ABOUT FEATURES OF SIMULATION MODULE IN SOLIDWORKS

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Abstract: In this paper are presented the additional features and analysis steps of Simulation module in SolidWorks, for conducting a more detailed study of designed parts. There are presented the module features, the settings that have to be done for such analysis and the results shown after analysis.

Keywords: SolidWorks, Simulation, FEM analysis.

1. INTRODUCTION

As presented in [1], SolidWorks SimulationExpress Module “offers an easy-to-use first pass stress analysis tool for SolidWorks users. SimulationXpress can help you reduce cost and time-to-market by testing your designs on the computer instead of expensive and time-consuming field tests”. As described in [2], without using a simulation tool, the design process is a time-consuming product development cycles. Typically must be taken following steps: build the model in SolidWorks CAD system, prototype the design, test the prototype in the field, evaluate the results of the field tests and modify the design based on the field test results. This process can continue until a satisfactory solution is reached. By using SimulationExpress/Simulation the same task can be accomplish, reducing costs, by testing the model using the computer rather than field tests, reducing time to market by reducing the number of product development cycles and optimize the designs by simulating concepts and scenarios before making final decisions [2].

Two modules are available for SolidWorks: SimulationXpress that handles part documents (only solid bodies), included in the core software, and Simulation that handles parts and assemblies too, which is available as a separate product, including configuration support and expanded options. SolidWorks Simulation is a design analysis system fully integrated with SolidWorks. SolidWorks Simulation provides simulation solutions for linear and nonlinear static, frequency, buckling, thermal, fatigue, pressure vessel, drop test, linear and nonlinear dynamic, and optimization analyses” [1].

Fig. 1. Generic steps on Simulation: Design → Mesh → Analysis and Results [1]
To begin *Simulation*, open a part, load *Simulation* Add-In and click *Simulation* tab (Command Manager tabs) or menu Simulation.

![Simulation tab or Simulation menu](image)

**Fig. 2. Simulation tab or Simulation menu**

There is a wizard interface of *Simulation* that guides through a step-by-step process to specify Study type, Material, Fixtures and loads, Mesh and Run the simulation, and view the Results.

2. SIMULATION TASK PANNE (SIMULATION ADVISOR)

When you have a part open, the Simulation Task Pane appears in the right side of working board to let you begin the step-by-step process of analysis.

2.1. Study

In this step it’s chosen the type of study: linear and nonlinear static, frequency, buckling, thermal, fatigue, pressure vessel, drop test, linear and nonlinear dynamic, and optimization. In this example was chosen the linear static analysis.

2.2. Material

In this step it is assigned material properties to the part, by choosing one material from library or by defining individual properties. This step is not required if material properties were defined previously in the CAD system.
2.3. Interactions

In this step it’s applied fixtures to faces or edges of the part, as shown in figure 4, and loads like forces or pressures, or both to faces of the part.
2.4. Mesh

In this step it is generated the mesh of finite elements for FEA. The program automatically creates a shell mesh for surfaces and sheet metals with uniform thicknesses. The structural elements are meshed with beam elements and solid bodies are meshed with solid elements. A mixed mesh is created when different geometries (solid, shell, structural elements, etc.) exist in the model. For the type of finite element and mesh density can be used changed settings, as described in [3], as shown in figure 5 – left pane.
2.5. Run

Uses the default settings or changed settings for the simulation and then run analysis. Practically it’s done a static analysis by FEM of studied part, with default or changed settings.

2.6. Results

After the running analysis it can be displayed simulation results as:
- Animation of the part as it stretches under the load and deformed shape of the model (figure 6);
- VonMises stress distribution in the model with or without annotation for the maximum and minimum stress values (figure 7);

![Image of animation and deformed shape of model]

Fig. 6. Animation and deformed shape of model

If the model doesn’t deform as expected, the analysis can be start over with the very first step of simulation for various modification.
Fig. 7. Stress distribution - Displacement distribution in the model with or without annotation for the maximum and minimum displacement values (figure 8);

Fig. 8. Displacement distribution - Factor of safety (FOS) distribution (fig.9).
4. CONCLUSIONS

After running the analysis of the designed part, the results are as follows:
- The allowed yield strength value for material (plain carbon steel) is \(2.2 \times 10^8\) \(\text{N/m}^2\).
- The software automatically adds the load of gravity of \(9.81\, \text{m/s}^2\) (the red arrow in the middle of the model).
- The maximum VonMises stress in the designed part is, at these dimensions, according to analysis results, \(1.229 \times 10^5\) \(\text{N/m}^2\), value significantly lower than the limit.
- The maximum displacement is, at these dimensions, according to analysis results, \(3.345 \times 10^{-5}\) m.
- The safety factor distribution can be displayed as an absolute value or related to VonMises stress, that cannot be overcome (FOS < 1).

In conclusion, by following the steps and the settings described in this paper, as stated in [4], one can perform a detailed study using Simulation module, that can lead to reducing the number of product development cycles and then reducing TTM (time to market). Further can be optimized the design and thus reducing cost by testing the model on the computer, rather than field tests.

For extended study it must be used SolidWorks Simulation that can handle parts and assemblies also. Although is available only as a separate product, it includes configuration support and expanded options. Some of these options are:
- Can be performed an optimization analysis on an assembly.
- Can be specified up to 20 dimensions as design variables.
- Can be specified up to 60 criteria for the model to satisfy.
- Can maximize or minimize any quantity that can be defined as a sensor.

It is obvious that is very useful, both in terms of time and money saving, leading indirectly to better sustainability of designed products.

REFERENCES

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