods to immobilize geometry, which can be applied to faces, edges, and vertices.



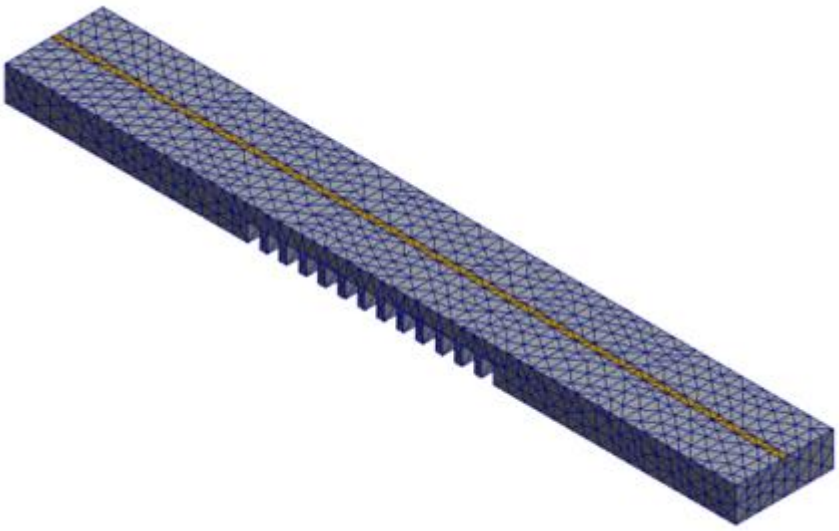
Fixed geometry fixture applied to end faces (Picture 8)

4. **Apply Loads:**
   Define external loading by specifying faces, edges, and vertices. Typical loads include forces, pressure, and torques, which can be uniform or non-uniformly distributed.



600N External force applied to side face (Picture 9)

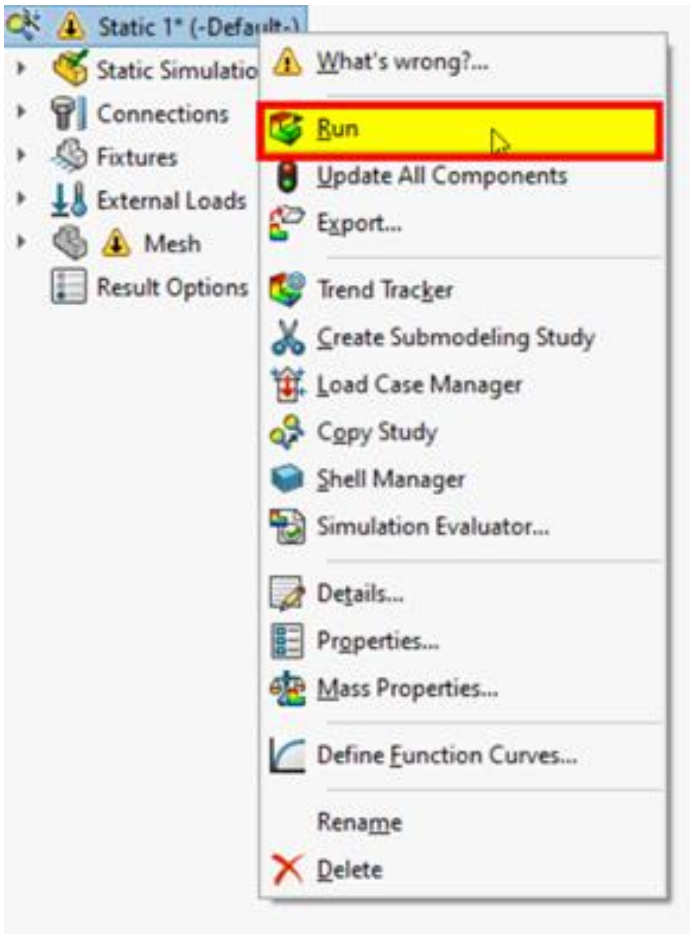
5. **Mesh the Model:**
   Divide the model into smaller triangular elements through meshing, a fundamental aspect of finite element analysis (FEA). Mesh density can vary from coarse to fine, striking a balance between computational time and result accuracy.



Mesh triangles applied to geometry (Picture 10)

6. **Run the Simulation:**

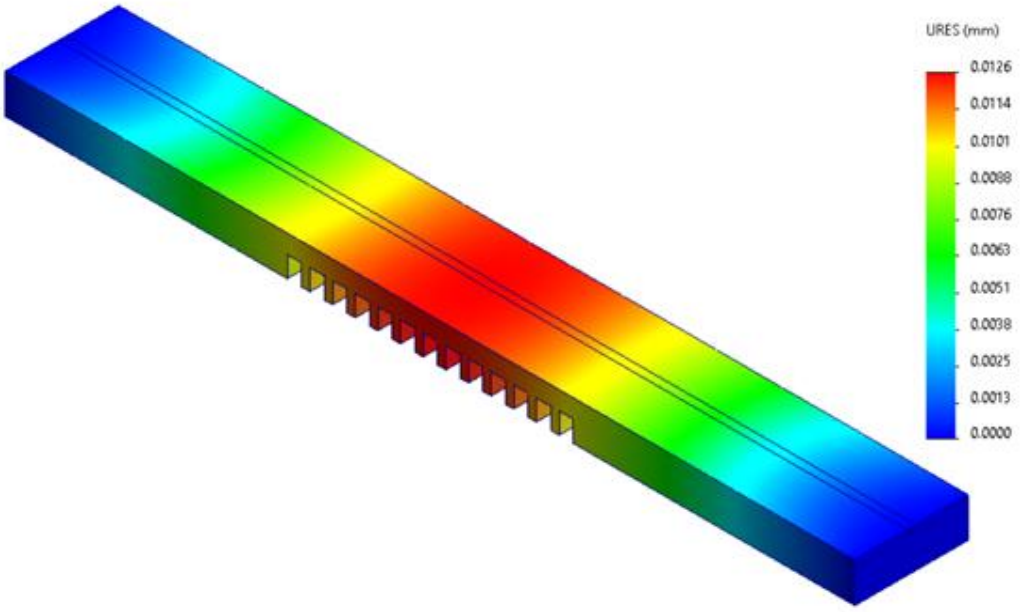
   Once setup is complete, initiate the simulation by right-clicking on the study in the simulation design tree and selecting "Run," or choose "Run This Study" from the Simulation tab in the Command Manager. Depending on model complexity and mesh density, this process may require some time.



Running a study from the Simulation design tree (Picture11)

Running a study from the Command Manager (Picture 12)

7. **Review the Results:**
   Evaluate simulation outcomes using visualization tools provided by SOLIDWORKS Simulation. These tools include stress, strain, displacement, and factor-of-safety plots, aiding in the identification of critical areas for informed design decisions.



View Displacement/ stresses from simulation results (Picture 13)

Conclusion:

Simulation setup is integral for ensuring structural integrity in design. By following these foundational steps, users can leverage simulation to optimize designs, reduce prototyping costs, and deliver superior products to market. As familiarity with SOLIDWORKS Simulation grows, exploration of advanced features and analyses can further enhance design refinement and validation.

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