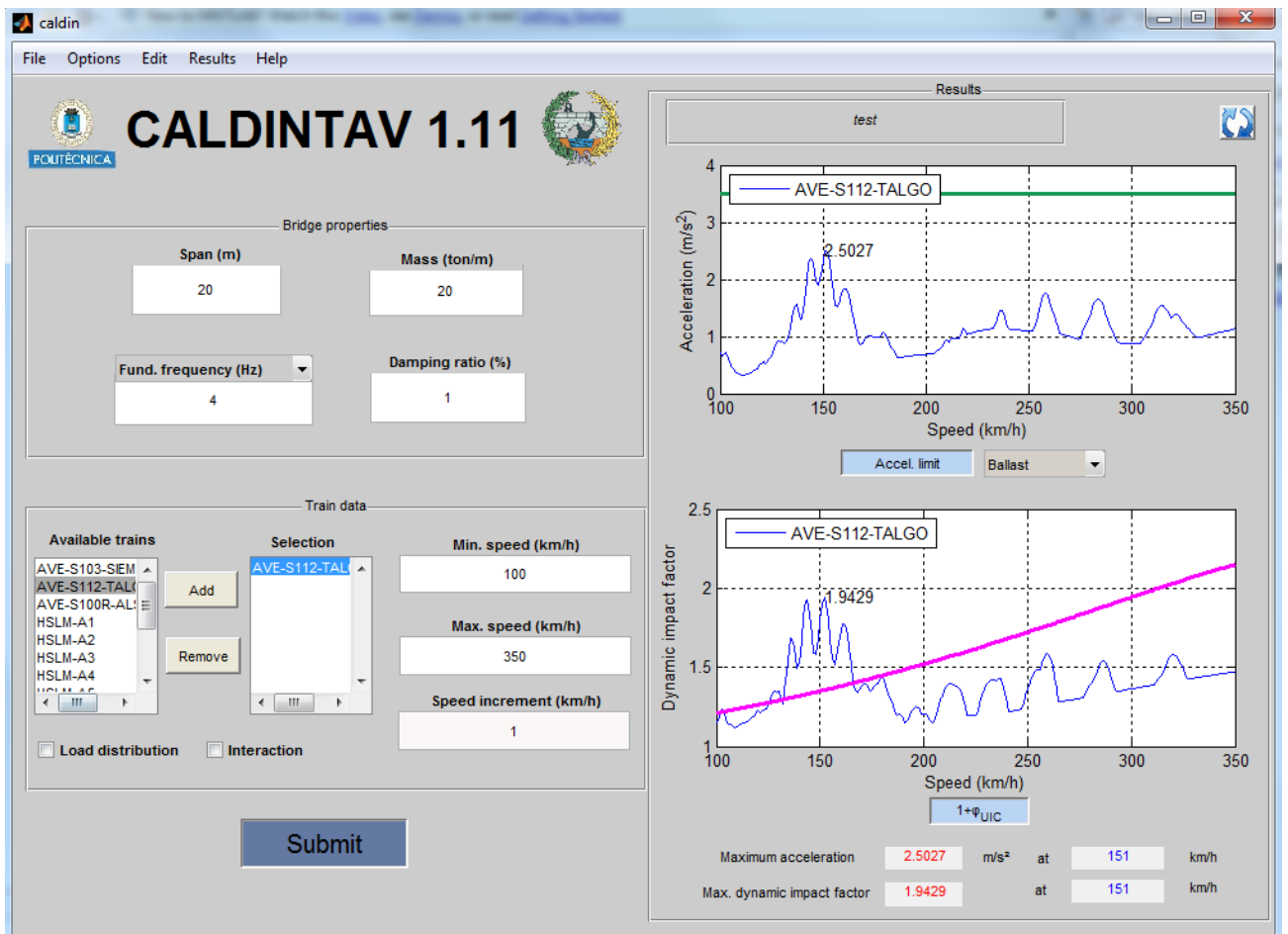


# CALDINTAV V1.11 : USER'S GUIDE

Madrid, November 7, 2017

Technical University of Madrid

Jose M. Goicolea, Pablo Antolín, Teresa Ancochea, J.Domínguez



# Contents

<b>1</b>	<b>Introduction</b>	<b>3</b>
1.1	Scope . . . . .	3
1.2	Basic concepts for structural dynamics . . . . .	3
1.3	Installation . . . . .	7
<b>2</b>	<b>Tutorial</b>	<b>8</b>
<b>3</b>	<b>Results</b>	<b>13</b>
<b>4</b>	<b>Advanced options</b>	<b>14</b>
<b>5</b>	<b>Define new trains</b>	<b>15</b>
5.1	Create new train . . . . .	15
5.2	Upload a train already created . . . . .	15
<b>6</b>	<b>Interaction between vehicle and structure models</b>	<b>16</b>
<b>7</b>	<b>Support</b>	<b>16</b>

# 1 Introduction

CALDINTAV is a program for dynamic analysis of railway bridges developed by the Computational Mechanics Group at the Civil Engineering School (Technical University of Madrid)

## 1.1 Scope

The objective of this program is to provide the user a fast and simple tool to analyse dynamic effects not only on simply supported bridges. Following versions will also cover continuous bridges.

The main measures of dynamic performance for the design of the bridge may be obtained by CALDINTAV. For instance, dynamic impact factor, acceleration and displacement on the deck of the bridge... etc.

A database with real trains of European network is provided, as well as the HSLM defined in Eurocodes. Additionally, the user can define additional trains for performing the calculations.

Real trains	Max. running speed (km/h)
AVE-S103-SIEMENS	420
AVE-S112-TALGO	420
AVE-S100R-ALSTOM	420

Table 1: Real trains provided with the program and maximum running speed

## 1.2 Basic concepts for structural dynamics

The actions to which a railway bridge is subjected are dynamic. Their effects can be higher than those due to static actions. This is due to:

- The mobile nature of the loads, which produces increases or decreases in the equivalent static load
- Repeated application of the loads which can lead to resonance.
- The irregularities of the track and wheels.

To evaluate the dynamic effects, all trains which could run on the line should be considered, and all possible running speeds.

For each structural element, the maximum dynamic stress envelope is defined by the impact factor, regardless the dynamic calculation procedure chosen. The dynamic impact factor is defined as:

$$\Phi = \frac{\max S_{din,max}}{S_{est,max}}$$

Where  $\max S_{din,real}$  is the maximum dynamic solicitation due to all possible trains and running speeds, and  $S_{est,tip0}$  is the maximum static solicitation due to the train.

For simply supported bridges CALDINTAV obtains this impact factor as the quotient between the maximum displacements at centre of span:

$$\Phi = \frac{\max \delta_{din,max}}{\delta_{est,max}}$$

Generally, the available dynamic analysis methods are:

1. Methods based on direct integration
2. Methods based on modal decomposition
3. Simplified methods for simply supported bridges based on train signature.

CALDINTAV uses the second type. For simply supported bridges, only symmetric vibration modes are considered, as the antisymmetric modes are zero at the centre of span (see figure 1). As default, the fundamental mode is considered. The vibration modes of a simply supported bridge could be described as follow:

$$\phi_i(x) = \sin\left(\frac{i\pi x}{L}\right)$$

Where  $i$  is the considered mode,  $L$  is the span length and  $x$  is the bridge local coordinate.

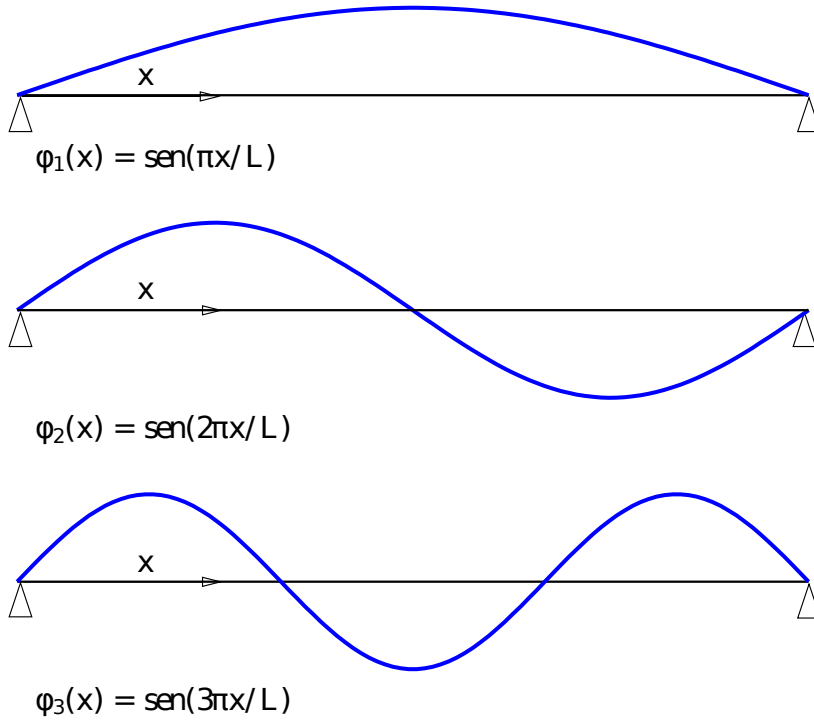


Figure 1: Three first vibration modes of a simply supported beam

## Types of models

### 1. Moving load models

Moving load models do not gather the vibration produced on the vehicle. Considering the vehicle as infinitely rigid, the load per axle have a constant value, equal to that with the train stopped.

Therefore, this model consists of a series of loads with a fixed value, spaced apart a constant distance, that are moving above the structure.

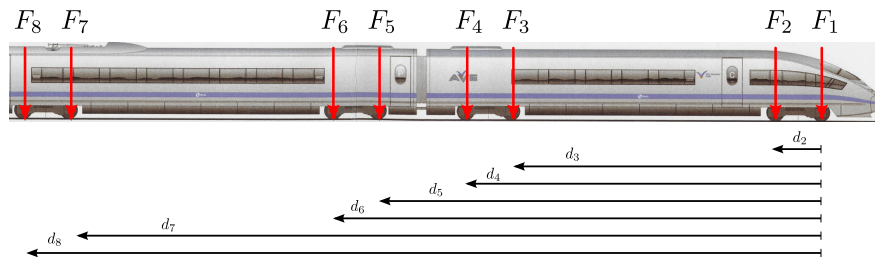


Figure 2: Moving loads schema. Relative distance to the first axle is shown.

This models gives a very accurate and, in general, conservative solution. Additionally, from the point of view of the bridge designer, vehicle properties that are necessary for interaction model (stiffness, mass of the suspension...etc.) are not generally available.

By default, CALDINTAV uses a moving load model.

## 2. Vehicle-bridge dynamic interaction model

The action transmitted to the rail from the wheels is not constant, it is variable according to the suspension of the train. Interaction models take into account the existence of mechanisms that are dissipating or exchanging energy.

In general, these models imply a reduction of the dynamic effects. Thus, in non resonant situations or continuous bridges are not decisive in the calculation, recommending the use of moving loads models.

On the other hand, these models have two main advantages respect to the moving loads ones: allow to know the lateral vehicle-structure phenomena in a more realistic way, and allow to know the movement of the vehicle, which facilitates the analysis of aspects such as the comfort of the passengers or the train safety.

CALDINTAV uses a simplified interaction model, in which the following things are considered:

- Primary suspension, with its values of stiffness and damping per axle.  $(K_p, C_p)$
- Unsprung mass, corresponding to the nominal mass of the axle and the proportional part of the sprung mass.
- Sprung mass, which in this case, is equivalent to the proportional part of the bogie's mass

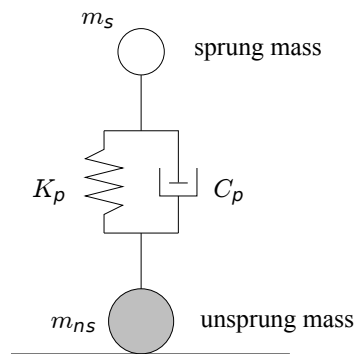


Figure 3: Simplified model of interaction between vehicle and structure.

### Load distribution between sleepers.

Considering load distribution means that the vertical load of each axle of the train, which is acting on the rail, is divided among three consecutive sleepers with a ratio of  $\frac{1}{4}$ ,  $\frac{1}{2}$  y  $\frac{1}{4}$  (Figure 4).

It is known that considering the load as concentrated, without distribution between the sleepers, causes higher accelerations in the deck of the bridge than if the load is distributed. Therefore, not considering load distribution, is the most conservative assumption, and consequently is more often used in dynamic analysis.

CALDINTAV allow to choose between considering load distribution or not.

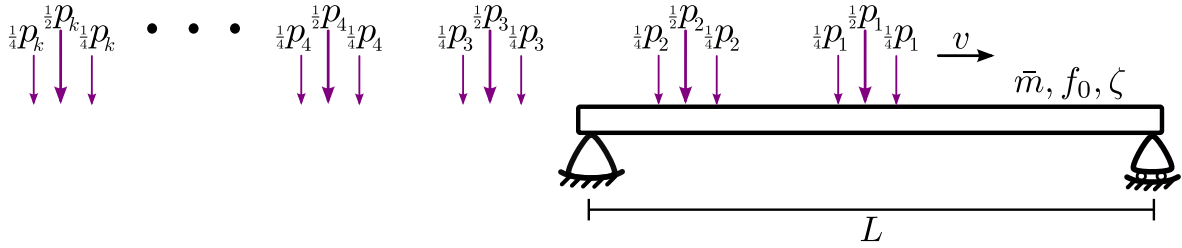


Figure 4: Load distribution schema

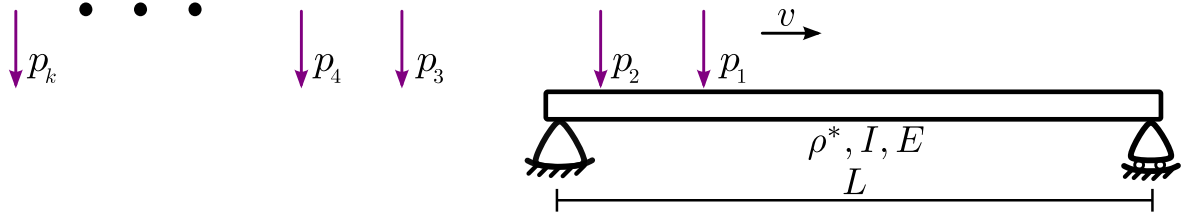


Figure 5: Moving and concentrated load schema

### 1.3 Installation

CALDINTAV is a tool developed in MATLAB. For running is not necessary to have installed MATLAB, only a MATLAB set of libraries that are freely distributed, called MATLAB Compiler Runtime (MCR). These libraries are incorporated in the provided package.

For installation on WINDOWS is necessary to follow the next steps:

1. Extract CALDINTAV.zip file
2. Inside the extracted file, execute MCRInstaller.exe. Follow the steps and complete the installation of the MCR library.

Once done, to run the program, double click on the file called Caldintav.exe. Depending on your computer this can last few minutes, please wait.

*NOTE: In case that when running the file CALDINTAV.exe the next message error appears: 'Could not find version 7.17 of the MCR. Attempting to load mcl ...' we should repeat the installation of the MCR module (running again MCRInstaller.exe). By repeating this operation the computer will solve its problem to detect the path of this file and we will be able to run the program correctly.*

## 2 Tutorial

We are going to introduce the use of the program by a simple example. A simply supported bridge with the following characteristic is considered:

- $L = 15$  m
- $f_1 = 7$  Hz
- $\rho = 12000$  kg/m
- $\zeta = 1.85$  %

Taking into account the value of the span length, mass and frequency, the bending stiffness of the bridge is:

$$EI = 12064000 \text{ kNm}^2$$

Lets calculate the dynamic response of a bridge under train ICE-3 (AVE S-103) for different running speeds, varying from 100 to 400 km/h. The steps are:

### 1. Running the program and name the project

If we are working in WINDOWS, double click on the icon CALDINTAV. It will start the interface, and then click on **START**.

Then, we are asked to introduce a project name. We type a name, which could contain spaces, but should not contain symbols or accents, as our output files are going to be saved with this name. We write, for example, 'Puente ejemplo' and then click on **Accept**.

### 2. Bridge properties definition.

We will define the bridge properties in the corresponding section of the main window (see figure 6) . We can choose between introduce the natural frequency of the bridge or the bending stiffness by clicking on the pop-up menu. The mass that we should introduce is the total mass per unit of length of the bridge (including both structural and non-structural mass).



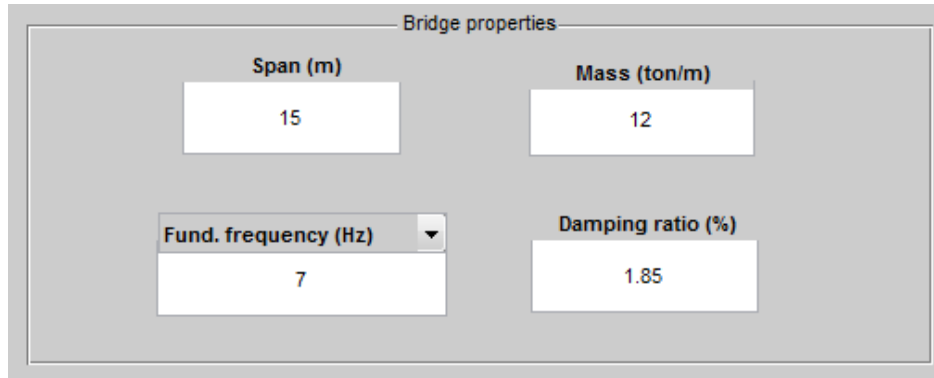


Figure 6: Defining bridge properties

### 3. Choosing trains and speeds.

In this case, we want to perform our analysis with one of the trains included in the program library, the AVE-S103-SIEMENS. Thus, we only have to select it from the list of available trains (*Available trains*) and pass it through the button **Add** to the list of trains that we are going to use for the analysis (*Selection*) (see figure 7).

This program allows the analysis with several trains at the same time, simply by including them in analogous manner as described. It also allows to define new trains or load the ones that we have already defined (see section 5).

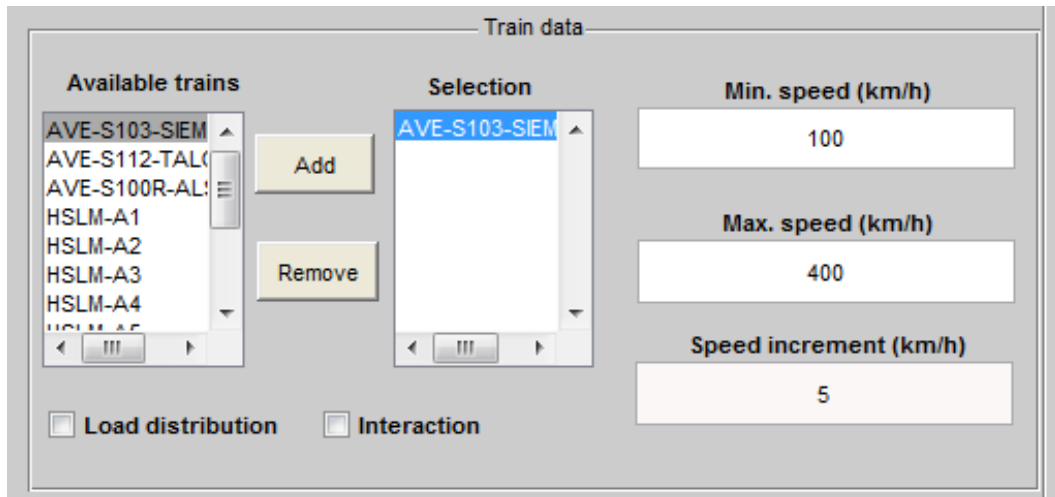


Figure 7: Select trains and running speeds

We write the minimum running speed, in our case 100 km/h, and the maximum running speed, 400 km/h. By default, the program carries out speed increments of 5 km/h. We can modify this increment if desired, and therefore the different speeds at which calculation is going to be made.

#### 4. Choosing type of analysis

If we want to consider load distribution we should check the box **Load distribution**. In this case, we are not going to consider load distribution so we are leaving this box without check.

Interaction between vehicle and structure can be also consider if desired.(See figure 15).

#### 5. Results

To perform the calculation we click on the button **Submit** of the main window. A window that inform us about the progress of the calculation will appear. At any time we can cancel the analysis we have sent by clicking on button **Cancel** of this window.

When the calculation is finished, two graphs will appear on the right of the main window, in the panel of results (*Results*). The graph on the top shows the maximum accelerations for each calculated speed. The lower graph shows the maximum dynamic impact factor for each running speed. Below this, maximum value of the acceleration and impact factor is shown, as well as the speed at which occurs. If you have carried out the analysis using several trains, there it will appear the value of the train which involves higher dynamic effects.

To facilitate the interpretation of the results, some buttons under each graph are provided. This buttons allow to plot different limits. In this case, we will plot the maximum limit of acceleration, defined in the IAPF-2007, by selecting the type of track, for example ballast track (**Ballast**) and then clicking on the button **Máx.Accel.Limit**.

In the figure 8 maximum limit of accelerations is shown for a ballast track. It can be seen that this limit is overpassed at 210 km/h, and also for speeds higher than 310 km/h.

Analogously for the impact factor graph it is available a limit which gives an idea of when significant dynamic phenomena are appearing. This limit is defined by the UIC-776 and we can plot it by clicking in the button  $1 + \varphi'_{\text{UIC}}$ .

In the figure 9 it can be seen that there are certain points in which value  $1 + \varphi'_{\text{UIC}}$  is overpassed, that approximately coincide with the ones that produce higher values of the acceleration.

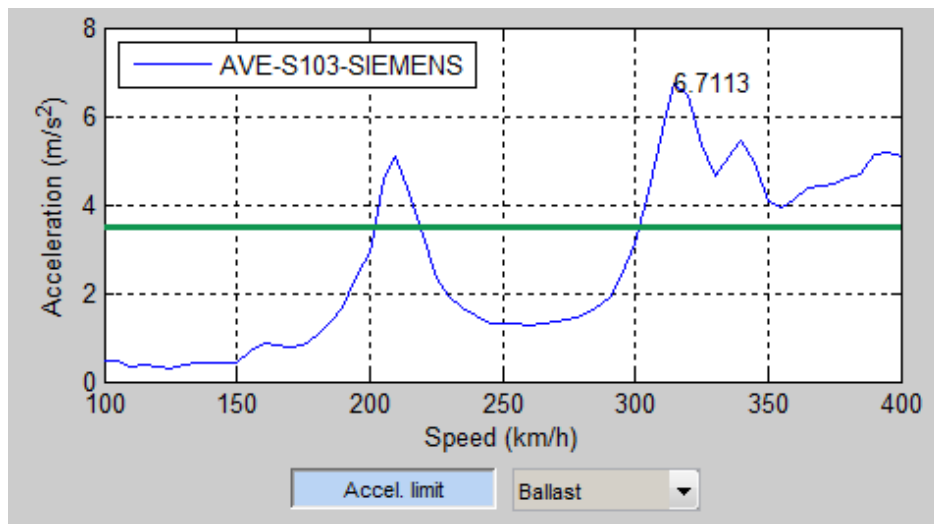


Figure 8: Maximum accelerations vs speed and limit for ballast track.

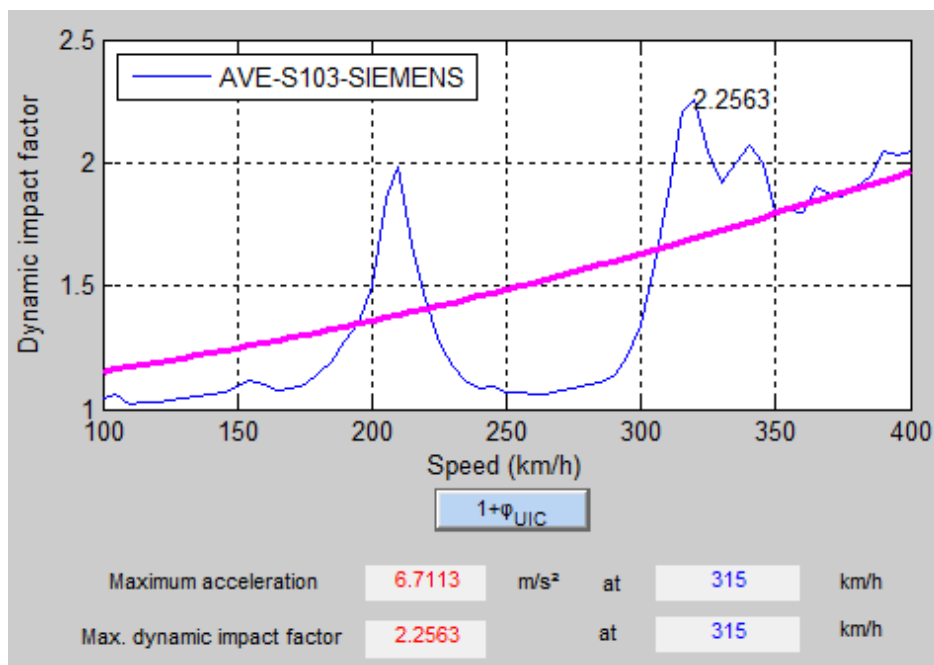


Figure 9: Maximum dynamic impact factor vs speed and limit defined by the UIC-776

To see the graphics again without the limits plotted, click on the *regenerate* button (figure 10), located in the upper right corner.



Figure 10: Regenerate button

In order to understand what is happening at 210 km/h, we can obtain the history of acceleration at this point. For this, we shall go to the menu bar: **Results** → **History of accelerations** or with the keyboard shortcut **CTRL+T**.

In order to load the trains that we have already used in the calculation, click **Load trains**. This way, trains are loaded in the pop-up menu. We choose the train, in this case AVE-S103/SIEMENS. Enter the speed at which you want to see the history of acceleration, for instance 210 km/h, and push *intro* in the same field in which we have edited the speed.

In figure 11 , for 210 km/h, a resonant pattern is shown. This causes a peak on the acceleration vs speed graph and in the impact factor vs speed graph that we have already detected. At 150 km/h (figure 12) this pattern does not appear.

Analogously we can obtain the history of displacement at a certain speed **Results** → **History of displacements** or with the keyboard shortcut **CTRL+D**.

It is possible to clean all the fields and graphs for performing a new analysis by going to the menu bar **Edit** and click on the option **Clear**, or with the keyboard shortcut **CTRL+C**.

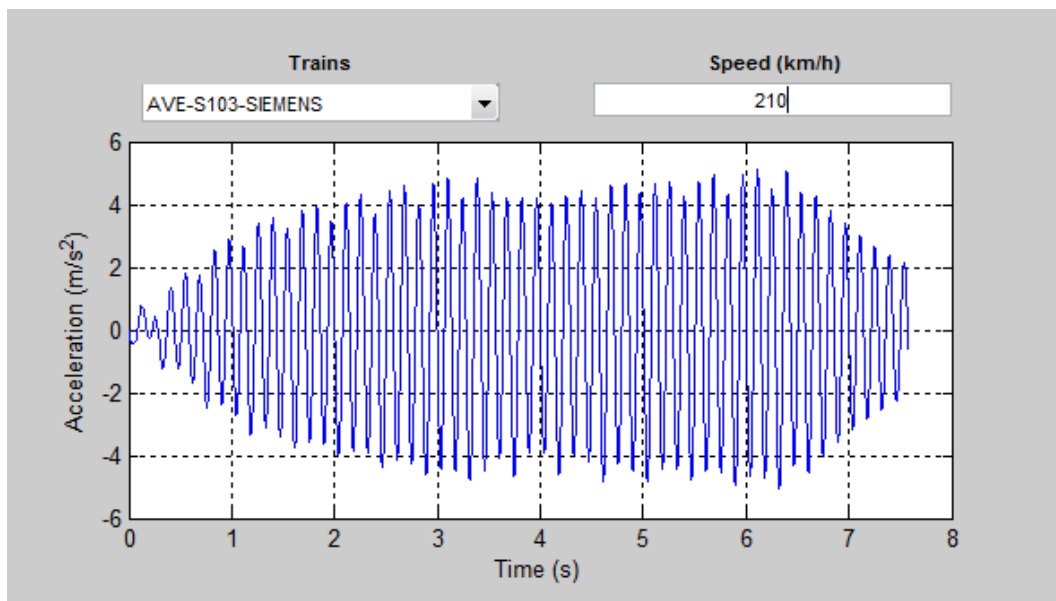


Figure 11: History of accelerations at 210 km/h

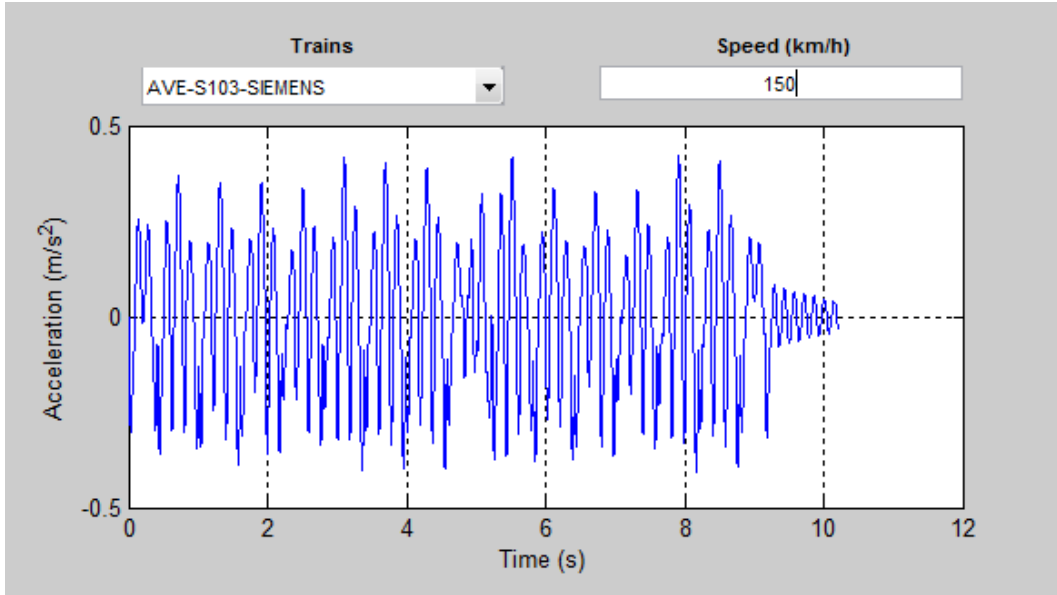


Figure 12: History of accelerations at a 150 km/h

### 3 Results

Apart from the results that we can see on the screen and we have described in the previous section, the program generates some output files for each speed and each train with the following name:

*Project\_name-Train-ml/inte-Lspan\_length-zdamping\_ratio-ffrequency-mmass-nmnumber\_of\_modes-vs speed.dat*

These files are saved in a folder with the name of the project (created by the program), located in the folder "results".

In our case, as we have carried out a moving load analysis (*ml*) and have not considered interaction between vehicle and structure (*inte*), a result file would be, for example:

Puente\_ejemplo-AVE-S103-SIEMENS-ml-L15-z1.8-f7.0-m12t-nm1-v100.dat

These are text files, in which we have four columns that contain the following data:

Time	Displacement	Speed	Acceleration	Dynamic Impact Factor
------	--------------	-------	--------------	-----------------------

Table 2: Meaning of the values gathered in columns, provided by the output files for each speed.

It also generates files containing the maximum values for each speed, that is to say, an envelope of the response. These files have the following name:

*Project\_name-Train-ml/inte-envelope-Lspan\_length-zdamping\_ratio-ffrequency-mmass-nmnumber\_of\_modes-vinitial\_speed-final\_speed.dat*

In our case the generated file have the following form:

Puente\_ejemplo-AVE-S103-SIEMENS-ml-envelope-L15-z1.8-f7.0-m12t-nm1-v100-400.dat

This text files contain columns with the following data:

Train Speed	Displacement	Speed	Acceleration	Dynamic impact factor
-------------	--------------	-------	--------------	-----------------------

Table 3: Meaning of the values gathered in columns provided by the envelope result files.

## 4 Advanced options

The user can specify the number of modes and the time step to be considered in the analysis. The time step indicates at which times is the problem going to be solved. Thus, the smaller the time step the more accurate the solution will be, but also the more time the calculation will take.

For simply supported bridges the program only considers symmetric modes of vibration, as the mode shape in the centre of span for anti-symmetric modes vanishes. By default the number of modes considered is 1. If we want to modify them, we shall go to **Options** → **Advanced**.

In figure 13 we have changed the number of considered modes to 3 and we have accepted the default time step (1e-3). To confirm we click on **Accept**.

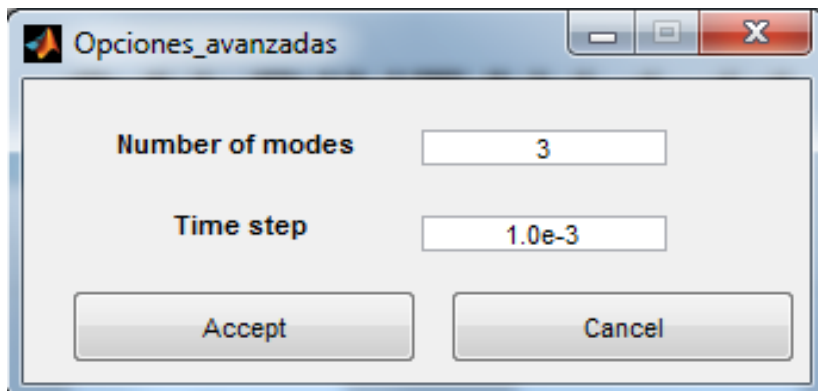


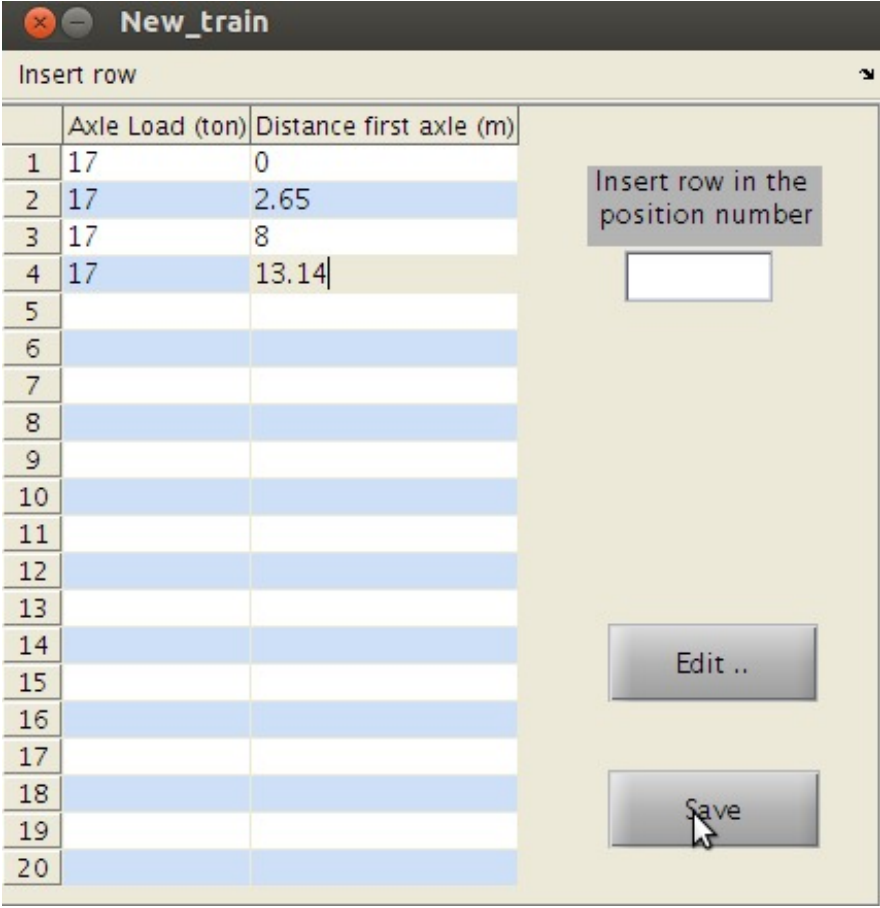
Figure 13: Modifying number of modes and/or time step

## 5 Define new trains

### 5.1 Create new train

To create a new train we click on **Options** → **Train** → **New**.

A new window will open, in which we can define the value of the load per axle in tons (**Axle Load (ton)**) and the distance from that axis to the first one. The distance from the first axis to itself is always zero (see 14). If our train has more than 20 axes we can increase the number of rows writing the desired number on the field **Number of rows** and push enter button. Finally we must specify the name we want to give to our new train in the field **Name** and click on **Save**. This way, a data file is created in the folder *trains*, in which we kept this train saved. (This files can be also modified by using any text editor).



The screenshot shows a window titled "New\_train" with a table and several buttons. The table has two columns: "Axle Load (ton)" and "Distance first axle (m)". The rows are numbered 1 to 20. The first four rows have data: (1, 17, 0), (2, 17, 2.65), (3, 17, 8), and (4, 17, 13.14). The rest of the rows are empty. To the right of the table, there is a text input field with the label "Insert row in the position number" above it. Below the input field are two buttons: "Edit .." and "Save". A mouse cursor is pointing at the "Save" button.

	Axle Load (ton)	Distance first axle (m)
1	17	0
2	17	2.65
3	17	8
4	17	13.14
5		
6		
7		
8		
9		
10		
11		
12		
13		
14		
15		
16		
17		
18		
19		
20		

Figure 14: Create new train

### 5.2 Upload a train already created

For achieving that an already created train appears in the list of available trains for the calculation, and therefore, we can use it in the analysis, we should go to **Options**

→ **Train** → **Load**. We must enter in the folder *trains* and load the file of our train (with extension *.dat*) by clicking on **Open**. The train will appear the last in the list of available trains and we can select it for the analysis by clicking on **Add**.

## 6 Interaction between vehicle and structure models

To perform a calculation considering the interaction between vehicle and structure it is necessary to have the data of the suspension of the train and define them by going to **Options** → **Interaction model**.

We should introduce the value of the sprung mass (corresponding to half bogie) in kg, the value of the stiffness of the suspension (corresponding to two axle-boxes) in N/m and, finally, the value of the suspension damping in Ns/m (see figure 15).

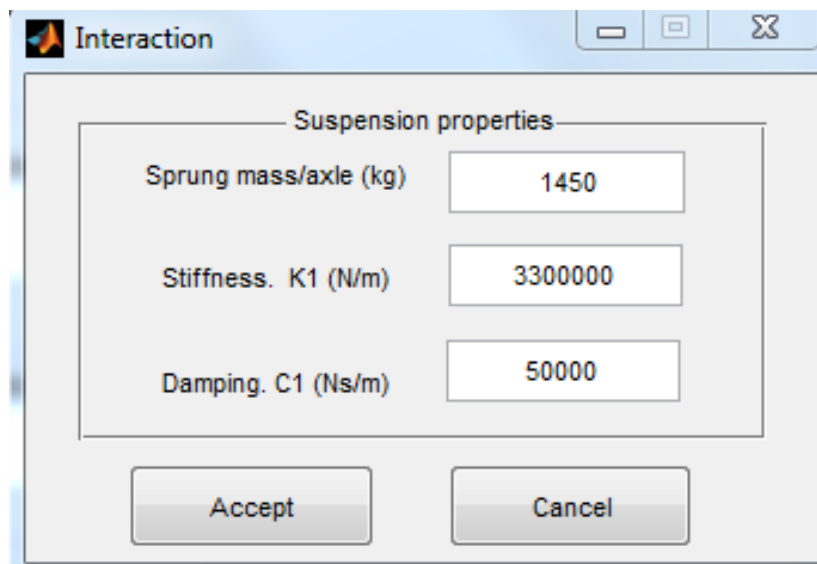


Figure 15: Definition of the properties of the suspension for the use of interaction models.

Finally, whenever we want to perform a calculation taking into account the interaction, it is necessary to activate the box **Interaction** that is located at the bottom left of the main window.

## 7 Support

For any questions you can contact us by email:

*caldintav@mecanica.upm.es*