Roll Cage Deformation COMSOL Simulation: Modeling Strain Under Different Loads with Nonlinear Geometry

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard.**

MODEL WIZARD

1. In the **Model Wizard** window, click 3D **3D.**
2. In the **Select Physics** tree, select **Structural Mechanics > Solid Mechanics.**
3. Click **Add.**
4. Click **Study**
5. In the **Select Study** tree, select **General Studies > Stationary.**
6. Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. In the table, enter the following settings:

Parameters

Name	Expression	Value	Description
F_Crash	m*delta_v/delta_t	28388 N	Force of Crash Impact
m	700[lb]	317.51 kg	Snowmobile Mass
delta_v	20[mph]	8.9408 m/s	Change in Velocity
delta_t	.1[s]	0.1 s	Time of Impact

GEOMETRY 1

1. In the **Model Builder** window, under **Component 1 (comp 1)** click **Geometry 1**.
2. In the **Settings** window for **Geometry**, locate the **Units** section.
3. From the **Length unit** list, choose **m**.

Import 1 (imp1)

1. In the **Geometry** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Import** section.
3. From the **Source** dropdown menu, choose 3D CAD file.
4. Select **Browse** and double click on the file labeled Rollcage_Chassis_forSimulation.x_t
5. In the **Import** toolbar, click **Import**.
6. In the **Graphics** window, click **Zoom Extents**.

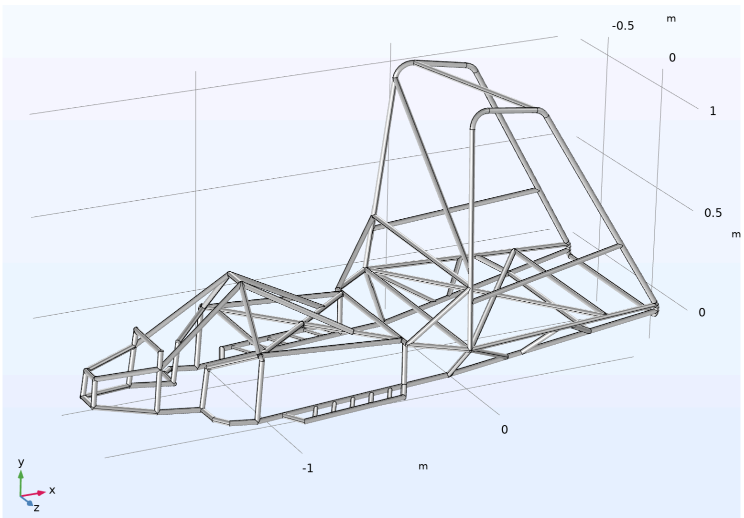


Figure 6: Imported Geometry

MATERIALS

Structural Steel

1. In the **Model Builder** window under **Component 1 (comp 1)**, right-click **Materials** and choose **Add Material from Library**.
2. In the **Add Material** window, search Steel. Select the Built-in material **Structural Steel** by double clicking. Close the **Add Material** window.
3. You will see that the **Material Contents** section has a the table with the following settings:

	Property	Variable	Value	Unit	Property group
☑	Density	rho	7850[kg/m^...	kg/m³	Basic
☑	Young's modulus	E	2e11	Pa	Young's modulus and Poisson's r...
☑	Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's r...
	Isotropic structural loss factor	eta_s	0.02	1	Basic
	Relative permeability	mur_iso ;...	1	1	Basic
	Heat capacity at constant pressure	Cp	475[J/(kg*K)]	J/(kg·K)	Basic

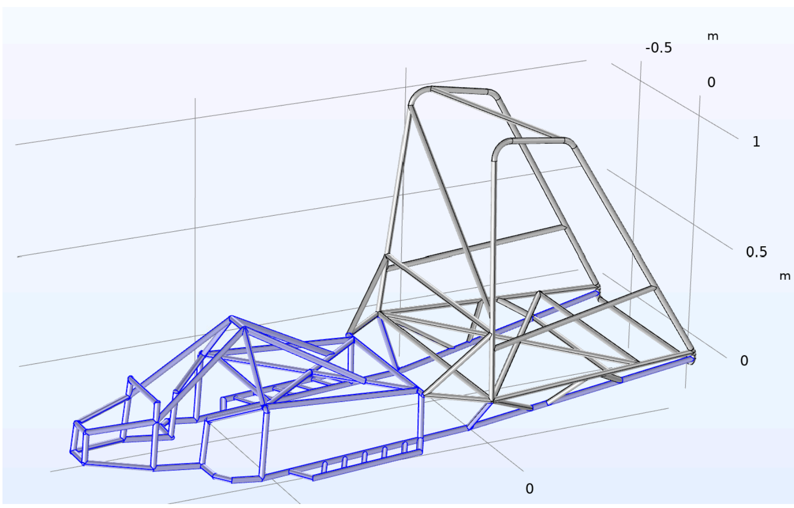
4. You will need to manually add the following values for E, rho, and nu:
    - rho = 7850[kg/m^3] ; E = 2e11[Pa]; nu = 0.3
5. Ensure that under **Geometric Entry Domain**, Domain 1 is selected.

**To represent different crash loading scenarios, we will change the boundary conditions but keep the rest of the study consistent. We will model three different crash scenarios (top-down full rollover, frontal rollover, and sideways rollover), which represent distinct impact conditions that test the roll cage under different loading directions. While the top-down case may be less realistic, it serves as a useful baseline for comparison due to its symmetry and severity.*

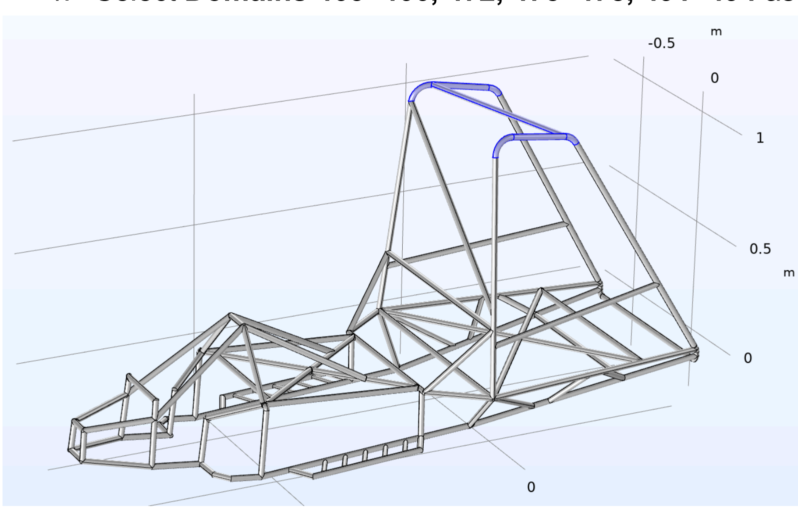
SOLID MECHANICS

Boundaries (Full Roll Over - Top Down Force)

1. In the **Physics** toolbar, select **Boundaries > Fixed Constraint**.
2. Select Domains as pictured below:



3. In the **Physics** toolbar, select **Boundaries > Boundary Load**.
4. Select **Domains 463–466, 472, 475–478, 491–494** as pictured below:

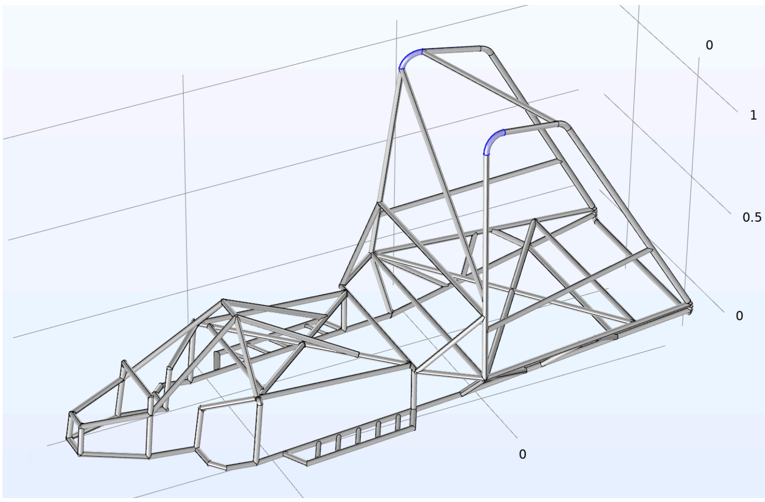


5. In the **Settings** window for **Boundary Load 1**, locate the **Force** section.
6. From the Load Type dropdown menu, select **Total force.**

7. In the Force box for the y dimension, enter -F_Crash.
8. The selections for Free Boundaries will update automatically.

Boundaries (Frontal Roll Over)

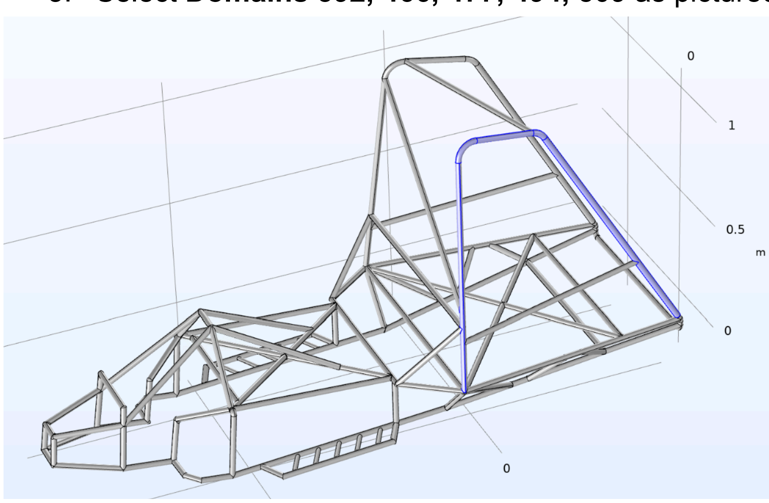
1. The **Fixed Constraint** is the same as in the first crash scenario
2. In the **Physics** toolbar, select **Boundaries > Boundary Load**.
3. Select **Domains 463–466** as pictured below:



4. In the **Settings** window for **Boundary Load 1**, locate the **Force** section.
5. From the Load Type dropdown menu, select **Total force.**
6. In the Force box for the x dimension, enter F_Crash/2.
7. In the Force box for the y dimension, enter -F_Crash/2.
8. The selections for Free Boundaries will update automatically.

Boundaries (Sideways Roll Over)

1. The **Fixed Constraint** is the same as in the first crash scenario
2. In the **Physics** toolbar, select **Boundaries > Boundary Load**.
3. Select **Domains 392, 466, 477, 494, 509** as pictured below:

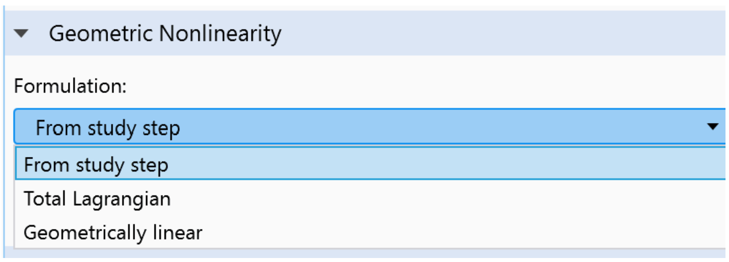


4. In the **Settings** window for **Boundary Load 1**, locate the **Force** section.
5. From the Load Type dropdown menu, select **Total force.**

6. In the Force box for the y dimension, enter -F_Crash/3.
7. In the Force box for the z dimension, enter -2*F_Crash/3.
8. The selections for Free Boundaries will update automatically.

Linear Elastic Material 1

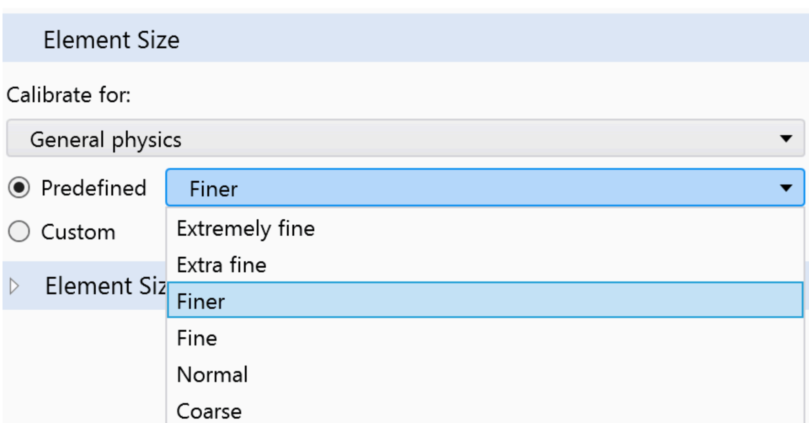
1. In the **Model Builder** window, select **Linear Elastic Material 1**
2. In the **Settings** window for **Linear Elastic Material**, locate the **Geometric Nonlinearity** section.
3. From the Formulation dropdown menu, select **From study step**. We will manually select “Include geometric nonlinearity” in the study settings.



Mesh

H-Refinement

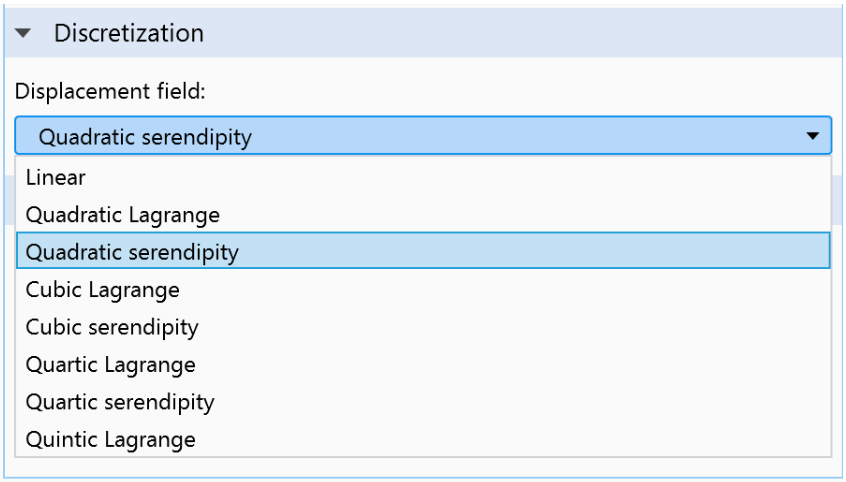
1. Select **Size** under the Mesh 1 dropdown and change the **Element Size** in the dropdown box to **Normal**. In the **Settings** section for Size, select **Build All**. This won’t work because the mesh is too large and causes geometric issues.
    - **Normal mesh failed due to small features**: Faces and edges were too narrow for the minimum element size, causing mesh generation errors.



2. Try again, this time selecting **Fine** from the mesh dropdown menu. The same thing will happen as the mesh is still too large.
3. Finally, try building the mesh using the **Finer** option from the menu. This time, the mesh will build without issue. We will use this mesh size for the simulation.

P-Refinement

1. In the **Model Builder** window, click on Solid Mechanics (solid) and expand the Displacement field dropdown menu under **Discretization** to see the options for the shape function polynomial order.



2. You will need to create a new Study for each P-value run and compare the resulting stress outputs. We will use the first Boundary setup, "*Sideways Roll Over*". Make sure to only use the original Parameter values from the table, otherwise the study will take unnecessarily long to compute. We will perform a parameter sweep later.

*Note: *Lagrange* elements include a full set of shape functions with an internal node for higher accuracy, while *Serendipity* elements omit the internal node for reduced computational cost at slightly lower accuracy.

Study 1 (Linear Shape Functions)

1. In the Model Builder window, under Study 1 click Step 1: Stationary.
2. In the Settings window for Stationary, click to expand the Study Settings section.
3. Click check box for "**Include geometric nonlinearity**"
4. In the **Study** toolbar, select **Show Default Solver**
5. In the Model Builder window, under Study 2 click Solver Configurations  .
6. In the settings window for Stationary Solver 1, under General, select Nonlinear from the Linearity dropdown menu
7. Make sure the Discretization dropdown is set to ***Linear***
8. Make sure to apply the boundary conditions for *Sideways Roll Over* and **Compute**.

Study 2 (Quadratic Serendipity Shape Functions)

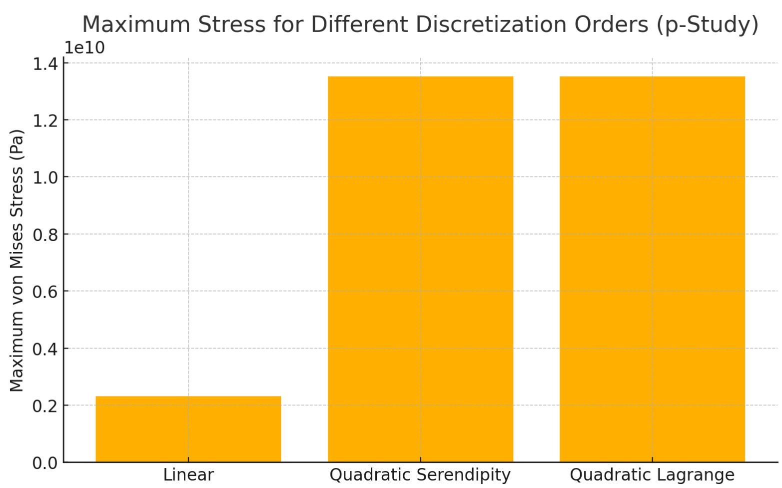
1. In the **Study** toolbar, select **Add Study > Stationary**.
2. In the Model Builder window, under Study 2 click Step 1: Stationary.
3. All study setting should be identical to *Study 1*
4. Make sure the Discretization dropdown is set to ***Quadratic Serendipity***
5. Make sure to apply the boundary conditions for *Sideways Roll Over* and **Compute**.

Study 3 (Quadratic LaGrange Shape Functions)

1. In the **Study** toolbar, select **Add Study > Stationary**.
2. In the Model Builder window, under Study 2 click Step 1: Stationary.
3. All study setting should be identical to *Study 1*
4. Make sure the Discretization dropdown is set to ***Quadratic LaGrange***
5. Make sure to apply the boundary conditions for *Sideways Roll Over* and **Compute**.

Visualizing P-Refinement

1. Compute each study separately using the desired discretization type (Linear, Quadratic Serendipity, or Quadratic Lagrange).
2. In the Model Builder window under the **Results** node, right-click **Derived Values** and choose: **Maximum > Volume Maximum**
3. In the **Settings** panel for the volume maximum, change the dataset to match the corresponding study.
4. In the Expression field, type: **solid.mises**
5. Click **Evaluate** to calculate the maximum stress value, which appears in a new table.
6. Create a new volume maximum node for each study to separately extract and record the stress values. Here I have plotted the values outside of COMSOL for visual reference:



7. As you can see, both quadratic studies produce the same result, but the linear study was highly inaccurate. We will use Quadratic Serendipity since it is faster to compute.

**We will use a parametric sweep from 10 to 50 mph in 10 mph increments to evaluate deformation under increasing crash loads and to reveal the solver's limitations in capturing post-buckling behavior, which despite accounting for geometric nonlinearity, remains challenging due to rapid stiffness loss and unstable equilibrium paths that standard solvers like Newton-Raphson cannot reliably follow.*

Parametric Sweep Studies for Increasing Crash Intensity

Study 1 (Full Roll Over - Top Down Stress)

1. In the **Study** toolbar, select **Add Study > Stationary**.
2. In the Model Builder window, under Study 1 click Step 1: Stationary.
3. In the Settings window for Stationary, click to expand the Study Settings section.
4. Click check box for "**Include geometric nonlinearity**"
5. In the **Study** toolbar, select **Show Default Solver**
6. In the Model Builder window, under Study 2 click Solver Configurations .
7. In the settings window for Stationary Solver 1, under General, select Nonlinear from the Linearity dropdown menu
8. Right-click *Study 1* and choose **Parametric Sweep**.
9. In the **Settings** window for **Parametric Sweep** click **Add** and select delta_v from the list of parameters.
10. Under *Values of Dependent Parameter*, enter: "range(10[mph], 10[mph], 50[mph])"
11. Make sure to apply the boundary conditions for *Full Roll Over* and select **Compute**.

*Note: You will notice that after running the study, the solver does not converge for the last parameter step in the sweep. This is because at this force value, the post-buckling behavior dominates the study and the results become unreliable. For this reason, we will change the sweep to sweep delta_v from 10 to 40 mph in 10 mph steps.

Study 2 (Frontal Roll Over)

1. In the **Study** toolbar, select **Add Study > Stationary**.
2. In the Model Builder window, under Study 2 click Step 1: Stationary.
3. In the Settings window for Stationary, click to expand the Study Settings section.
4. Click check box for "**Include geometric nonlinearity**"
5. In the **Study** toolbar, select **Show Default Solver**
6. In the Model Builder window, under Study 2 click Solver Configurations .
7. In the settings window for Stationary Solver 1, under General, select Nonlinear from the Linearity dropdown menu
8. Copy and Paste the **Parametric Sweep** from *Study 1* into *Study 2*.
9. Under *Values of Dependent Parameter*, enter the range:
    a. range(10[mph], 10[mph], 40[mph])
10. Make sure to apply the boundary conditions for *Frontal Impact* and select **Compute**.

Study 3 (Sideways Roll Over)

1. In the **Study** toolbar, select **Add Study > Stationary**.
2. In the Model Builder window, under Study 2 click Step 1: Stationary.
3. In the Settings window for Stationary, click to expand the Study Settings section.
4. Click check box for "**Include geometric nonlinearity**"
5. In the **Study** toolbar, select **Show Default Solver**
6. In the Model Builder window, under Study 2 click Solver Configurations .

7. In the settings window for Stationary Solver 1, under General, select Nonlinear from the Linearity dropdown menu
8. Copy and Paste the **Parametric Sweep** from *Study 2* into *Study 3*.
9. Make sure to apply the boundary conditions for *Side Impact* and select **Compute**.

*After running the parameter sweep deformation simulation for the sideways rollover scenario, the roll cage was observed to deform significantly into the rear passenger head space. To evaluate a potential improvement, we will import an updated CAD model that includes an additional support bracket designed to resist deformation in this direction.

Study 4 (Sideways Roll Over - Updated Geometry)

1. Repeat the steps for importing geometry, but use the file “Rollcage_Chassis_forSimulation_Modified.x_t”
2. Apply the same fixed boundary constraints as with the other model.
3. In the **Study** toolbar, select **Add Study > Stationary**.
4. In the Model Builder window, under Study 2 click Step 1: Stationary.
5. In the Settings window for Stationary, click to expand the Study Settings section.
6. Click check box for “**Include geometric nonlinearity**”
7. In the **Study** toolbar, select **Show Default Solver**
8. In the Model Builder window, under Study 2 click Solver Configurations .
9. In the settings window for Stationary Solver 1, under General, select Nonlinear from the Linearity dropdown menu
10. Copy and Paste the **Parametric Sweep** from *Study 2* into *Study 3*.
11. Make sure to apply the boundary conditions for *Side Impact* and select **Compute**.

Results

Since we ran the studies using a parametric sweep for delta_v, the results will be stored as a group of outputs for each study (one for each parameter in the sweep). We can use this to create a GIF for each crash scenario that progresses through the resulting deformation for each impact force. This will give us a good idea of what a worst case scenario crash might look like.

Crash Deformation GIFs

1. In the Model Builder window, expand the corresponding Study under **Results**.
2. Expand **Volume 1** and select **Deformation**
3. In the Settings window for Deformation, click to expand the Scale Settings section.
4. Make sure the Scaling is set to 1 so it shows the realistic deformation
5. Select **Plot**
6. Rotate and zoom your 3D model to the orientation you want.
7. Go to the **Results** tab at the top ribbon and click **Animation > Player Settings**
8. In the Settings pane, under **Animation Settings**, change **Output** to *file*.
9. Choose **GIF** as the *File Type*.
10. Specify the output **Filename** and **location**.
11. In the *Animation* settings, ensure the **Dataset** corresponds to the correct study.
12. Click **Export**