# POST RIG SOFTWARE TUTORIAL

**Aditya Mhatre**

**17033712@brookes.ac.uk**

# Table of Contents

# List of Figures

# Introduction

The Post Rig is a software which help the user study how a One Degree of Freedom Spring Mass Damper System responds to different input conditions. This document is a step by step guide for any new user to start using the software.

# 1. Starting the Software

In order to start the software first you need you install it. This topic explains how to install the software. The first step in doing so is to download the PostRig Zip Folder from the link in the email.

## 1.1. Download and Extract

Once downloaded, the zip folder will can be accessed in the Download folder in you Computer. The PostRig folder needs to be extracted which can be done by clicking the Extract All button on the top which will open and Extraction Wizard asking for the location as seen in Figure 1. Extract the folder to the desired location and click Extract.

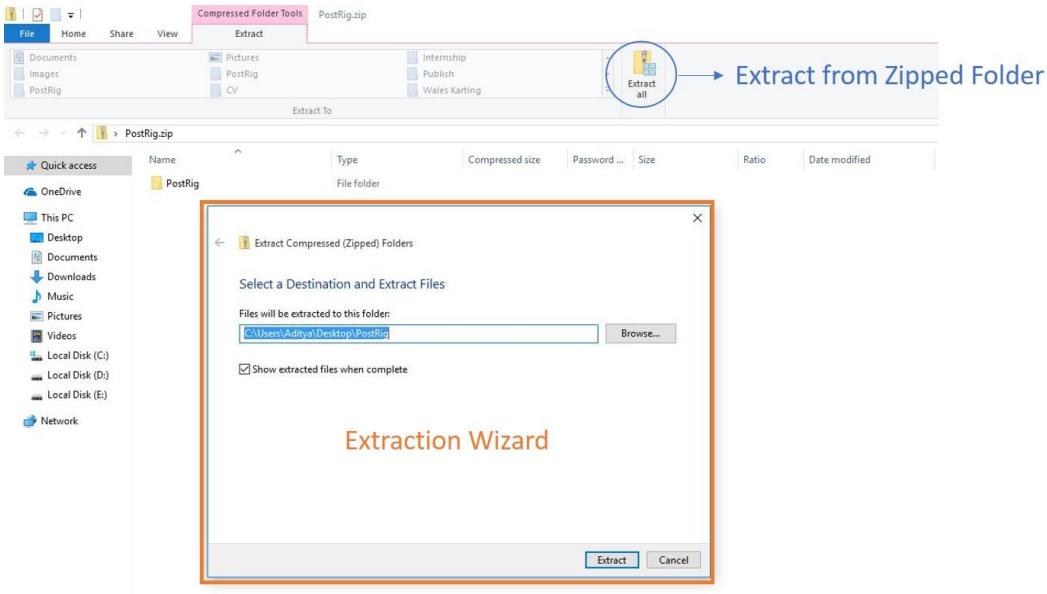


*Figure 1: Extract Folder*

## 1.2. Installation

Once Extracted, please go the location where the folder is extracted and open it. This folder will contain the Setup File. Run the setup.exe, which will open the Application Installer (Figure 2). Click on install and the software will open once the download is complete. The software can also be found in the Start Up Menu (Figure 3) and also on the desktop for easy access. This will open the software home page (Figure 5).

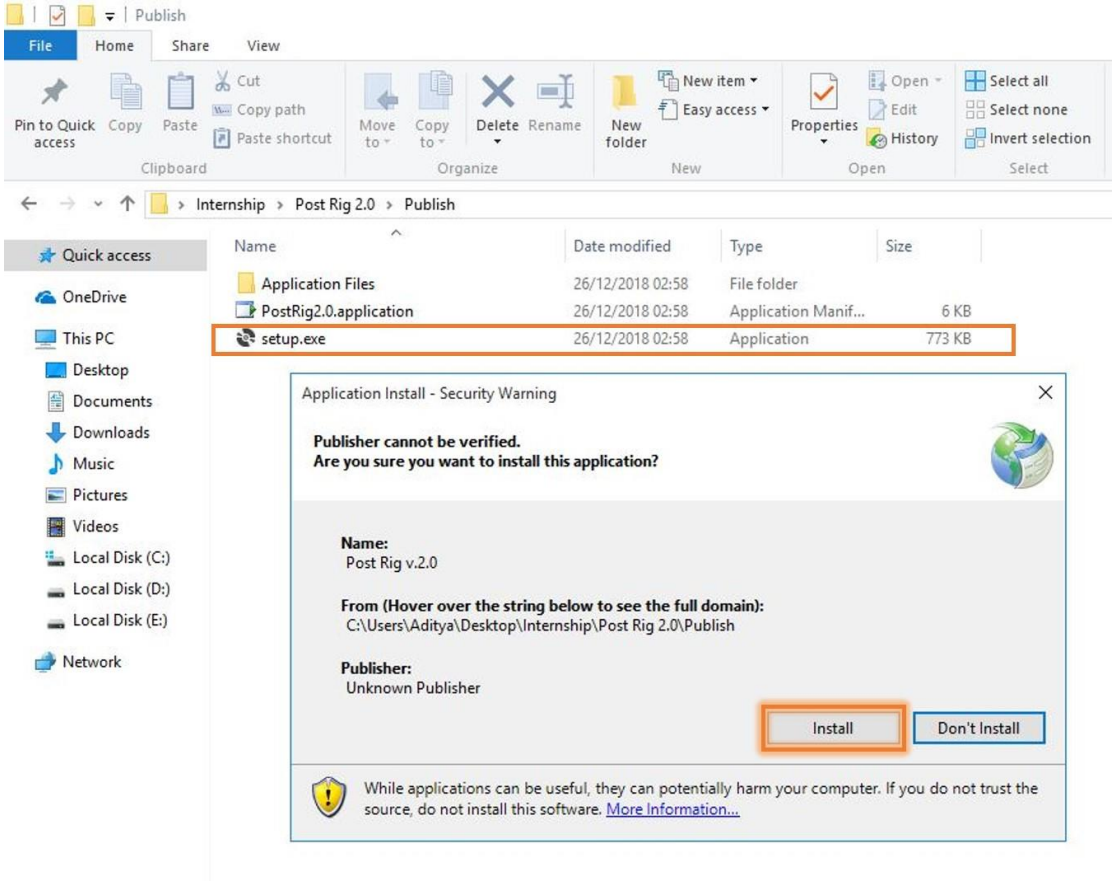


*Figure 2: PostRig Application Installer*

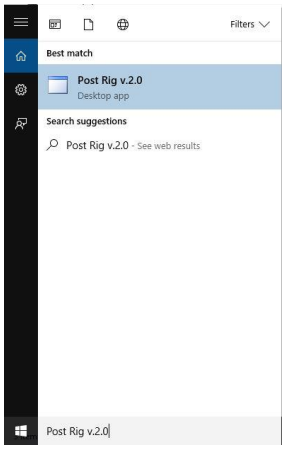


*Figure 3: Start Up Menu*

## 1.3. Uninstalling

The software can be uninstalled from the Apps & Features in the setting of your computer. Uninstall it from there by searching for Post Rig and clicking uninstall (Figure 4.)

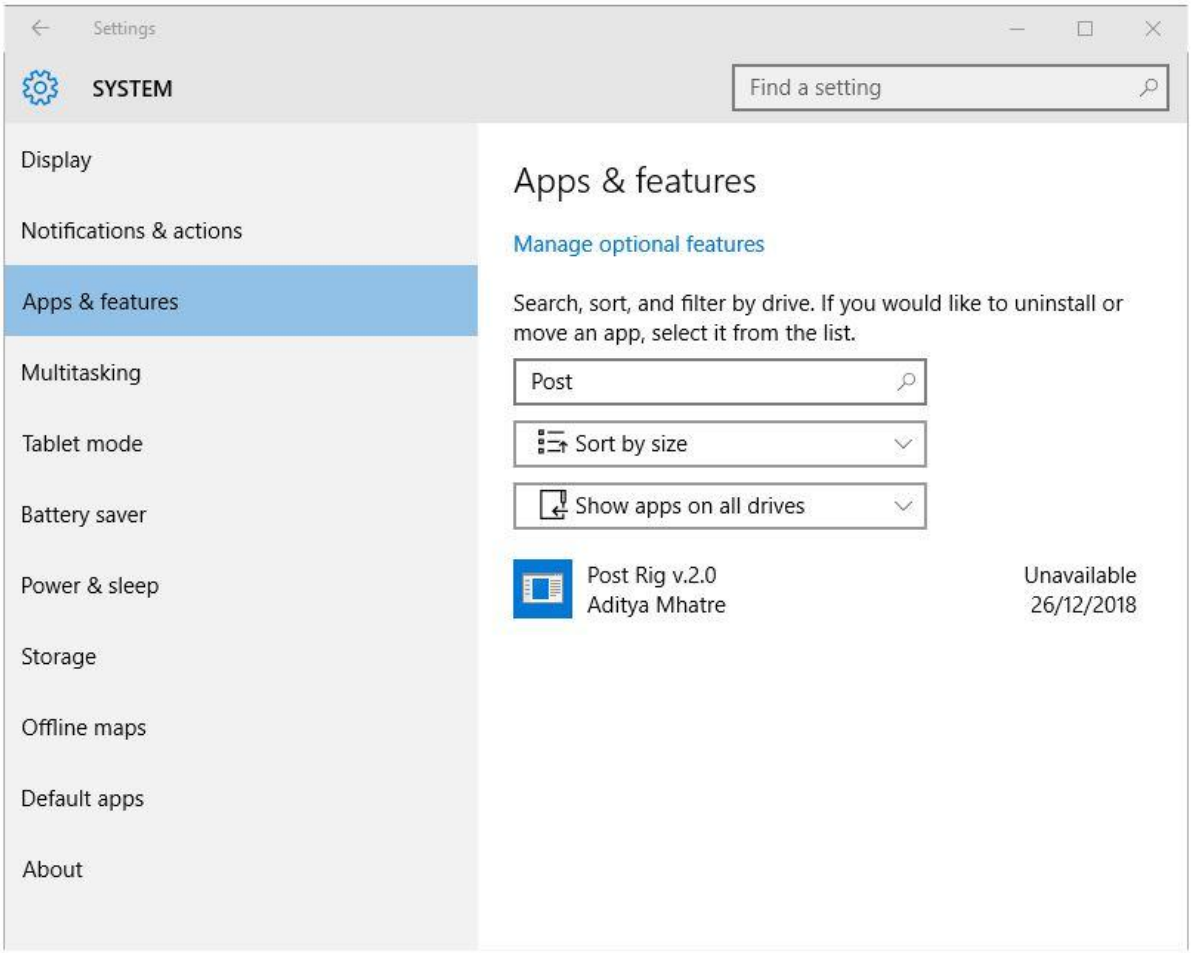


*Figure 4: Uninstall*

# 2. Software Start Up

The software will open as soon as the installation is complete. The software can also be opened by the icon created on the desktop or in the start-up menu. Once the software is opened it will display the home page as shown in Figure 5.

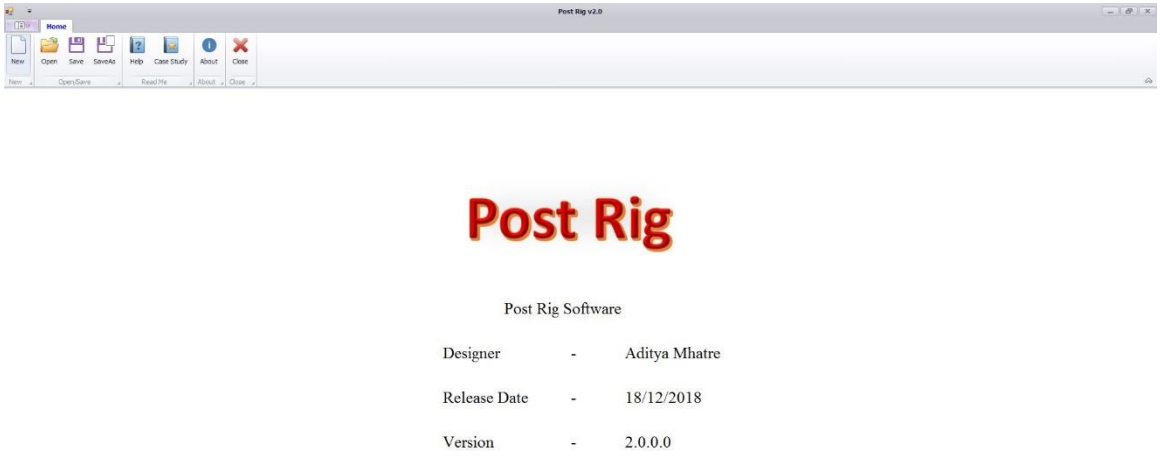


*Figure 5: Home Page*

The options available in the Home Tab are shown in Figure 6. This contain options such New, Open, Save, Save As, Help, Case Study, About and Close. As brief descriptions and functionalities of the above options are mentioned below.

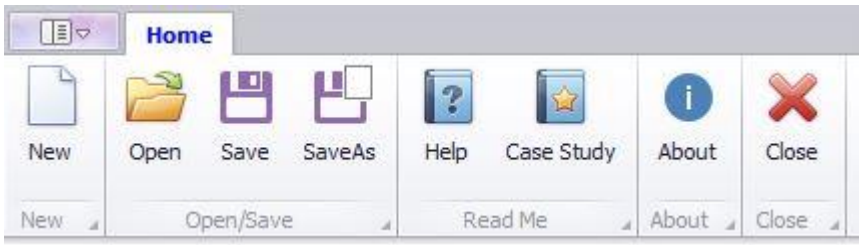


*Figure 6: Home Tab*

## 2.1. New

This is used to start a new project. On clicking this option, a new project is started and the Design, Simulation Setup and Results tabs are visible. These tabs as are only visible when a new project or an existing project is started.

## 2.2. Open

This is used to open an existing project. This software will only open Post Rig files (.postrig).

### 2.3. Save

If a project has not been saved earlier, this will save a new prostrig file(.postrig) file which can be saved with any name desired, but if a project has already been saved this will save an updated version.

### 2.4. SaveAs

This allows the user to save a new postrig file.

### 2.5. Help

This opens this help file in the software, which explains the workings of the software. Note: By holding Control key and using the scroll wheel, the user can zoom in and out of the PDF.

### 2.6. Case Study

This contains a case study which show how the software can be used. Note: By holding Control key and using the scroll wheel, the user can zoom in and out of the PDF.

### 2.7. Close

This closes the software. Note: The software does not prompt to save if changes have been made. Please ensure the changes are saved before closing the software.

# 3. Design Tab

In the Design Tab the user can build a car modelled as single lumped mass on a moving platform supported with a spring and damper assembly. The input required for the software are corner mass, spring stiffness and damping coefficient.

In the Design Tab, 3 templates are available; Road Car, Race Car and Rally Car as shown in Figure 8. These are just different names used to define an Underdamped (Damping Ratio < 1), Critically Damped (Damping Ratio = 1.0) and Over Damped (Damping Ratio > 1.0) system (The values provided by these templates are assumed values and do not hold exact resemblance to any car). By selecting either of them, the corner weights, spring stiffness and damping coefficients are filled in automatically in their respective textboxes. For these templates, the corner weight is calculated by assuming a Front-Rear and Right-Left weight distribution of 50-50. The values for these templates are mentioned in Table 1.

| Template | Total Weight (kg) | Corner Weight (kg) | Spring Stiffness (N/m) | Damping Coefficient (N/(m/s)) | Damping Ratio |
|---|---|---|---|---|---|
| Road Car | 1600 | 400 | 80000 | 4000 | 0.35 |
| Race Car | 1600 | 400 | 120000 | 13850 | 1.00 |
| Rally Car | 1600 | 400 | 150000 | 17050 | 1.10 |

*Table 1: Template Data*

To begin, a new car must be built, which can be done by clicking the Build Car Icon, in the design tab (Figure 7). Once this is done, a template panel open on the left and a model representation on the right are displayed.


*Figure 7: Design Tab*

Apart from these templates, the user can build a custom car. This can be done by simply changing the respective field values to user defined values. Reverting back to the templates, can be done just by the clicking any of the template options.

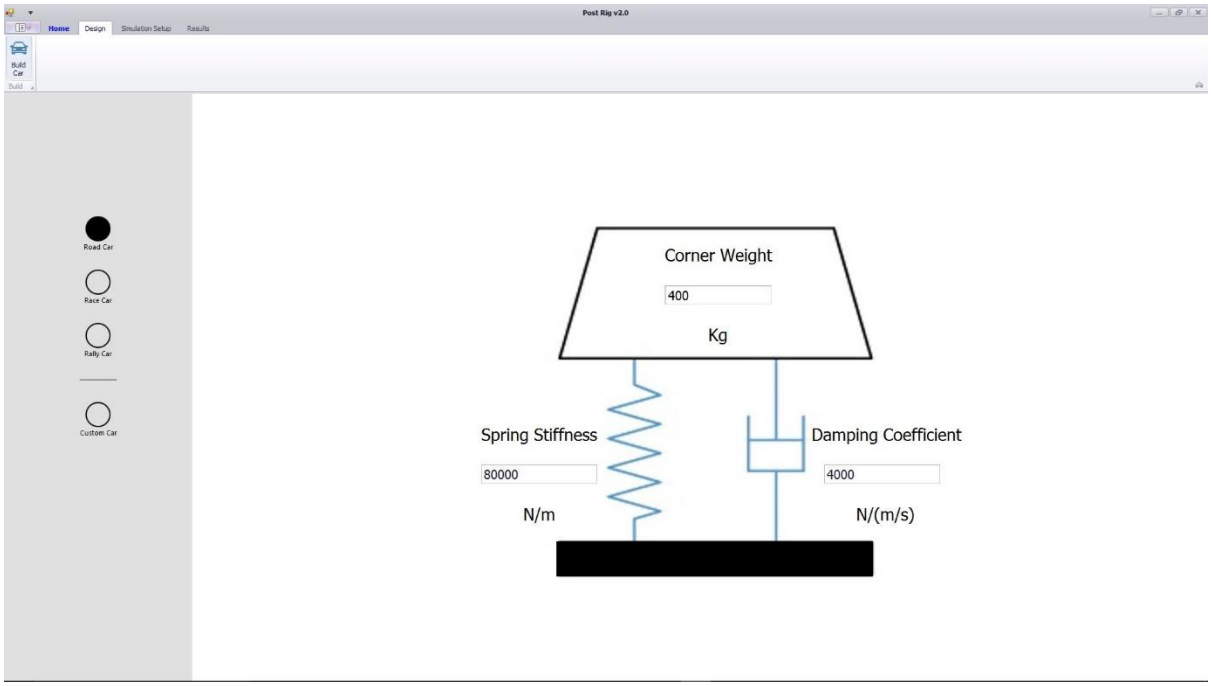


*Figure 8: Design Panel*

**Note:** If changes are made to the Vehicle Parameters and if any template is clicked, the values will be reset to corresponding values in Table 1.

# 3. Simulation Setup and Run:

Once all the values have been entering or a template has been selected, switch over to the Simulation Setup Tab next to the Design Tab.

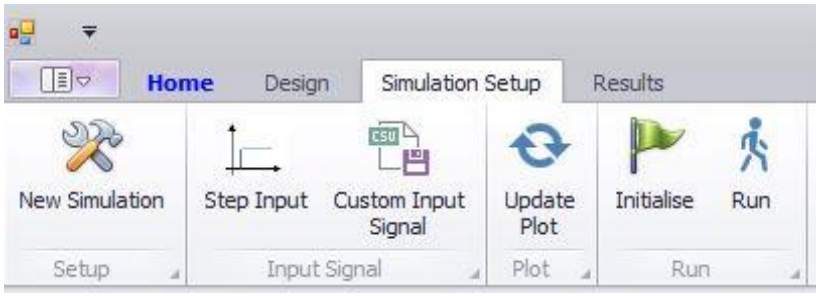


*Figure 9: Simulation Setup Tab*

To begin setting up a simulation, Select New Simulation. On doing so, the software will hide the Design Panel and open the Simulation Setup Panel. This will open a blank Simulation Setup Panel. Depending on the Input Signal Selected, corresponding data will be displayed. These are explained below.

## 3.1. Step Input

On selecting the Step Input, a data panel will appear on the left. The results that are calculate are based on a fixed time frame. By default, the time frame for the simulation start at zero and spans for five seconds with a time step of 0.01 seconds. The signal steps up to the defined amplitude at one second, this time is fixed and cannot be changed.

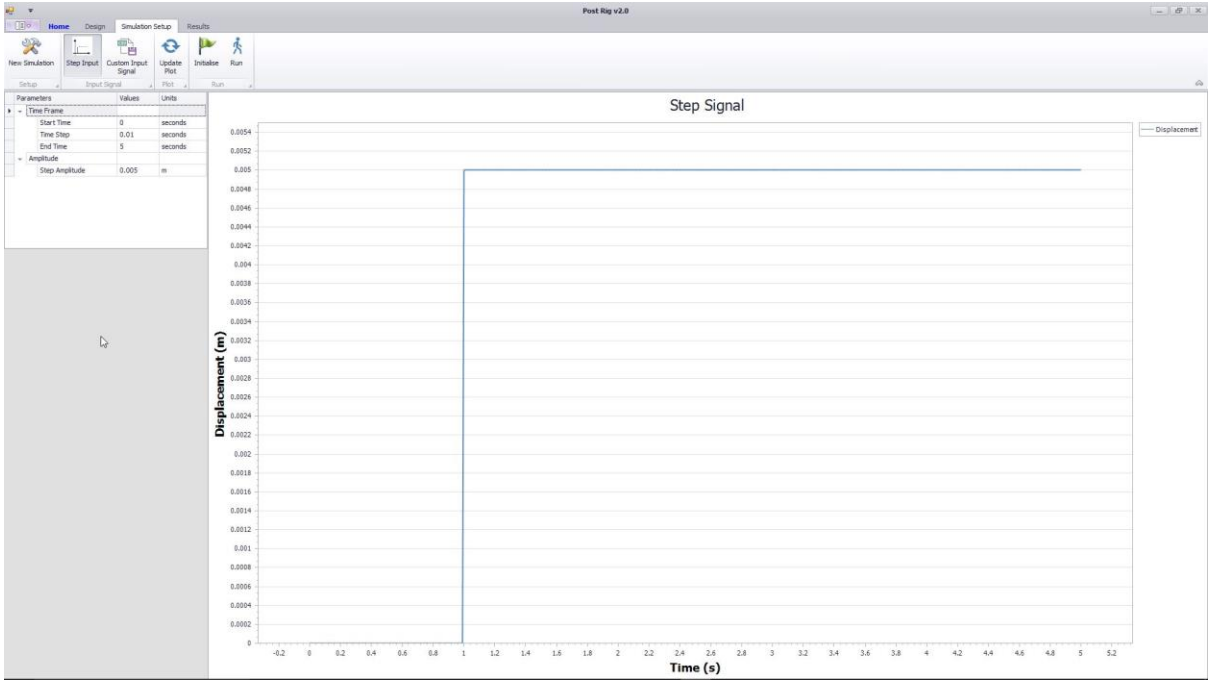


*Figure 10: Step Input Signal*

All other values like the Start Time, Time Step, End Time and Step Amplitude can be changed to representative values.

Once these values have been finalised, the simulation can be initialised by clicking the Initialise button in the Simulation Setup Tab. This initialises the simulation and displays the plot of the step input as shown in Figure 10.

## 3.2. Custom Input Signal

Using this option, the user can import a CSV file of the Input Signal. This signal must be a Time vs Vertical Displacement signal. The CSV file must contain only numerical values, the title for the columns must **NOT** be included. Once the CSV is imported, the simulation automatically Initialises the simulation and displays the plot of the signal (Figure 11).

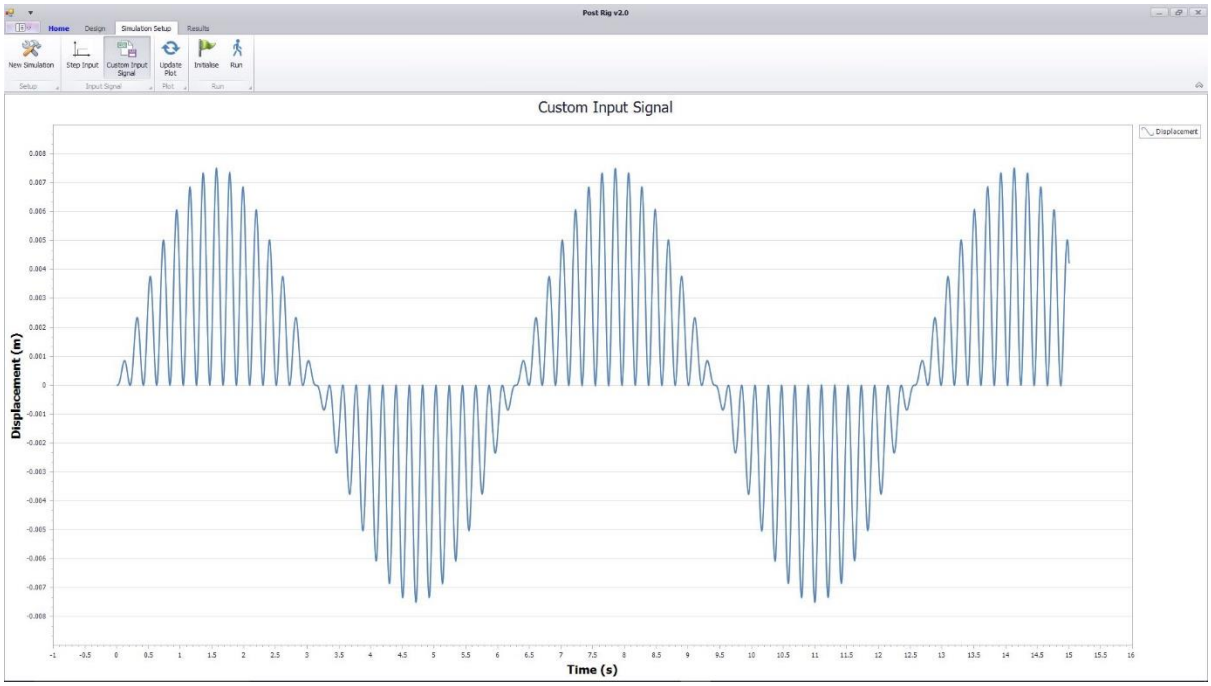


*Figure 11: Custom Input Signal.*

Once the simulation is Initialised, it is ready to run. This is done by clicking the Run button in the Simulation Setup Tab. A prompt will appear once the simulation has run successfully.

The Custom Input Signal plot can be zoomed and panned as the user pleases with the use of the mouse scroll wheel or the track pad on laptops.

**Note:** Any changes made to the simulation setup data such as, changing the design template or simulation setup data, the simulation must be initialised before running again.

# 4. Results

Once the simulation has successfully run, the results can be viewed in the software. The output option can be seen in the Results Tab.

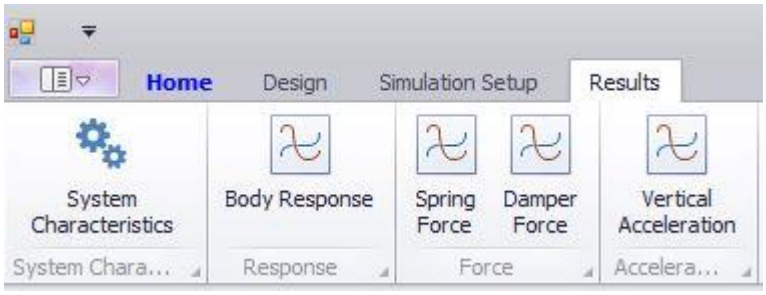


*Figure 12: Results Tab*

The output that the software is capable of producing are mentioned in Table 2.

| **Output Data** | |
|---|---|
| System Characteristics | Units |
| Natural Frequency | Hz |
| Critical Damping Coefficient | N/(m/s) |
| Damping Ratio | - |
| **Output Plots** | |
| Body Response vs Time | m |
| Spring Force vs Time | N |
| Damper Force vs Time | N |
| Vertical Acceleration vs Time | G |

*Table 2: Output*

The System Characteristics are displayed in the panel on the left. The plots are displayed in the plot area as show in fig.

The Body Response displays the response of the body to the give input conditions in pure vertical motion. Depending on the vertical body motion and the base motion, the Spring and Damper Forces are plotted. The vertical acceleration of the body can also be displayed. These plots can be displayed by selecting the respective plot option from the Results Tab.

The Result in the software are displayed as shown in Figure 13.

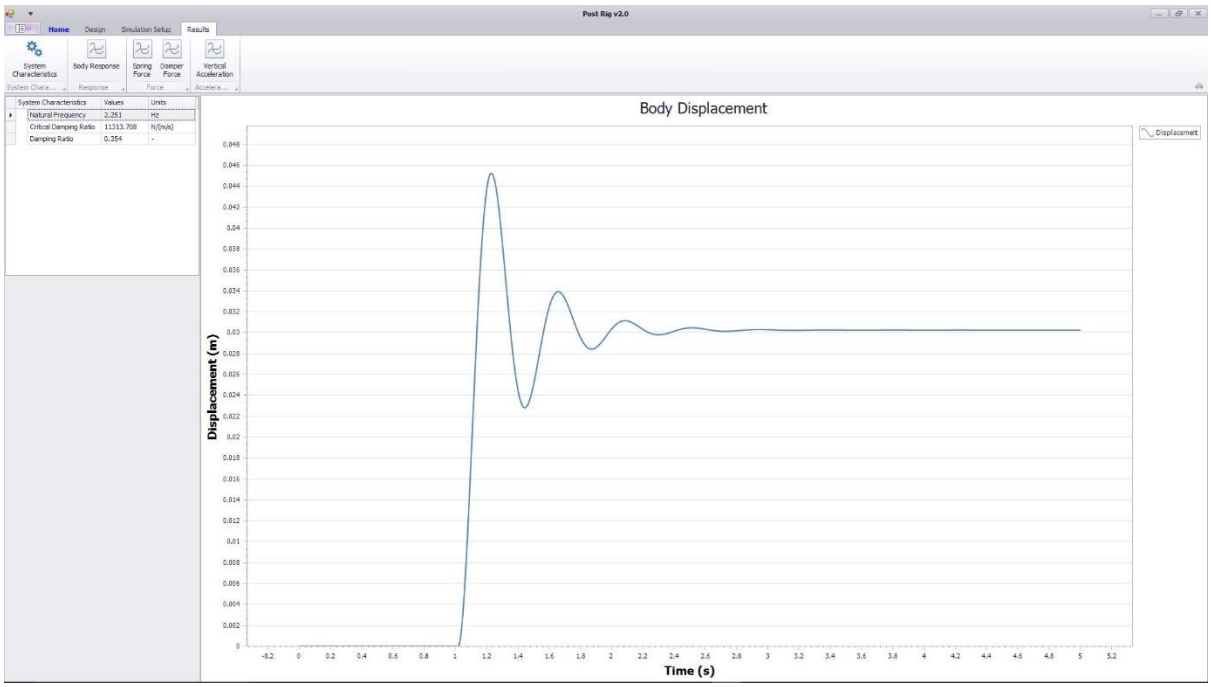


*Figure 13: Results Panel*

**Note:** Once a new simulation is run, the updated plot for the new results will be displayed only after clicking the respective output plot from the Results Tab.