

## Suspension Modeling

### Introduction:

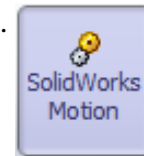
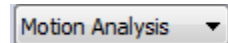
The purpose of this Case Study is to familiarize oneself with the basic concepts of SolidWorks Motions study abilities and analyze a simple spring suspension for a RC car. By the end of this lesson you should know how to apply forces, springs, and calculate the results.

1. Open SolidWorks Assembly Suspension Study.

Go to the Office Product Tab and click SolidWorks Motion Button.


Open Motion Study 1.

Select Motion Analysis in the drop down box.

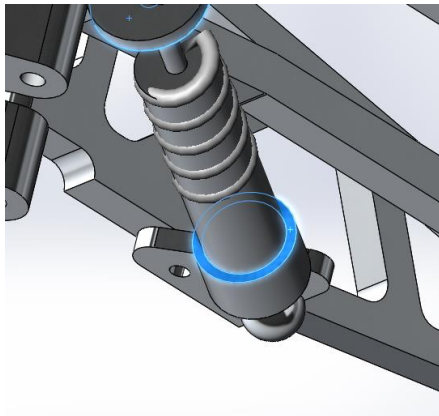
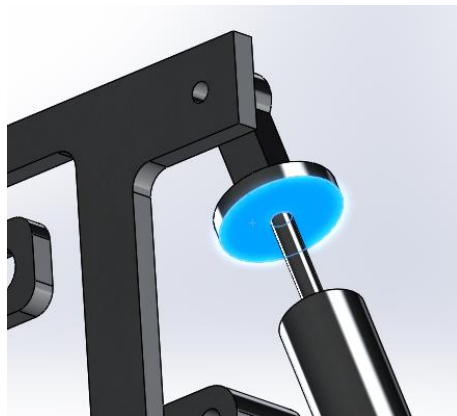


2. We would like to analyze the forces of the system for 10 seconds, so select the black diamond of the Suspension Study in the Motion Tab and drag it to 10 seconds.

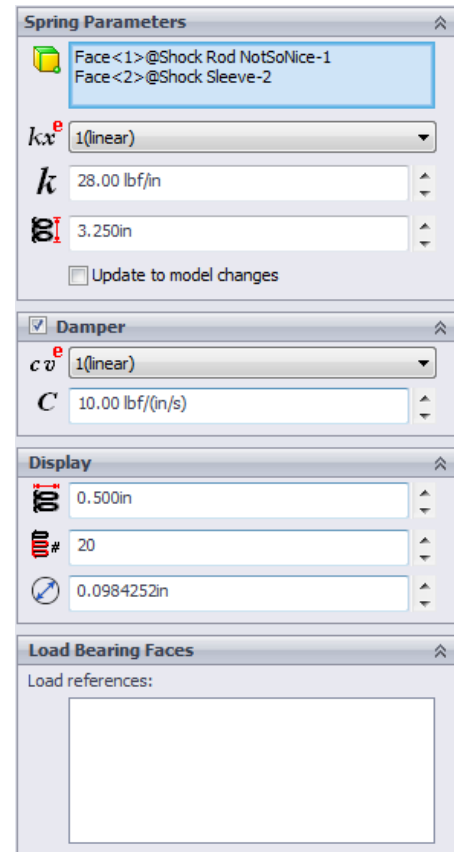
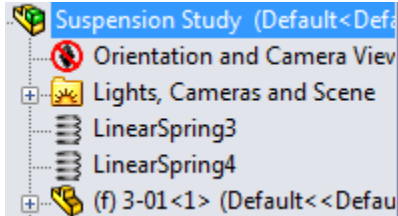


3. Now that the time has been determined, we would like to add two springs to the system to model the shocks. The springs will have both damping and spring forces, to accurately model actual shocks. This can be done with the **spring** button. 


4. For the Spring Endpoints select the inner face of the upper disk of the shock and the lower edge of the shock. They should look something like the images below.

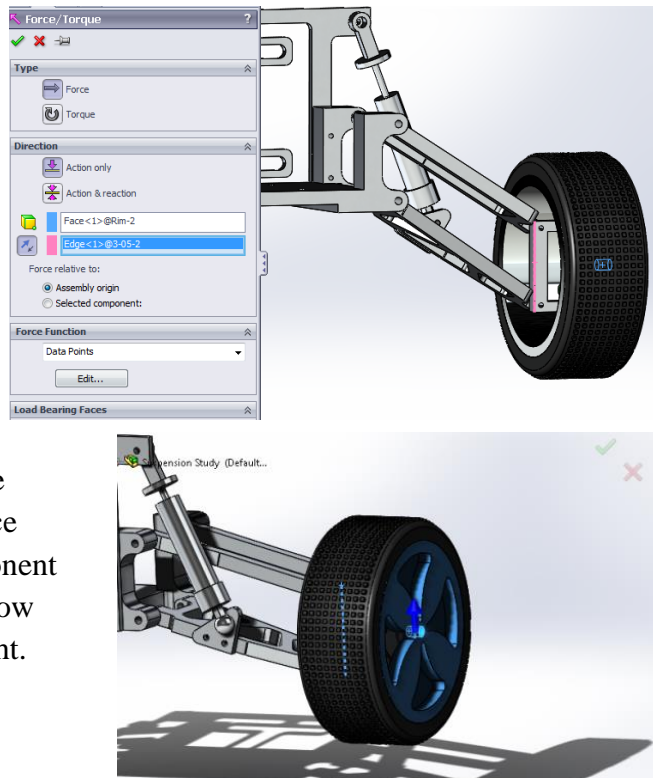


5. Input the spring and damper parameters as shown to the right. These values were from experimentation with the car and from SolidWorks.
6. Do this for both sides of the suspension, so that they are mirrored. Your Motion Tree should look something like the image below.

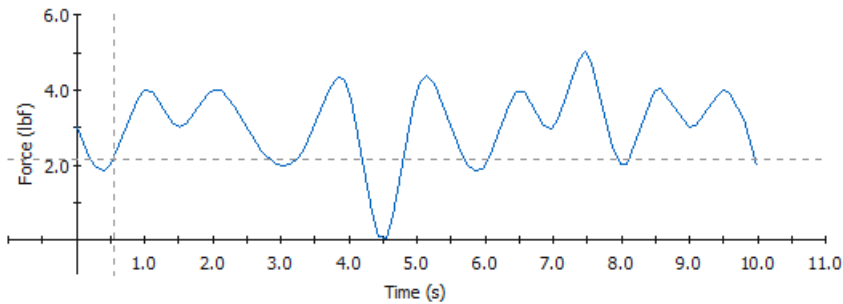



7. Next, add the forces acting on the system. The weight of the car is split between the four tires of the vehicle. Since the car weighs 8lbs, a minimum of 2lbs of contact force for the car is required at each wheel. Since the Car will be moving and contacting different obstacles in the 10 seconds, a constant 2lbs is not sufficient in modeling the suspension. A force analysis of the 10 sec has been predetermined within an excel sheet for you.



8. Select the **force** button option.  Select the inner circular face of the rim as the force action point. The force is transmitted from the outer tire to this point, so it is safe to model a contact force here, even without taking the moment into account. Select an edge or line in the vertical direction to determine the force direction. If the force is negative switch it to positive with the reverse direction button. For the Force relative to: Select the selected component option and select the rim. Your window should look similar to the images right.

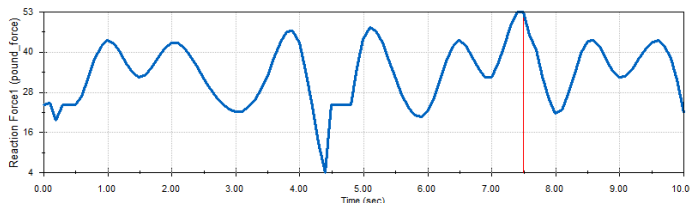


9. Now select the data points option under force function section. Select Import Data... Import the excel file. (When importing excel files, save excel files as \*.csv for conversion to be compatible with solidworks.) The imported data should be like in the figure below. Do the same for both wheels.

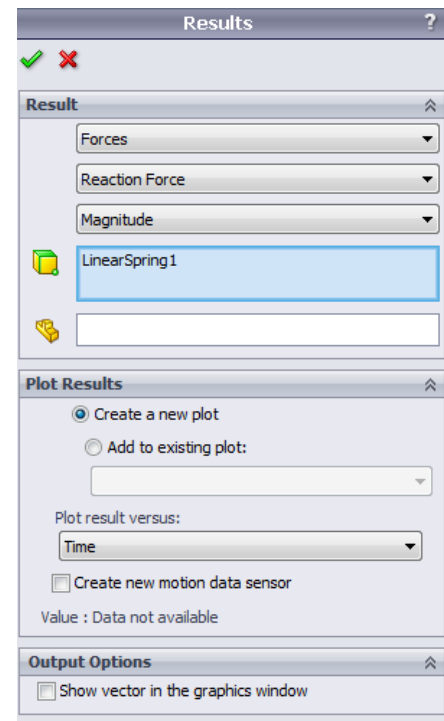


10. After both of the springs and forces have been determined for the simple suspension analysis, hit the **calculate** button  to simulate the motion and reactions.

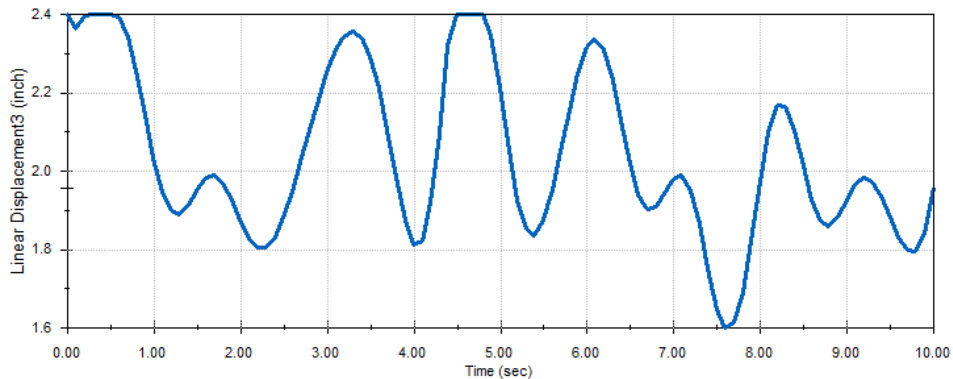
11. Once the motion had been calculated the simulation can be ran a multiple times with the play button . Then we can calculate the wanted results of the spring force over the period and the spring displacement over the period. Click the Results and Plots button . Now we would like to calculate the reaction force of the spring over the time period, so follow the image to the right. The plot should look something like the lower image.



The force is the overall reaction force due to the vertical force acting on the tire and the moment produced on the A-arms due to this force.



12. Follow the same procedure to find the spring displacement over the time. The graph should look like this.



The displacement output for Solidworks is the distance between the two endpoints of the spring. To find the spring's actual displacement due to the forces, we take its original length 3.25" and subtract the linear displacement from the SolidWorks Plot.

13. Let's analyze what damping does to the system by modifying our current simulation. Go to your two springs, right click, hit edit feature. Change the Damper C to 100 lbf/(in/s). Now calculate. What happens to the system from this change in damping?
14. Now return the Damping back to 10 lbf/(in/s), and reduce the spring k to 15 lbf/in. What happens to the system due to this change in spring constant k. What happens to Solidworks ability to solve this analysis? Why might this be? Now increase the spring k to 35 lbf/in. How does the system react to this? Is this a good representation of how an actual spring mass damper acts in real motion?