SimStreamlinedforDesign:

TUTORIAL:
Understanding The Simulation Streamlined Process

Introduction

This tutorial walks you through the straightforward process for evaluating structural performance of your design. The Steps in the Simulation Streamlined process are explained in clear language with illustrations of what to expect as you go through the process. Understanding the Simulation Streamlined process will enable you to effectively use structural analysis to understand your design's structural performance.

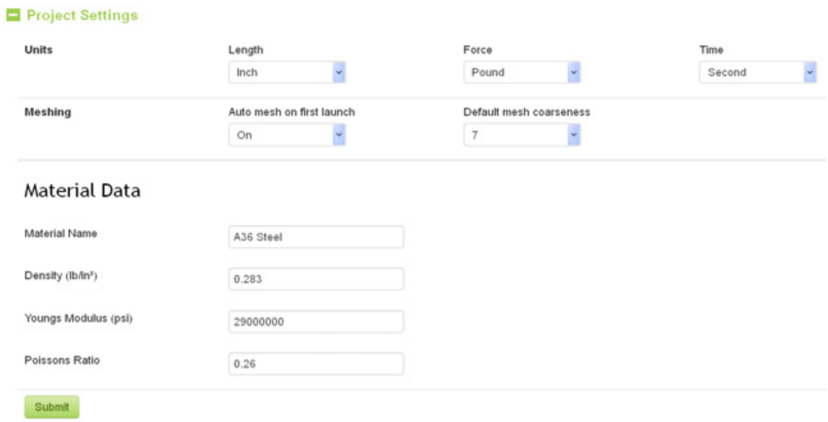
Let's start with a few fundamental concepts. SimForDesign.com utilizes an organizational structure of design Projects and analysis Cases. Projects are designed to group together different evaluations in order to support a design performance understanding or a design decision. You may group analysis Cases within a project as you choose, however, the intention is to group together analysis Cases aimed at achieving a common objective. Splitting analysis Cases into different Projects will allow for easier access and understanding of results.

Creating a Design Project

Creating a Project with SimForDesign.com is very straightforward. Once you have logged in to your account you will be placed in the Projects Page. From this page you can easily add a new project by entering a project name and description and selecting the Create Project button near the top of the page. This will result in opening a table entry for you to provide a Project name and description.

Once you have created the project you can edit the Project by selecting the green edit button for that project from the table of available projects. This will result in taking you to the edit for this Project. The Project edit page has four sections as follows: 1. Project Users, 2. Project settings for units and behavior, 3. Project Users (we will discuss this in a different tutorial), 4. Change Name.

The first step is to define the Project Settings to control Units and meshing. Pull down menus are provided to specify units, material data and Mesh settings to be used by this project. Initial Project Settings are automatically created based on your Account Settings (defaults to inch-pound-second units,A36 Steel and auto meshing on).

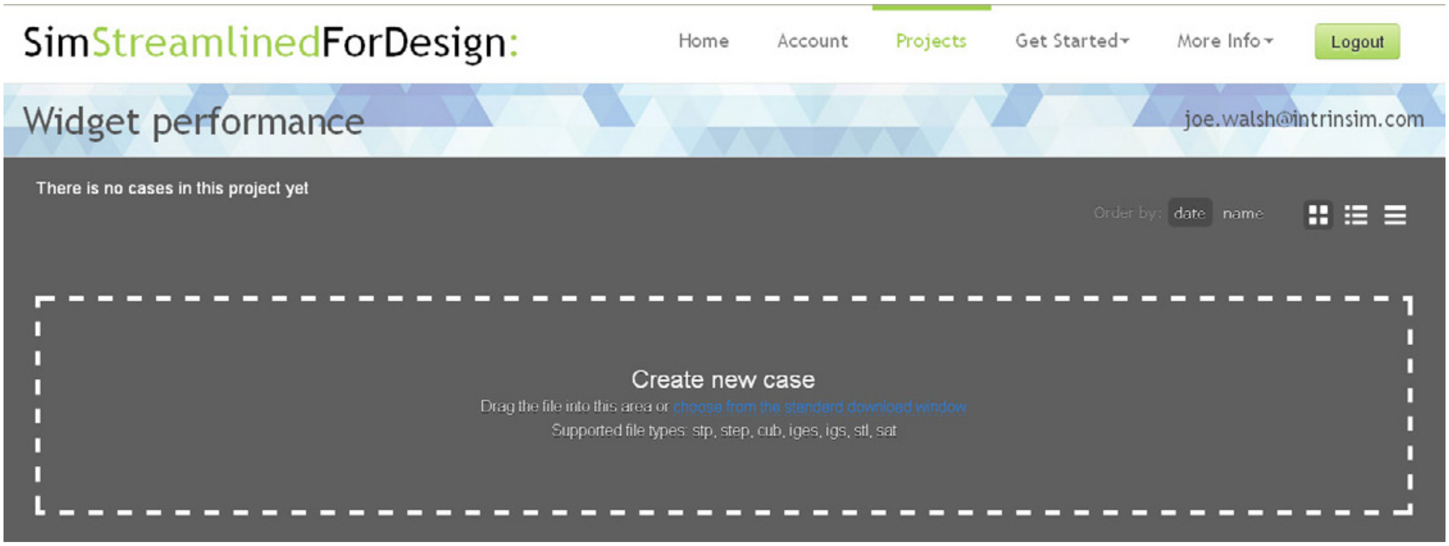


SimStreamlinedforDesign:

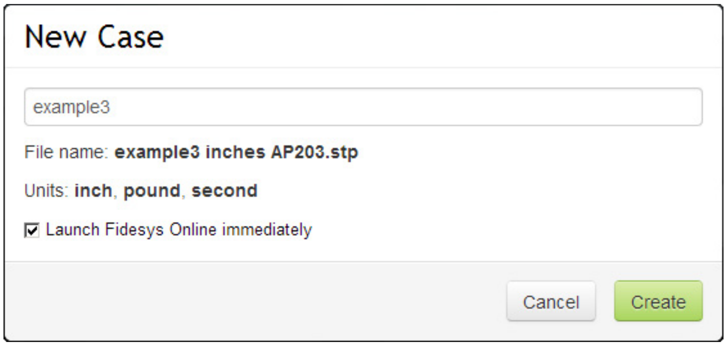
TUTORIAL: *Understanding The Simulation Streamlined Process*

Creating an Analysis Case in a Project

Once the Project has been created and Project Settings defined you can easily define new analysis Cases in the Project by selecting the Project name from the table of Projects or from the Banner section of the page if you are in the Project Edit page. This will place you in the project list of cases with the directions to create a new analysis case.



Selecting a file through the methods provided will result in you being prompted to supply a Case name and the option to select Create Case to activate the Case creation.



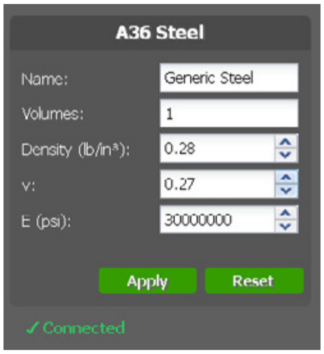
Upon Selecting Create Case your information will be uploaded and your case will be opened in a workbench Powered by Fidesys Online. For this tutorial we uploaded a model of a simplified Universal Joint and displayed the mesh with and without mesh edges.

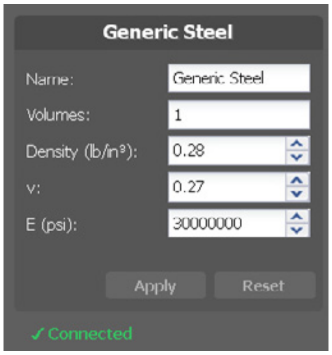
SimStreamlinedforDesign:

TUTORIAL: *Understanding The Simulation Streamlined Process*

Editing Material Properties

Although the Project default material properties were defined in the SimForDesign Project page, you also have the ability to edit those for each analysis case to evaluate the use of alternative materials. By selecting the material in the console you can verify that the material properties have been transferred correctly and modify them to evaluate other materials. A refresh of the page is necessary if you change the material name.



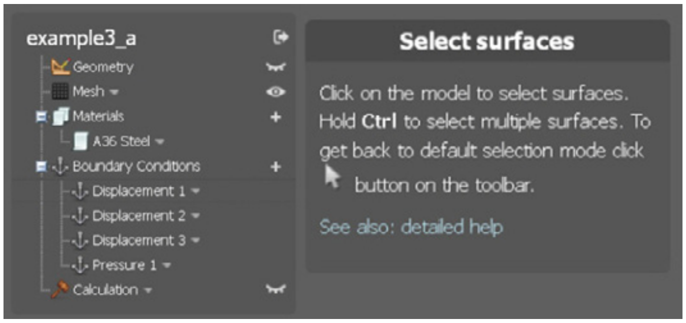


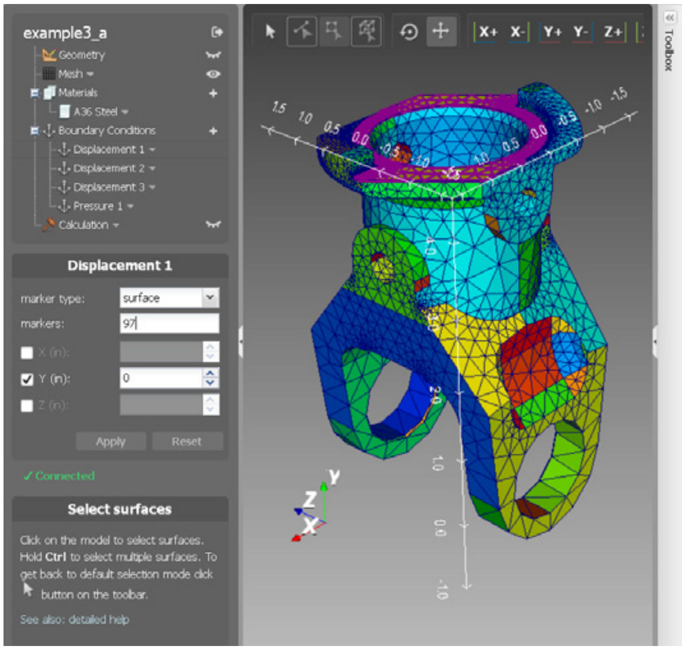
Defining the simulation problem through Boundary Conditions

This step may sound complex but it is actually quite straightforward. There are two parts to defining the simulation problem. The first part is defining the k. This means defining where on your design part that movement is either known or restricted and how that movement is restricted. Displacement constraints are applied in the X, Y, and Z directions either collectively or independently.

Defining constraints correctly involves thinking through how your design is held in place when a load is applied. For this tutorial we are going to restrain the faces on each of the holes of the universal connection normal to these holes. We will start by defining a Displacement of zero in the Y direction for the main face on the top interface of the universal joint. First select the + sign on the Boundary Conditions in the Console and select Displacement. This will open a panel for defining the displacement. After this opens select that you want to define Displacement of 0 in the Y direction (X, Y, and Z are selected by default)

The next step is to define the parts of the model that we wish to apply this Displacement to. The marker type determines the model entity type that you want to select (surface, volume, or curve). The default marker type is surface which we will use. Select the markers input box to make it active. Once the markers input box is active select the desired surfaces using the left mouse button (for multiple surfaces hold the Ctrl button down during selection).

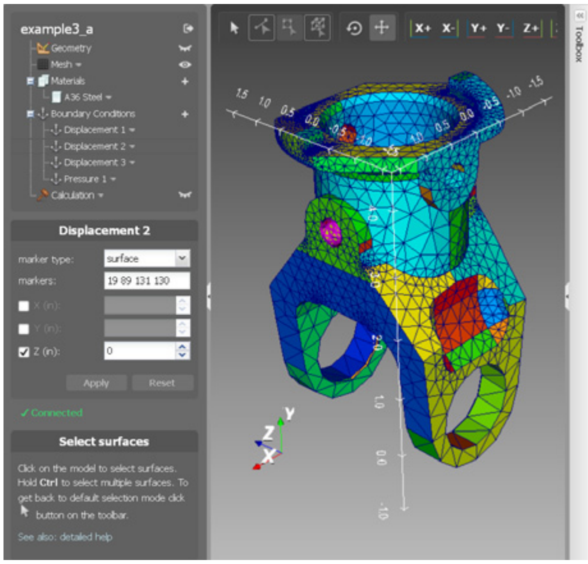




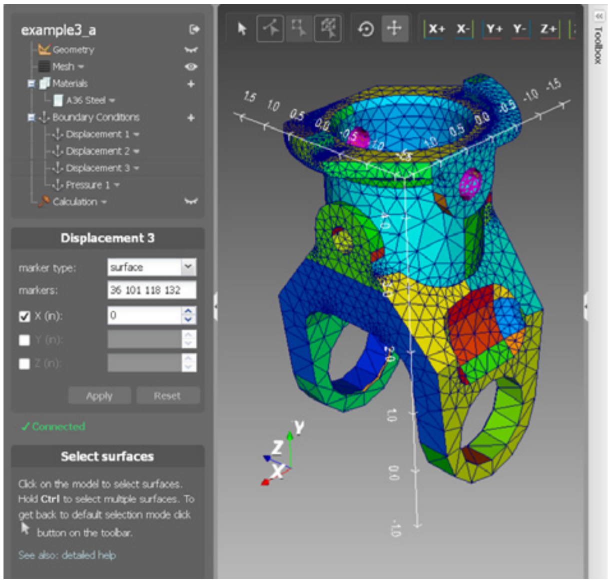
SimStreamlinedforDesign:

TUTORIAL: *Understanding The Simulation Streamlined Process*

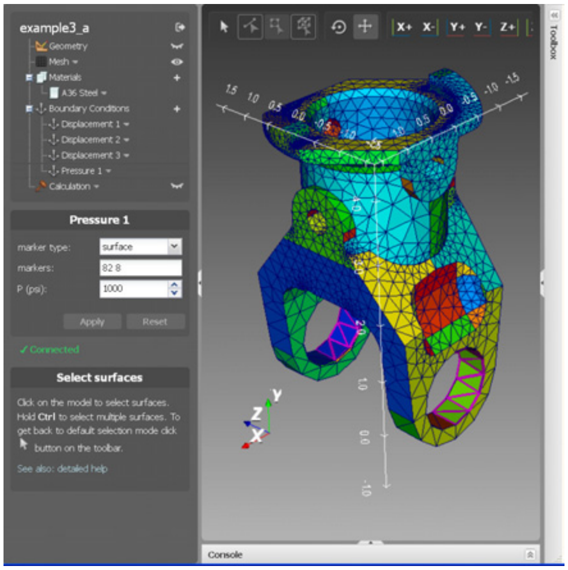
Next we will create a Displacement constraint of 0 in the Z direction on the surfaces of both sides of the lower pin of the universal joint.



Next we will create a Displacement constraint of 0 in the X direction on the surfaces of both sides of the upper pin of the universal joint.



The final step is defining the load that is applied. For this problem we are going to apply a pressure on one side of the faces of the larger pin holes to simulate the load from that connection of the universal joint. Once again we select the + sign on the Boundary Condition in the Console and then we will select Pressure and specify a value of 100 PSI. We then select the marker input box to make it active. Once the markers input box is active select the desired surfaces for the Pressure using the left mouse button (for multiple surfaces hold the Ctrl button down during selection).



SimStreamlinedforDesign:

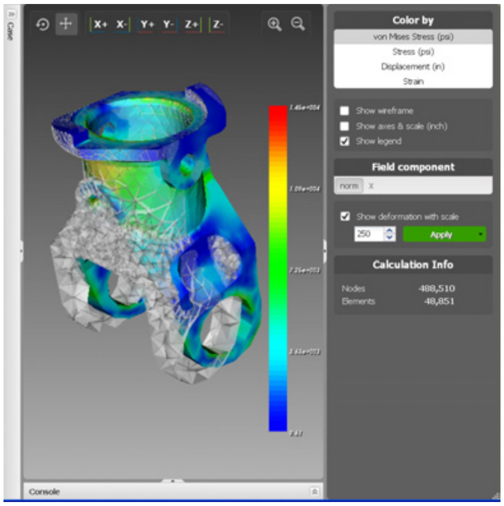
TUTORIAL: *Understanding The Simulation Streamlined Process*

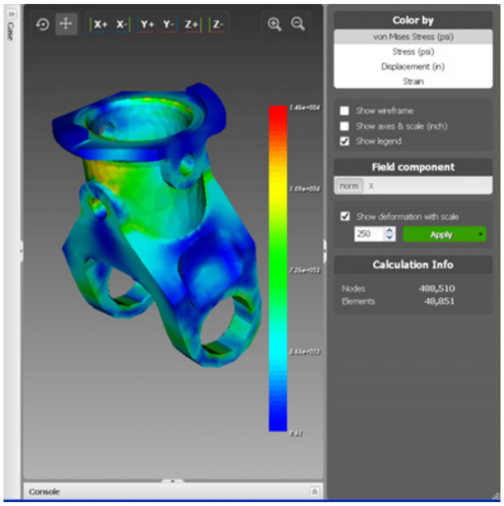
Running the simulation problem

To run the simulation problem we simply select AnalysisCalculation-->Run Calculation or the hammer icon next to calculation. The hammer icon will change to a spinning wheel and a messege will be displayed in the message console "Calculating…." Once the calculation is complete the screen will refresh with the Von Mises Equivalent Stress field displayed. Turning on the legend shows that the maximum Von Mises Equivalent Stress is 1.08e4 or 10,800 PSI and is around the constrained pin locations as expected. . It is also clear that the peak stresses are highly localized around the closest pin hole to the load and that changing the pin hole size and local geometry would have the biggest impact on the structural performance for this loading condition.

Visualizing the results

To evaluate the performance of our design we will need to visualize and understand the results. The default display shows the contours of the Von Mises Equivalent Streses in an undeformed shape but we can also select Show Deformation with scale and apply with or without ghost of underformed at actual scale or an exaggerated scale to understand the behavior.





The next step is getting an understanding of Displacements under load. We can select Displacement field and a component. The default component for displacement is called norm and refers to the magnitude of the Displacement. The image on the right illustrates the Displacement magnitudes under this load. The Displacement magnitude is calculated as .00346 inches where the pressure is applied.